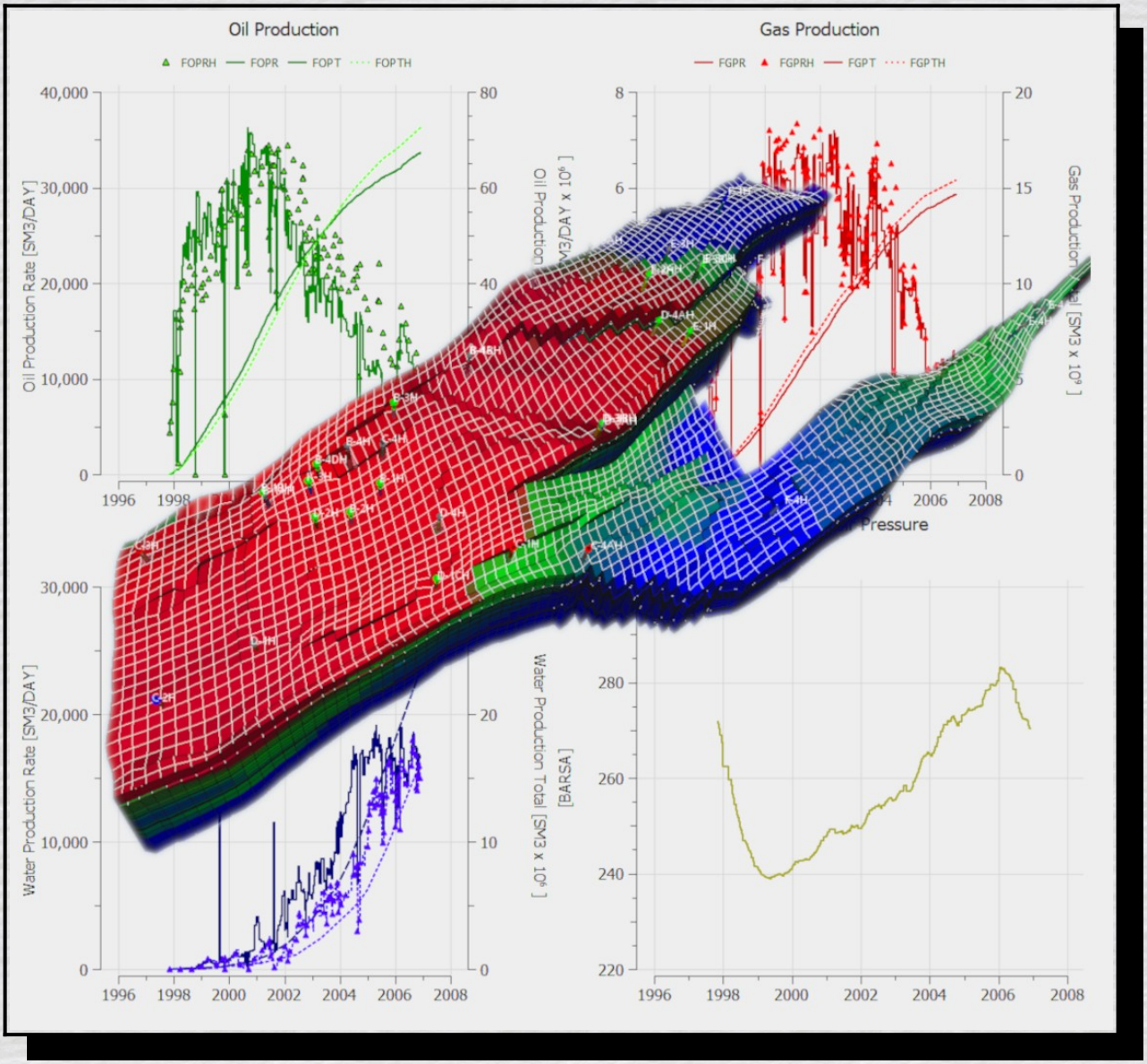
OPEN POROUS MEDIA

OPM Flow Reference Manual



OPM FLOW VERSION: 2023-10

MANUAL REVISION: Rev-0

OPEN POROUS MEDIA
OPM Flow Reference Manual
(2023-10)

DOCUMENT CONTROL	
Company	**Equinor ASA**
Confidentiality, copyright and reproduction	 **Documentation Contributors and Developers:** Jostein Alvestad, Kai Bao, David Baxendale, Markus Blatt, Joshua Charles Bowden, Paul Egberts, Joakim Hove, Håkon Hægland, Matthew Goodfield, Vegard Kippe, Peter Kirkham, Stein Krogstad, Arne Morten Kvarving, Kjetil Olsen Lye, David Landa-Marbán, Andreas Lauser, Cintia Goncalves Machado, Halvor Møll Nilsen, Atgeirr Flø Rasmussen, Antonella Ritorto, Alf Birger Rustad, Tor Harald Sandve, Ove Sævareid, Bård Skaflestad, Torbjørn Skille, Pieter J.Verveer
Document Type	OPEN POROUS MEDIA
Document Title	OPM Flow Reference Manual (2023-10)
Document Version	Rev-0
Document Ref.	2023-10-RPT
Author and Editor	OPM-OP AS Oscars Gate 27, 0352 Oslo, Norway
	since 2023: Matthew Goodfield, 2017-2023 David Baxendale
	Please report problems, errors, and suggestions on *github.*
Document Name	OPM_Flow_Reference_Manual_2023-10_Rev-0.fodt

OPEN POROUS MEDIA

OPM Flow Reference Manual

(2023-10)

Document Control			
Document Revision	Date	Version and Revision	Status
	March 15, 2024	2023-10 Rev-0	Final
	February 15, 2024	2023-10 Rev-A	Developer's Review Version
	June 8, 2023	2023-04 Rev-0	Final
	May 19, 2023	2023-04 Rev-D	Developer's Review Draft
	May 8, 2023	2023-04 Rev-C	Developer's Release Notes Review Draft
	January 31, 2023	2023-04 Rev-A/B	Working Drafts
	December 14, 2022	2022-10 Rev-0	Final
	November 18, 2022	2022-10 Rev-A	Developer's Review Draft
	May 18, 2022	2022-04 Rev-0	Final
	May 6, 2022	2022-04 Rev-A	Developer's Review Draft
	November 17, 2021	2021-10 Rev-0	Final
	October 22, 2021	2021-10 Rev-A	Developer's Review Draft
	June 28, 2021	2021-04 Rev-1	Final (Fixed Keyword Footer)
	June 17, 2021	2021-04 Rev-0	Final
	June 7, 2021	2021-04 Rev-A	Developer's Review Draft
	December 23, 2020	2020-10 Rev-0	Final
	December 10, 2020	2020-10 Rev-A	Developer's Review Draft
	July 9, 2020	2020-04 Rev-1	Final
	June 25, 2020	2020-04 Rev-B	Developer's Review Draft for Appendix D
	May 20, 2020	2020-04 Rev-0	Final
	May 8, 2020	2020-04 Rev-A	Developer's Review Draft
	December 5, 2019	2019-10 Rev-0	Final
	November 6, 2019	2019-10 Rev-0	Final Draft
	June 20, 2019	2019-04 Rev-0	Final

DOCUMENT CONTROL			
	December 31, 2018	2018-10 Rev-2	Fix Broken Links etc.
	November 6, 2018	2018-10 Rev-1	Final (DRSDTR & DRVDTR)
	November 5, 2018	2018-10 Rev-0	Final
	October 4, 2017	2017-04 Rev-0	Final

TABLE OF CONTENTS

INDEX OF TABLES

INDEX OF FIGURES

CHAPTER 1: INTRODUCTION

1.1 OVERVIEW

The Open Porous Media ("OPM") initiative was started in 2009 to encourage open innovation and reproducible research on modeling and simulation of porous media processes. OPM was initially founded as a collaboration between groups at Equinor (formerly Statoil), SINTEF, the University of Stuttgart, the University of Bergen, but over time, several other groups and individuals have joined and contributed. What today forms the OPM suite of software, has mainly been developed by SINTEF, NORCE (formerly IRIS), Equinor, Ceetron Solutions (OPM Resinsight), Poware Software Solutions, Dr. Blatt HPC-Simulation-Software & Services, and OPM-OP. The initial vision was to create long-lasting, efficient, and well-maintained, open-source software for simulating flow and transport in porous media. The scope has later been extended to also provide open data sets, thus making it easier to benchmark, compare, and test different mathematical models, computational methods, and software implementations.

All software in OPM is distributed under the GNU General Public License ("GPL"), *version 3*, with the option of using newer versions of the license. Datasets are distributed under the Open Data Commons Open Database License ("ODbL"), *version 1.0*, with individual content under the Database Contents License ("DbCL"), *version 1.0*

The project's numerical simulator, OPM Flow, is a fully-implicit, black-oil simulator capable of running industry-standard simulation models[1]. The simulator's input file is compatible with the commercial simulator, although OPM Flow has additional keywords for supporting additional functionality not found in the commercial simulator. The simulator's output files, except for the print and debug files, are compatible with the commercial simulator, and are readable by commercial post-processing software as well as by OPM ResInsight. OPM ResInsight is the Open Porous Media's post-processing software.

The simulator is implemented using automatic differentiation to enable rapid development of new fluid models. In most simulators the coefficients of the linearized systems of equations (Jacobian matrix) required by the non-linear Newton Solver, have traditionally been computed by evaluating the closed-form expressions obtained by differentiating the discretized flow equations analytically. Differentiating these flow equations manually and programming the resulting formulas is time-consuming and error-prone. Thus, even simple extensions in functional dependencies can be difficult to accommodate into existing simulators. Automatic differentiation eliminates these drawbacks.

OPM Flow Runs under Linux and Windows Subsystem for Linux, and the program is in active development with new features added in each bi-annual release. Currently the program has the following functionality:

Model Formulation

- Black-oil with dissolved gas and vaporized oil.
- Rock-dependent capillary and relative-permeability curves.
- End-point scaling and hysteresis.
- Oil vaporization controls (VAPPARS).
- Rock compaction.

Enhance Oil Recovery Models

Enhanced Oil Recovery ("EOR") is a term applied to methods used for recovering oil from a petroleum reservoir beyond that recoverable by primary and secondary methods. Primary recovery normally refers to production using the energy inherent in the reservoir from gas under pressure or a natural water drive. Secondary recovery usually refers to injection of water or water flooding. Thus, Enhanced Oil Recovery is often synonymous with tertiary recovery. Improved Oil Recovery ("IOR") and Advanced Oil Recovery ("AOR") have similar meaning, except they also apply to primary and secondary methods, and sometimes

[1] *Rasmussen, A.F., Sandve, T.H., Bao, K., Lauser, A., Hove, J., Skaflestad, B., Klöfkorn, R., Blatt, M., Rustad, A.B., Sævareid, O. et al. [2019]. The Open Porous Media Flow Reservoir Simulator. arXiv preprint arXiv:1910.06059.*

EOR methods can be used earlier in the sequence. Originally water flooding was considered as Enhanced Oil Recovery, but now EOR is generally considered to follow water flooding.

There are four major EOR groups of methods:

- thermal recovery,
- gas miscible recovery,
- chemical flooding, and
- microbial flooding.

These main groups are further subdivided into individual processes as illustrated in Figure 1.1. For example the thermal recovery methods consist of steam flooding and cyclic steam stimulation, as well as in-situ combustion. Figure 1.1 also includes hot water as a process for thermal EOR, but this process is rarely used in the industry as a drive mechanism, as steam contains significantly more heat than hot water and therefore steam is more effective in heating up the in situ oil. **Note that microbial EOR methods, are not shown in Figure 1.1, due to there limited use in the industry.**

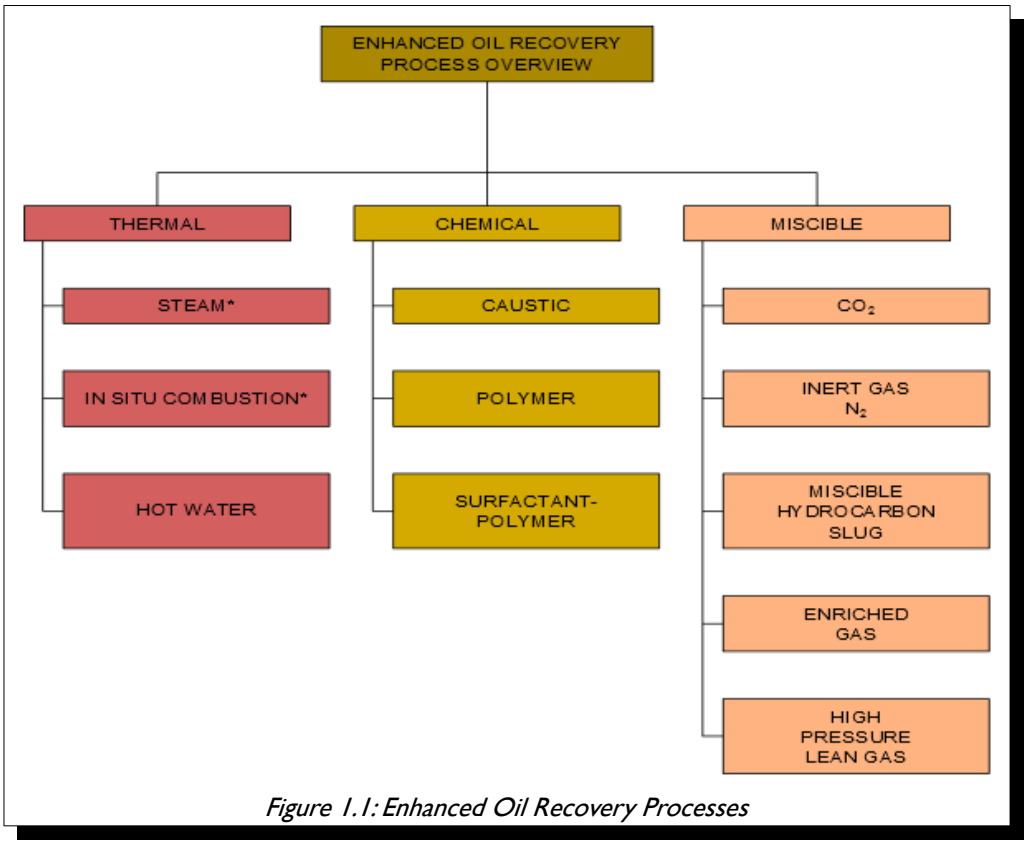


Figure 1.1: Enhanced Oil Recovery Processes

Chemical flooding methods include polymer flooding (including polymer gels), micellar-polymer flooding, and alkaline flooding. Quite often these type of floods are combined to form a more efficient displacement process, for example in an Alkaline Surfactant Polymer ("ASP") process.

Conventional gas flooding is an immiscible process, that is the oil and gas phases remain separate during the displacement drive. For the EOR gas drive process, the displacement is miscible, here the gas "mixes" with the oil to form a single phase fluid in the reservoir. Gas miscible processes are subdivided into carbon dioxide flooding, cyclic carbon dioxide stimulation, nitrogen flooding and nitrogen-CO2 flooding.

OPM Flow has several EOR models based on extending the black-oil model with a fourth component in the simulator.

- CO_2 Standard EOR Model: OPM Flow's CO_2 Standard EOR model is based on on the simulators Solvent model in which uses an extra component by extending the black-oil oil formulation with a fourth component in the simulator by adding a CO2 component to the gas phase. This is the standard formulation used by most black-oil simulators.

- CO_2 Dynamic EOR Model: Although the CO2 Standard EOR model is commonly used, the formulation cannot account for the mass transfer of the various components and phases. The extended black-oil formulations often poorly represent the PVT properties of the oil-CO2 mixtures, resulting in poor agreement with the compositional formulation. OPM Flow's CO2 Dynamic EOR Model black-oil formulation attempts to overcome this limitation by making the black-oil properties dependent on the fraction of CO2 in the cell[2]. This approach models the oil-CO2 mixture more accurately, and thus give results closer to the compositional simulator. Note the model is not restricted to CO2 utilization, as long as the appropriate oil-injection-gas dependent properties are entered in the model.

- Foam Model: An experimental foam module has been added to OPM Flow 2019-10 release. With this it is possible to simulate certain types of surfactant injection. Such injection stimulates formation of foam to change mobility ratios, and give better reservoir sweep. The implemented foam model treats surfactant transported in the gas or water phase, and reduces the mobility of that phase depending on the surfactant concentration. In addition to mobility reduction, foam adsorption to the reservoir rock is also included in the model.

- Polymer Model (Standard): OPM Flow's polymer model is based on a black-oil polymer formulation, which is developed by extending the black-oil model with a polymer component. The effects of the polymer mixing are simulated based on the Todd-Longstaff[3] mixing model, and adsorption, dead pore space, and permeability reduction effects are also considered. A logarithmic shear thinning/thickening model has also been incorporated since the 2015-10 Release (see *Flow-polymer*). Note that the Polymer model has now been incorporated into the main OPM Flow simulator and is no longer a separate simulator.

- Polymer Model (Molecular Weight Transport): In addition to the standard polymer model, the simulator also has the Polymer Molecular Weight Transport option, that uses the polymer molecular weight in calculating the polymer viscosity. Here, the *Mark-Howink equation*, also known as the Mark-Howink-Staudinger equation[4], [5], and [6] is used to calculate the intrinsic viscosity as a function of the molecular weight of the polymer.

- Solvent Model: Similar to the polymer model, OPM Flow's solvent model again uses an extra component by extending the black-oil oil formulation with a fourth component in the simulator by adding a solvent component to the gas phase (see *Flow-solvent*). Note that Solvent model has now been incorporated into the main OPM Flow simulator and is no longer a separate simulator.

CO2 Storage:

- Brine-CO2 PVT module that generates brine and CO2 PVT properties directly eliminating user input for PVT.

[2] *T. H. Sandve, O. Sævareid and I. Aavatsmark: "Improved Extended Blackoil Formulation -- for CO2 EOR Simulations." in ECMOR XVII – The 17th European Conference on the -- Mathematics of Oil Recovery, September 2020.*

[3] *M. R. Todd and W. J Longstaff, The Development, Testing, and Application Of a Numerical Simulator for Predicting Miscible Flood Performance". In: J. Petrol. Tech. 24.7 (1972), pages 874-882.*

[4] *H. Mark, in R. Saenger, Der feste Koerper, Hirzel, Leipzig, 1938.*

[5] *R. Houwink , J. Prakt. Chem., Vol. 157, Issue 1-3, p. 15 (1940).*

[6] *H. Staudinger, Die Hochmolekulare Organischen Verbindungen, Julius Springer, Berlin 1932.*

- Thermal properties generated by OPM Flow include the CO_2 enthalpy represented as a table internally in the simulator. The liquid enthalpy depends on the dissolved CO2 and salinity as well as pressure and temperature. The water enthalpy is automatically calculated and modified to account for salinity and for CO2 following.

- Diffusion for fine grid modeling is implemented via diffusion coefficients that depend on temperature, pressure, and salinity. The diffusion coefficient is computed internally for pure water and modified to account for salinity. The effect of the porous media on the diffusion is modeled using an industry standard relationship. In addition, the coefficient can also be given as an input parameter using a keyword.

- CO_2 convective dissolution of carbon dioxide (CO2) into in situ brine within a grid cell is implemented together with a convective dissolution parameter for controlling the rate of dissolution.

- Microbially Induced Calcite Precipitation ("MICP") model for the modeling biochemical processes in creating barriers by calcium carbonate cementation, the technology has the potential to be used for sealing leakage zones in geological formations[7] and [8].

Other OPM Flow Specific Enhancements:

- The Salt Precipitation and Water Evaporation Model: The model accounts for water evaporation of the in situ brine and the resulting increasing dissolved salt concentration. Once the salt solubility limit is exceeded, salt precipitation occurs, that results in a decrease in porosity and permeability, and consequently a decrease in gas production. This feature is primary used in modeling gas fields under depletion.

- Thermal Model Enhancements: The energy black-oil implementation in OPM Flow is a mixture of the commercial simulators black-oil temperature and compositional thermal options. These commercial simulator options are two separate modeling facilities in the commercial simulator. OPM Flow's thermal implementation is based on solving the energy equation fully coupled with the black-oil equations, so the results are not directly equivalent to the commercial simulator's black-oil temperature or compositional thermal formulations. In addition, OPM Flow can take into account the Joule–Thomson effect for oil, gas and water. This behavior is especially important in gas wells, when a real gas expands, resulting in the temperature of the gas dropping.

Description of Geology:

- Rectilinear and fully-unstructured grid.

- Corner-point grids from the commercial simulator input, including fault and region multipliers, minimum pore volume, pinch outs, etc.

- Spider grids that are similar to radial grids for modeling near well bore effects that enable visualization in OPM ResInsight.

- Analytical and numerical aquifers.

Well and Group Controls:

- Bottom-hole pressure and surface/reservoir rate.

[7] *Landa-Marbán, D., Tveit, S., Kumar, K., Gasda, S.E., 2021. Practical approaches to study microbially induced calcite precipitation at the eld scale. Int. J. Greenh. Gas Control 106, 103256. https://doi.org/10.1016/j.ijggc.2021.103256.*

[8] *Landa-Marbán, D., Kumar, K., Tveit, S., Gasda, S.E., 2021. Numerical studies of CO2 leakage remediation by micp-based plugging technology. In: Røkke, N.A. and Knuutila, H.K. (Eds) Short Papers from the 11th International Trondheim CCS conference, ISBN: 978-82-536-1714-5, 284-290.*

- Group controls, including voidage replacement, network model, gas lift optimization, guide rate control for producers and injection groups and well, etc.
- Shut/stop/open individual completions.
- History-matching wells.
- Multi-segment wells together with various inflow control devices ("ICDs")
- Well lists.
- Action conditions and command processing via the ACTIONX and UDQ keywords employed in the commercial simulator.
- Python interface to control and process simulator data.

Input and Output:

- General reader/parser for Eclipse input decks.
- XML-based or simple text-format input of additional parameters.
- Flexible output of summary and restart files in he commercial simulator's format.
- Logging to terminal and print file.
- Front-end graphical interface for running the simulator with various tools, including: case compression and uncompression, keyword generation etc.

Simulation Technology:

- Fully-implicit in time.
- Automatic differentiation
- Two-point flux approximation in space with upstream-mobility weighting.
- Flexible assembly through the use of automatic differentiation.
- Bi-conjugate Gradient method (BiCG-stab) and a restarted Generalized Minimal Residual ("GMRes") iterative linear solvers with ILU0 and Constraint Pressure Residual ("CPR") preconditioners.
- Adaptive step-size controls.
- Parallel processing.

The following section briefly the describe the simulator's formulation, equation, and the various numerical procedures available to solve the equations.

Note

The material in the following sections is taken from Rasmussen et al.[9], and the reader is encouraged to read the full paper to obtain a fuller understanding of the simulator's formulation and the resulting equations.

[9] *Rasmussen, A.F., Sandve, T.H., Bao, K., Lauser, A., Hove, J., Skaflestad, B., Klöfkorn, R., Blatt, M., Rustad, A.B., Sævareid, O. et al. [2019]. The Open Porous Media Flow Reservoir Simulator. arXiv preprint arXiv:1910.06059.*

1.2 BLACK-OIL MODEL EQUATIONS

The black-oil equations constitute the most widely used flow model in the simulation of hydrocarbon reservoirs. The model is based on the premise that we have three different fluid phases (aqueous, oleic, and gaseous), and three (pseudo) components: water, oil, and gas. Oil and gas are not single hydrocarbon species, but represent all hydrocarbon species that exist in liquid and vapor form at surface conditions. Mixing is allowed, in the sense that both oil and gas can be found in the oleic phase, in the gaseous phase, or in both. The amount of dissolved gas in the oleic phase and vaporized oil in the gaseous phase must be kept track of on a grid cell basis. In the following discussions the subscripts: w represents the water, o the oil, and g the gas quantities related to each of the phases or components.

The black-oil equations can be deduced from conservation of mass for each component with suitable closure relationships such as Darcy's law and initial and boundary conditions. The equations are discretized in space with an upstream finite-volume scheme using a two-point flux approximation, and in time using an implicit (backward) Euler scheme. The resulting equations are solved simultaneously in a fully implicit formulation by a Newton-type linearization with a properly preconditioned, iterative linear solver (see Rasmussen et al.[10]).

The conservation laws form a system of partial differential equations, one for each pseudo component α:

$$\frac{\partial}{\partial t}\left(\phi_{ref} A_\alpha\right) + \nabla \cdot \mathbf{u}_\alpha + q_\alpha = 0 \tag{1.1}$$

where the accumulation terms and fluxes are given by:

$$A_w = m_\phi b_w s_w \qquad \mathbf{u}_w = b_w \mathbf{v}_w \tag{1.2}$$

$$A_o = m_\phi \left(b_o s_o + r_{og} b_g s_g\right) \qquad \mathbf{u}_o = b_o \mathbf{v}_o + r_{og} b_g \mathbf{v}_g \tag{1.3}$$

$$A_g = m_\phi \left(b_g s_g + r_{go} b_o s_o\right) \qquad \mathbf{u}_g = b_g \mathbf{v}_g + r_{go} b_o \mathbf{v}_o \tag{1.4}$$

and the phase fluxes are given by Darcy's law:

$$\mathbf{v}_\alpha = -\lambda_\alpha \mathbf{K}\left(\nabla p_\alpha - \rho_\alpha \mathbf{g}\right) \tag{1.5}$$

In addition, the following closure relations should hold:

$$s_w + s_o + s_g = 1 \tag{1.6}$$

$$p_{c,ow} = p_o - p_w \tag{1.7}$$

$$p_{c,og} = p_o - p_g \tag{1.8}$$

[10] *Rasmussen, A.F., Sandve, T.H., Bao, K., Lauser, A., Hove, J., Skaflestad, B., Klöfkorn, R., Blatt, M., Rustad, A.B., Sævareid, O. et al. [2019]. The Open Porous Media Flow Reservoir Simulator. https://doi.org/10.1016/j.camwa.2020.05.014.*

where:

ϕ	=	porosity, can be a function of (oil) pressure: $\phi = m_\phi(p_o)\,\phi_{ref}$.
m_ϕ	=	pore volume multiplier as function of pressure.
ϕ_{ref}	=	reference porosity, a constant in time but varying in space.
p_α	=	pressure for phase α.
b_α	=	shrinkage/expansion factor for phase α defined as the ratio of the surface volume at standard conditions to the reservoir volume for a given amount of fluid: $b_\alpha = V_{surface;\alpha} \,/\, V_{reservoir;\alpha}$. For the oil phase, b_o is called the shrinkage factor, whereas for gas b_g is called the expansion factor. The reciprocal quantity is called the formation volume factor, and is usually denoted with a capital B: $B_\alpha = 1 \,/\, b_\alpha$. The variable is usually a function of phase pressure and composition: $b_\alpha = b_\alpha(p_\alpha, r_{go}, r_{og})$.
r_{go}	=	gas-oil ratio ("GOR"), ratio of dissolved gas to oil in the oleic phase, commonly called r_s or R_s in other literature.
r_{og}	=	oil-gas ratio or condensate-gas ratio ("CGR"), ratio of vaporized oil to gas in the gaseous phase, commonly called r_v or R_v in other literature.
s_α	=	saturation of phase α, the pore volume fraction occupied by the phase. The saturations of all phases sum to one.
$p_{c,\alpha\beta}$	=	capillary pressure between phases α and β. Typically a function of saturation: $p_{c,\alpha\beta} = p_{c,\alpha\beta}(S_\alpha)$.
$\mathbf{K}$	=	permeability of the porous medium.
$k_{r;\alpha}$	=	relative permeability for phase α modeling the reduction in effective permeability for a fluid phase in the presence of other phases. Typically a function of saturation.
μ_α	=	viscosity of phase α, typically a function of phase pressure and composition.
λ_α	=	mobility of phase α, given by $\lambda_\alpha = k_{r,\alpha} \,/\, \mu_\alpha$.
$\mathbf{v}_\alpha$	=	velocity of phase α.
$\mathbf{u}_\alpha$	=	velocity of component α.
$\rho_{S,\alpha}$	=	surface density of phase α at one atmosphere, a given constant.
ρ_α	=	density of phase α in the reservoir. For water, $\rho_w = b_w \rho_{S,w}$. For the oleic phase the relationship is more complex since it must include dissolved gas: $\rho_o = b_o(\rho_{S,o} + r_{go}\rho_{S,g})$, and similar for the gaseous phase we have: $\rho_g = b_g(\rho_{S,g} + r_{og}\rho_{S,o})$.
$\mathbf{g}$	=	gravitational acceleration vector.
q_α	=	well out flux density of pseudo component α, (negative for well in-flow). The form of this term depends on the well model used, either the standard well model or the multi-segment model.

Equations (1.1) to (1.8) are then transformed to the discrete form using a forward finite difference scheme. In the following, let subscript i denote a discrete quantity defined in cell i and subscript ij denote a discrete quantity defined at the connection between two cells i and j. Two cells share a connection if they are adjacent geometrically in the computational grid or share an explicit Non-Neighbor Connection ("NNC")[11]. For example $v_{o,ij}$ is the oil flux from cell i to cell j. For oriented quantities such as fluxes, the orientation is taken to be from cell i to cell j, and the quantity is skew-symmetric ($v_{o,ij} = -v_{o,ij}$), whereas non-oriented quantities such as the transmissibility T_{ij} are symmetric ($T_{ij} = T_{ji}$). Quantities with superscript 0 are taken at the start of the discrete time step, other quantities are at the end of the time step. Superscripts or subscripts applied to an expression in parenthesis apply to each element in the expression.

[11] *Connections between cells due to geometric adjacency are automatically calculated by the simulator, including across faults in the grid. In addition, the user can specify extra connections between arbitrary cells using the NNC keyword in the GRID section.*

The discretized equations and residuals are, for each pseudo-component α and cell i :

$$R_{\alpha,i} = \frac{\phi_{ref,i} V_i}{\Delta t}\left(A_{\alpha,i} - A^0_{\alpha,i}\right) + \sum_{j \in C(i)} u_{\alpha,ij} + q_{\alpha,i} = 0 \tag{1.9}$$

where the accumulation terms are the same as equations (1.2) to (1.4) and fluxes u_α are of similar form to the velocities, $\mathbf{u}_\alpha$ in the aforementioned equations; thus we have

$$A_{w\,i} = \left(m_\phi b_w s_w\right)_i \qquad u_{w\,i} = \left(b_w v_w\right)_i \tag{1.10}$$

$$A_{o\,i} = \left(m_\phi\left(b_o s_o + r_{og} b_g s_g\right)\right)_i \qquad u_{o\,i} = \left(b_o v_o + r_{og} b_g v_g\right)_i \tag{1.11}$$

$$A_{g\,i} = \left(m_\phi\left(b_g s_g + r_{go} b_o s_o\right)\right)_i \qquad u_{g\,i} = \left(b_g v_g + r_{go} b_o v_o\right)_i \tag{1.12}$$

In addition, equations (1.6) to (1.8) also hold for each cell i, that is:

$$\left(s_w + s_o + s_g\right)_i = 1 \tag{1.13}$$

$$\left(p_{c,ow}\right)_i = \left(p_o - p_w\right)_i \tag{1.14}$$

$$\left(p_{c,og}\right)_i = \left(p_o - p_g\right)_i \tag{1.15}$$

Finally, the fluxes are given for each connection ij by the following relationships:

$$\left(b_\alpha v_\alpha\right)_{ij} = \left(b_\alpha \lambda_\alpha m_T\right)_{U_{(i,\alpha,ij)}} T_{ij} \Delta\Phi_{\alpha,ij} \tag{1.16}$$

$$\left(r_{\beta\alpha} b_\alpha v_\alpha\right)_{ij} = \left(r_{\beta\alpha} b_\alpha \lambda_\alpha m_T\right)_{U_{(i,\alpha,ij)}} T_{ij} \Delta\Phi_{\alpha,ij} \tag{1.17}$$

$$\Delta\Phi_{\alpha,ij} = p_{\alpha,i} - p_{\alpha,j} - g\rho_{\alpha,ij}\left(z_i - z_j\right) \tag{1.18}$$

$$\rho_{\alpha,ij} = \frac{\left(\rho_{\alpha,i} + \rho_{\alpha,j}\right)}{2} \tag{1.19}$$

$$U(\alpha,ij) = \begin{cases} i, & \Delta\Phi_{\alpha,ij} \geq 0 \\ j, & \Delta\Phi_{\alpha,ij} \geq 0 \end{cases} \tag{1.20}$$

where:

V	=	cell volume.
Δt	=	time step length for the current Euler step.
v_α	=	volume flux of phase α (oriented quantity).
u_α	=	surface volume flux of pseudo component α (oriented quantity).
T_{ij}	=	transmissibility factor for a connection, derived from permeability.
m_T	=	transmissibility multiplier as function of pressure.
$C(i)$	=	connections from cell i, i. e. the set of cells connected to it.
$U(\alpha,ij)$	=	upwind cell for phase α for the connection between cells i and j.
$\Delta\Phi_{(\alpha,ij)}$	=	potential difference for phase α for the connection between cells i and j (oriented quantity).
g	=	gravitational acceleration in the z-direction.
z_i	=	depth of center of cell i.

For a grid containing a total of n active cells, the material balance relationship expressed by equation (1.9) results in a system of 3n equations for the three phase formulation (oil, water, gas). Similarly, for the two phase model, oil-water or gas-water, there are 2n equations to be solved. In addition, there are also the sink and source terms associated with the wells. The solution variables for the system of equations is dependent on the state of a cell, basically, for:

1) Non-miscible: Here pressure of one of the phases (which we take to be oil pressure p_o), s_w, and s_g as our primary variables. The pressure p_o will then behave mostly in an elliptic fashion, whereas the saturations s_w and s_g are more hyperbolic.

2) Miscible flow: Since the gaseous phase may disappear if all the gas dissolves into the oleic phase, and similarly the oleic phase may disappear if all the oil vaporizes into the gaseous phase, one cannot always use s_g for our third variable. Instead we use the third variable to track the composition of the phase that has not disappeared. We therefore use either s_g, r_{go}, or r_{og} as our third primary variable, depending on the fluid state, and we call that variable x:

$$x = \begin{cases} s_g, & \text{all three phases present,} \\ r_{go}, & \text{no gaseous phase,} \\ r_{og}, & \text{no oleic phase.} \end{cases} \tag{1.21}$$

This choice is made separately for each cell i depending on its state.

The next two sections describe the formulation of the two well models implemented in OPM flow: the Standard Well Model and the Multi-Segment Well Model.

1.3 STANDARD WELL MODEL

For the standard well model, in a three phase black-oil system, there are four primary variables, three fluid variables: Q_t, F_w, and F_g, which describe the fluid composition within the wellbore[12], as depicted in the following equation:

$$Q_t = \sum_{\alpha \in \{o,g,w\}} g_\alpha Q_\alpha, \qquad F_w = \frac{g_w Q_w}{Q_t}, \qquad F_g = \frac{g_g Q_g}{Q_t} \tag{1.22}$$

where:

Q_t = the weighted total flow rate,
F_w = the weighted fraction of water, and
F_g = the weighted fraction of gas.

Here, Q_α is the flow rate of component α under surface conditions and g_α is a weighting factor. For gas, this factor is typically chosen to be a small value, for example, 0.01 to avoid gas fractions close to unity[12].

The fourth variable is the bottom-hole pressure (p_{bhp}), that is the pressure in the wellbore at the datum depth.

Volumetric inflow rates at reservoir conditions are calculated as:

$$q^r_{\alpha,j} = T_{w,j}\, M_{\alpha,j} \left[p_j - (p_{bhp,w} + h_{w,j}) \right] \tag{1.23}$$

where:

$q^r_{\alpha,j}$ = is the flow rate of phase α through connection j. The rate is negative for flow from the wellbore to reservoir, and positive for flow into the opposite direction.

[12] *J. Holmes, Enhancements to the strongly coupled, fully implicit well model: wellbore crossflow modeling and collective well control, in: SPE Reservoir Simulation Symposium, Society of Petroleum Engineers, 1983.*

$T_{w,j}$	=	connection transmissibility factor.
$M_{\alpha,j}$	=	the mobility for phase α at the connection j. For injection wells, the total mobility of the injecting phase is used, as per Holmes12.
p_j	=	pressure of the grid block that contains the connection j.
$p_{bhp,w}$	=	the bottom-hole pressure of well w.
$h_{w,j}$	=	pressure difference within the wellbore between connection j and the well's bottom-hole datum depth. The pressure differences $h_{w,j}$ between connection j and the datum point are computed explicitly based on the fluid composition in the wellbore at the start of the time step.

To keep the system closed, we introduce conservation equations for each component using:

$$R_{\alpha,w} = \frac{A_{\alpha,w} - A^0_{\alpha,w}}{\Delta t} + Q_\alpha - \sum_{j \in C(w)} q_{\alpha,j} = 0 \tag{1.24}$$

Equation (1.24) is solved in a fully implicit and coupled way with the black-oil equations, as per equation (1.9)). Here, $C(w)$ is the set of connections of the well w, $q_{\alpha,j}$ is the flow rate of phase α through connection j under surface condition, which can be calculated from $q^r{}_{\alpha,j}$ defined in equation (1.23); the relationship is similar to the one between component and mass fluxes, as per equations (1.2) through (1.4). The storage term $A_{\alpha,w}$ describes the amount of component α in the wellbore (a small volume, that is 0.1 ft3 as per Holmes12), and it is introduced for better stability of the well solution.

In addition, we need equations that model how the wells are controlled. For wells controlled by a prescribed bottom-hole pressure (bhp) target, we have:

$$R_{c,w} = p_{bhp,w} - p^{target}_{bhp,w} = 0 \tag{1.25}$$

whereas for a rate-controlled well we have:

$$R_{c,w} = Q_\alpha - Q^{target}_\alpha = 0 \tag{1.26}$$

Where Q^{target}_α is the desired surface-volume rate of the controlled component α, typically oil or gas rate for a production well. The Standard Well model handles cross-flow, that is flow exiting from one well connection and entering another connection; however, this model assumes that the fluid composition is uniform throughout the wellbore (given by Fg and Fw), and thus the same composition will be injected through all injecting connections. When this is not a reasonable assumption, then the Multi-Segment model should be used instead, which is the topic of the next section.

1.4 MULTI-SEGMENT WELL MODEL

Multi-segment wells were introduced to handle more complex well configurations, including for example horizontal, multi-lateral and fish bone well types, as well Inflow Control Devices ("ICD")[13]. This model divides the wellbore into a number of arbitrary segments. Each segment has a segment node and a flow path to the neighboring segment in the direction towards the well head, that defines as the outlet segment (Figure 1.2). Most segments also have inlet segment neighbors in the direction away from the well head. For multi-lateral wells, a segment node must be placed at the branch junction. Segments at branch junction points have additional inlet segments and segments at the end of branches do not have inlet segments. The top segment of each well does not have an outlet segment. The pressure at this segment is the bottom-hole pressure of the well, whereas the component rates equal the component rates for the well as a whole. For example, Figure 1.3 on the following page, shows the main wellbore branch together with four offsetting branches.

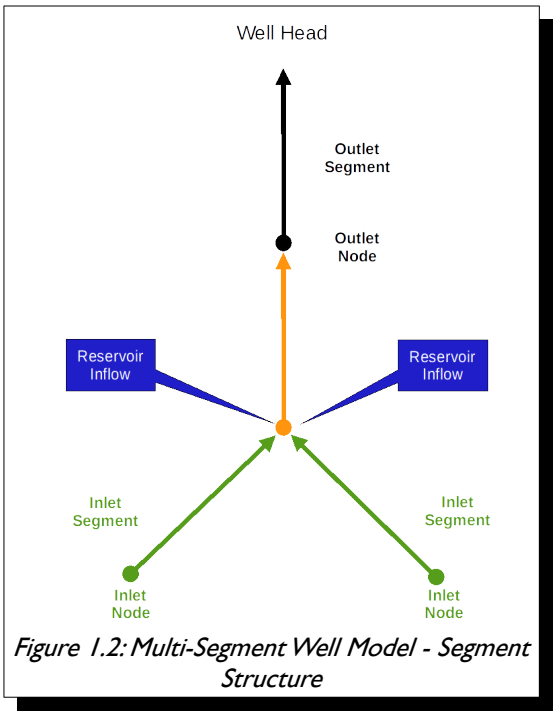


Figure 1.2: Multi-Segment Well Model - Segment Structure

Each segment has the node pressure p as a primary variable in addition to the primary variables Q_t, F_w, and F_g introduced for the standard well model, as per equation (1.22). Similarly, the component equations, (1.24), are extended to include the additional $Q_{\alpha,i}$ terms that represent the flow rate from the inlet segments $I(n)$ of segment n, as well as the inflow from the reservoir through the segment's connections $C(n)$. Thus, the equivalent standard well equation (1.24) for multi-segment wells is:

$$R_{\alpha,n} = \frac{A_{\alpha,n} - A^0_{\alpha,n}}{\Delta t} + \sum_{i \in I(n)} Q_{\alpha,i} - \sum_{j \in C(w)} q_{\alpha,j} + Q_{\alpha,n} = 0 \qquad (1.27)$$

Each segment can thus receive inflow from more than one connection or no connection through the segment node, whereas each connection can only contribute to one segment. In the standard well model, the connection is located at the centroid of the grid block. In the multi-segment well model, the connection and segment node can be located at any place within the grid block. The inflow calculation of the connection in the multi-segment well model is as follows:

$$q^r{}_{\alpha,j} = T_{w,j} M_{\alpha,j} \left(p_j + H_{c,j} - p_n - H_{nc} \right) \qquad (1.28)$$

where:

p_j = the pressure at the cell center of the grid block that contains connection j,

$H_{c,j}$ = the hydrostatic pressure difference between the cell center and the connection,

p_n = the pressure of segment n, and

H_{nc} = the hydrostatic pressure difference between the connection and the segment.

[13] *J. Holmes, T. Barkve, O. Lund, Application of a multisegment well model to simulate ow in advanced wells, in: European Petroleum Conference, Society of Petroleum Engineers, 1998.*

The last equation for each segment describes the pressure relationships between the segment n and its outlet segment m:

$$R_{p,n} = p_n - p_m - H_h - H_f - H_a = 0 \tag{1.29}$$

Here, the H terms represent the hydrostatic, frictional, and acceleration pressure drops between the segments.

The top segment does not have an outlet segment, and hence the pressure equation is replaced by a well control equation, which is the same as for the standard well model, that is either equation (1.25) or (1.26).

As with the standard well model, the well equations are solved with the reservoir mass conservation equations in a coupled, fully implicit way. The well equations are added to the global system using a Schur[14] complement approach:

$$\left(A - \sum_w C_w D_w^{-1} B_w\right) x_r = R_r - \sum_w C_w D_w^{-1} R_w \tag{1.30}$$

$$x_w = D_w^{-1}\left(R_w - B_w x_r\right)$$

Figure 1.3 depicts a multi-segment well with the main wellbore branch as the vertical segment, together with four offsetting horizontal branches.

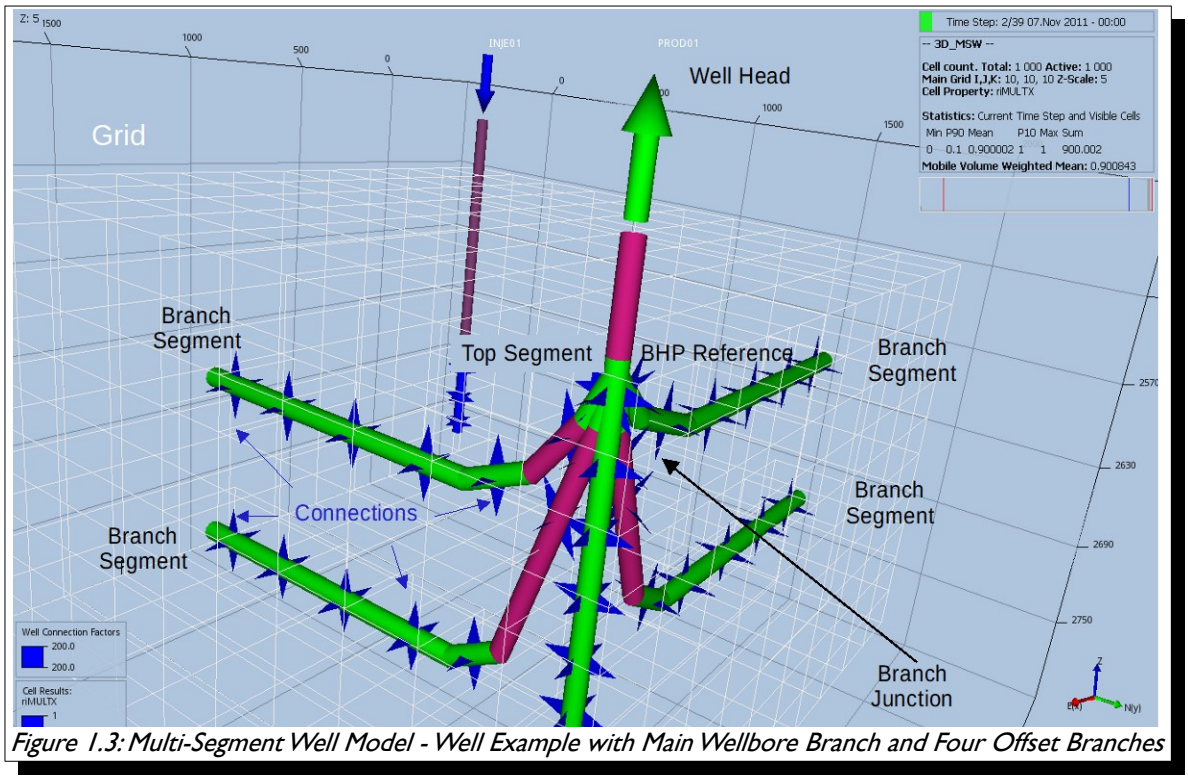


Figure 1.3: Multi-Segment Well Model - Well Example with Main Wellbore Branch and Four Offset Branches

[14] *Zhang, Fuzhen (2005). Zhang, Fuzhen (ed.). The Schur Complement and Its Applications. Numerical Methods and Algorithms. Vol. 4. Springer. doi:10.1007/b105056. ISBN 0-387-24271-6.*

1.5 SALT PRECIPITATION MODEL

OPM Flow's salt precipitation and water evaporation model is an extension of the black-oil model with a water in gas component, plus addition of a solid salt phase component to the salt transport equation. The effects of porosity and permeability reduction effects are also considered. The oil, gas and water and salt conservation equations including these extensions are given by equations (1.31) to (1.34).

Oil:

$$\frac{\partial}{\partial t}\left[\phi\left(b_o s_o + r_{og} b_g s_g\right)\right] + \nabla\cdot\left(b_o v_o + r_{og} b_g v_g\right) + q_o = 0 \tag{1.31}$$

Gas:

$$\frac{\partial}{\partial t}\left[\phi\left(b_g s_g + r_{go} b_o s_o\right)\right] + \nabla\cdot\left(b_g v_g + r_{go} b_o v_o\right) + q_g = 0 \tag{1.32}$$

Water:

$$\frac{\partial}{\partial t}\left[\phi\left(b_w s_w + r_{wg} b_g s_g\right)\right] + \nabla\cdot\left(b_w v_w + r_{wg} b_g v_g\right) + q_w = 0 \tag{1.33}$$

Salt:

$$\frac{\partial}{\partial t}\left[\phi b_w s_w c_w^{salt} + m_\phi \phi_0 s_s \rho^{salt}\right] + \nabla\cdot\left(c_w^{salt} b_w v_w\right) + c_w^{salt} q_w = 0 \tag{1.34}$$

where:

ϕ	=	porosity.
m_ϕ	=	pore volume multiplier as function of pressure.
ϕ_0	=	reference porosity, a constant in time but varying in space.
p_α	=	pressure for phase α.
b_α	=	shrinkage/expansion factor for phase α defined as the ratio of the surface volume at standard conditions to the reservoir volume for a given amount of fluid: $b_\alpha = V_{surface;\alpha} \,/\, V_{reservoir;\alpha}$. For the oil phase, b_o is called the shrinkage factor, whereas for gas b_g is called the expansion factor. The reciprocal quantity is called the formation volume factor, and is usually denoted with a capital B: $B_\alpha = 1 / b_\alpha$. The variable is usually a function of phase pressure and composition.
r_{go}	=	gas-oil ratio ("GOR"), ratio of dissolved gas to oil in the oleic phase, commonly called r_s or R_s in other literature.
r_{og}	=	oil-gas ratio or condensate-gas ratio ("CGR"), ratio of vaporized oil to gas in the gaseous phase, commonly called r_v or R_v in other literature.
r_{wg}	=	evaporated water in gas ratio.
s_α	=	saturation of phase α, the pore volume fraction occupied by the phase.
s_s	=	saturation of precipitated salt (volume fraction).
ρ_α	=	surface density of phase α at one atmosphere, a given constant.
ρ^{salt}	=	density of the solid salt.

q_α = well out flow of component α, (negative for well in-flow).
The form of this term depends on the well model used, either the standard well model or the multi-segment model.

c_w^{salt} = salt concentration in water (mass per unit volume).

The extension has two extra primary variable switching mechanisms. One between s_w and r_{wg}, when water disappears and reappears, and one between c_w^{salt} and s_s when the salt concentration reaches the (user defined) solubility limit and salt precipitates or when solid salt dissolves. Furthermore, in the new formulation porosity becomes:

$$\phi = \left(1 - s_s\right) m_\phi \phi_0 \tag{1.35}$$

The resulting (reduced) permeability, k, is based on a user specified permeability-porosity relation $k = k(\phi)$.

Both for dry and humid gas and wet and humid gas the PVT data can be provided using dedicated PVT tables. For the capillary pressure data it is recommended to extend its values towards zero water saturation to permit water saturations below the irreducible water saturation due to water evaporation.

For further details see Machado et al.[15]

[15] *IC. G. Machado, P. Egberts, J. Alvestad, O. S. Hustad. Salt Precipitation and Water Evaporation Modelling in a Black-Oil Reservoir Simulator. SPE Reservoir Simulation Conference 2023 (SPE-212257).*

1.6 NUMERICAL SOLUTION OF EQUATIONS

Efficiently solving the aforementioned equations is non-trivial, and involves a step iterative process that consists of an outer, Newton-Raphson type method, of a series of iterations, and an inner set of linear iterations, as depicted in Figure 1.4, which shows a simplified schematic of the iterative scheme.

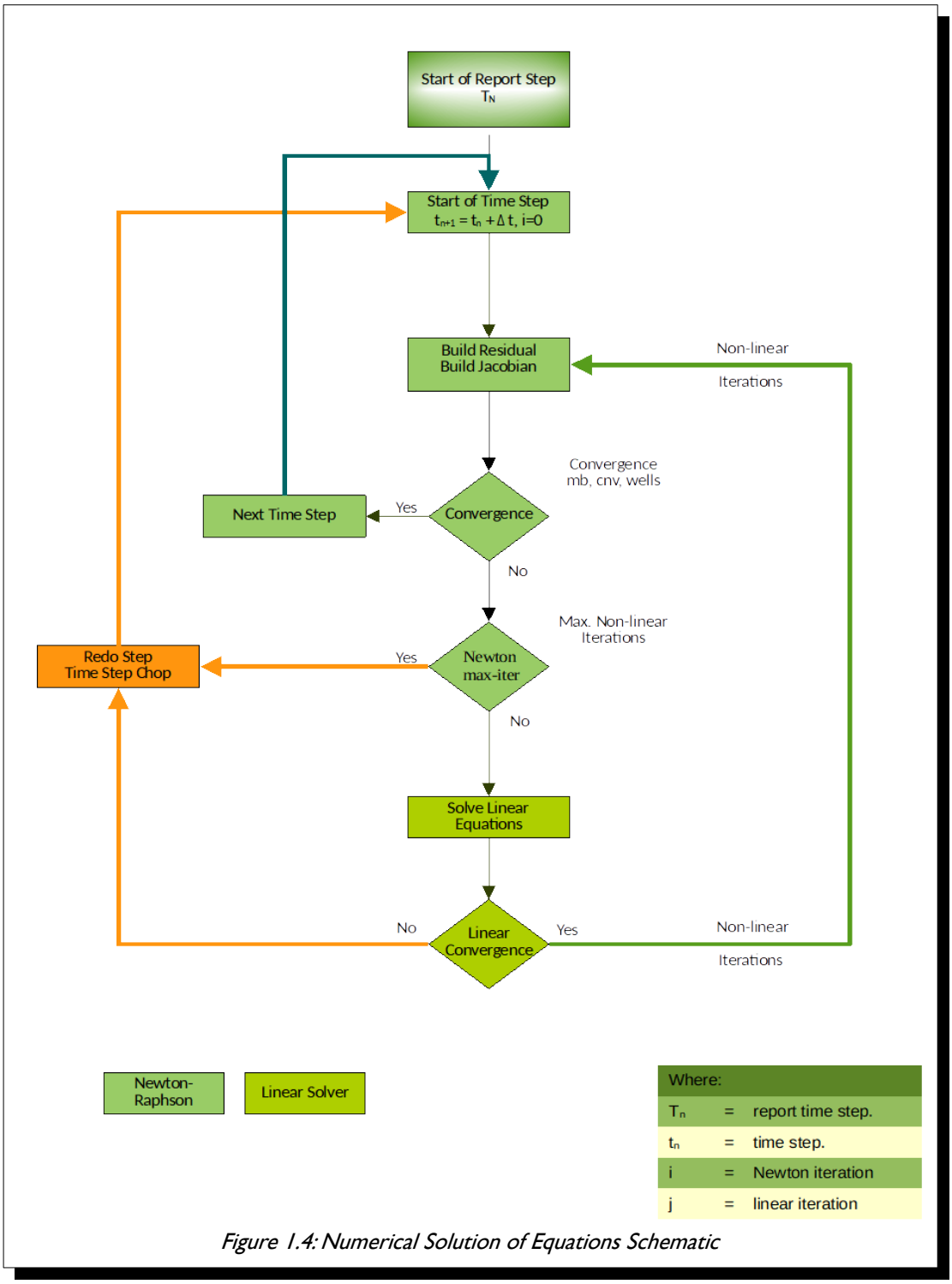


Figure 1.4: Numerical Solution of Equations Schematic

The next two sections attempt to give an overview of the two numerical schemes.

1.6.1 NEWTON-RAPHSON SOLVER (OUTER ITERATION)

The aforementioned reservoir and well equations define a large system of (fully implicit) non-linear equations, which one can write in compact residual form as R(y) = 0, where y is the vector of primary variables, p_o, s_w, x. This system is solved using a Newton-Raphson type method.

Let $y_n=(p_o, s_w, x)$ be the primary variables after n Newton iterations, and given the initial state y_0, the solution of $R(y)=0$ can be found by iteratively solving:

$$J(y_n)(y_{n+1} - y_n) = -R(y_n) \tag{1.36}$$

until $R(y_n)$ satisfies a given tolerance. Creating the Jacobian matrix $J(y_n)$ and solving this linearized problem is the core computational task of the simulator, and its performance depends largely on how this part of the code is programmed.

The coefficients of the linearized systems required by the non-linear Newton solver, $J(y_n)$ in equation (1.36), have traditionally been computed by evaluating closed-form expressions obtained by differentiating the discretized flow equations analytically. Differentiating these flow equations manually and programming the resulting formulas is both time-consuming and error-prone. The problem is particularly pronounced when extending an existing simulator with new functional relationships. If a quantity is set to depend on a different or new (primary) variable, the changes in derivatives will propagate through the chain rule to the Jacobians of all quantities that depend on the modified quantity. Thus, even very simple extensions in functional dependencies may cause large modifications throughout the simulator code.

Automatic Differentiation ("AD") is a way to mitigate this problem[16] and [17], and there are many examples of AD tools used for scientific computing, such as Sacado[18], ADOL-C[19] or ADETL[20]; with the latter used for the AD-GPRS[21] reservoir simulator at Stanford University.

The OPM Flow's implementation of AD introduces a class that mimics the behavior of the built-in floating point types of C++ (double and float) as closely as possible. The AD object contains a value and a fixed number of derivatives and defines all basic arithmetic operators as well as common mathematical functions. Also, to allow easy comparisons with code that uses standard floating-point objects, all functions that work on AD-objects can also be used with objects of built-in floating-point types. For further detailed information on OPM Flow's implementation of AD, again see Rasmussen et al.[22].

When solving the Newton-Raphson equation (1.36), the first iteration at the new time step uses the solution at the end of the previous time step (t_n), as an initial estimate for the new time step (t_{n+1}). Thus, the Newton-Raphson iteration is expected to converge rapidly for sufficiently small time steps. For large time steps and/or large changes in the solution, typically caused by changing well controls, the Newton-Raphson method may converge poorly or not at all. There are several means in which one can increase the probability of convergence, namely:

[16] *R. D. Neidinger, Introduction to automatic differentiation and matlab object-oriented programming, SIAM Rev. 52 (3) (2010) 545{563. doi:10.1137/080743627.*

[17] *A. Griewank, A. Walther, Evaluating Derivatives, Principles and Techniques of Algorithmic Differentiation, second edition, SIAM, Philadelphia, 2008.*

[18] *Eric T. Phipps, Roscoe A. Bartlett, David M. Gay, Robert J. Hoekstra, Large-Scale Transient Sensitivity Analysis of a Radiation-Damaged Bipolar Junction Transistor via Automatic Differentiation Advances in Automatic Differentiation, Springer, 2008.*

[19] *A. Walther, A. Griewank, Getting started with ADOL-C, in: U. Naumann, O. Schenk (Eds.), Combinatorial Scientific Computing, Chapman-Hall CRC Computational Science, 2012, Ch. 7, pp. 181-202.*

[20] *Rami M. Younis, Modern Advances In Software And Solution Algorithms For Reservoir Simulation, PhD thesis, Stanford University, (August 2011)*

[21] *AD-GRPS, Stanford Earth, Stanford University, https://supri-b.stanford.edu/research/ad-gprs.*

[22] *Rasmussen, A.F., Sandve, T.H., Bao, K., Lauser, A., Hove, J., Skaflestad, B., Klöfkorn, R., Blatt, M., Rustad, A.B., Sævareid, O. et al. [2019]. The Open Porous Media Flow Reservoir Simulator. arXiv preprint arXiv:1910.06059.*

1) Restrict the change allowed per iteration and prevent the solution from jumping far across the boundary between saturated and undersaturated states in a single iteration. Thus, like the commercial simulator, OPM Flow implements the Appleyard[23] chop technique, which is often sufficient for the method to converge. However, time step chops[24] may still occur in order to obtain convergence for some difficult cases, particularly for models with very fine grids.

2) Secondly, since the third primary variable, x ,may change solution state from one iteration to the next, this may disturb the convergence, and therefore a small threshold is applied (ε) to the variable-switching logic for equation (1.21), in order to prevent oscillation between states.

3) Thirdly, and frequently the main cause of non-convergence and the lack of smoothness of the residuals, is the discontinuity or inconsistent input data, particularly the relative permeability and capillary pressure functions. Thus, using Corey[25] or Lomeland et al.[26] type of curves for these data may well improve convergence performance, see *SGOFLET– Gas-Oil LET Relative Permeability Functions*, *SGWFLET – Gas-Water LET Relative Permeability Functions*, and *SWOFLET – Water-Oil LET Relative Permeability Functions*, for OPM Flow's implementation of the Lomeland type of curves.

4) The final point that influences the convergence of the non-linear solver is the accuracy of the linear solvers, as discussed in the next section.

For more information about run time options and numerical controls, see section *2.2 Running OPM Flow 2023-10 From The Command Line* on how to invoke various numerical schemes via the OPM Flow command line interface.

[23] *J. Appleyard, I. M. Cheshire, Nested factorization, in: 7th SPE Symposium on Reservoir Simulation, San Francisco, USA, Society of Petroleum Engineers, 1983. doi:10.2118/12264-MS.*

[24] *A time step chop occurs when convergence is not achieved within the allowable number of iterations. For example, if the simulator was attempting to take a 30 day time step and failed, then the time step would be cutback, to say ten days, and the iterations restarted. If this 10 day time step also failed to converge, then a second time step chop would occur reducing the new time step, to say three days. This process is repeated until either convergence is achieved, or the time step is too small to chop and the simulation stops.*

[25] *Corey, A. T. : "The Interrelation Between gas and Oil Relative Permeabilities", Production Mon., 19. 38. (1954).*

[26] *Lomeland F. , 2018.. Overview Of The Let Family Of Versatile Correlations For Flow Functions. Paper SCA2018-056 presented at the International Symposium of the Society of Core Analysts held in Trondheim, Norway, 27-30 August 2018.*

1.6.2 LINEAR SOLVER (INNER ITERATION)

As mentioned previously, efficiently solving the aforementioned equations is non-trivial, and involves a step iterative process that consists of an outer, Newton-Raphson type method, series of iterations, and an inner set of linear iterations, as depicted in the detailed work flow of the iterative process in Figure 1.5.

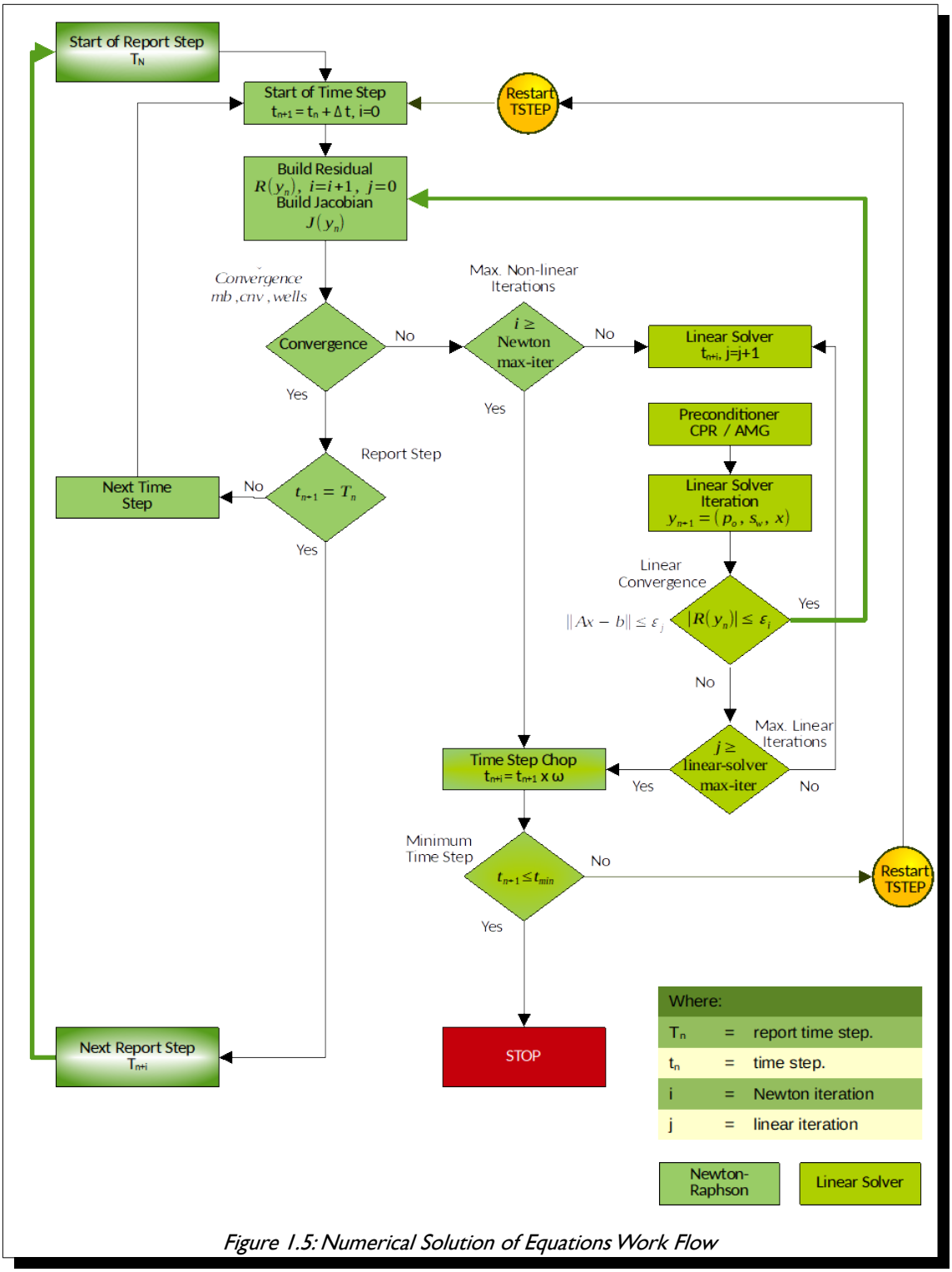


Figure 1.5: Numerical Solution of Equations Work Flow

Depending on the fluid properties and other reservoir specific parameters, the system of equations as described by equation (1.9), that is:

$$R_{\alpha,i} = \frac{\phi_{ref,i} V_i}{\Delta t}\left(A_{\alpha,i} - A^0{}_{\alpha,i}\right) + \sum_{j \epsilon C(i)} u_{\alpha,ij} + q_{\alpha,i} = 0 \quad \textbf{(1.9)}$$

could exhibit elliptic or degenerate parabolic behavior, or could even be hyperbolic for specif setups. Therefore, solving the linearized system stemming from these equations is challenging, since it is likely non-symmetric and ill-conditioned. Thus, instead of solving equation (1.36), that is:

$$J(y_n)\left(y_{n+1} - y_n\right) = -R(y_n) \quad \textbf{((1.36))}$$

directly, the solution is approximated by an iterative linear solver. Two variants are available in OPM Flow:

1) a stabilized Bi-conjugate Gradient method (BiCG-stab), and
2) a restarted Generalized Minimal Residual (GMRes) solver.

By default, BiCG-stab is used. In addition, there are several options available for preconditioning the linear system. The default choice being the incomplete lower triangular-upper triangular factorization with level of fill in 0 (ILU0). More sophisticated choices are based on an algebraic multi-grid (AMG) method for preconditioning, such the Constraint Pressure Residual (CPR) preconditioner (originally introduced by Wallis[27] using the approach of Scheichl et al.[28]). The solvers and preconditioners are implemented in the Dune module dune-istl and described by Blatt et al.[29], [30], and [31]. The interface between OPM Flow and the linear solver packages has been designed in a flexible way so that besides dune-istl other linear algebra packages could be used.

For more information about run time options and numerical controls, see section *2.2 Running OPM Flow 2023-10 From The Command Line* on how to invoke various numerical schemes via the OPM Flow command line interface. In addition, section *2.4 Improving Simulator Convergence and Numerical Performance* provides some practical advice on how to address numerical performance issues.

[27] *J. Wallis, Incomplete Gaussian Elimination as a Preconditioning for Generalized Conjugate Gradient Acceleration, in: 7th SPE Symposium on Reservoir Simulation, San Francisco, USA, Society of Petroleum Engineers, 1983. doi:10.2118/12265-MS.*

[28] *R. Scheichl, M. Roland, J. Wendebourg, Decoupling and block preconditioning for sedimentary basin simulations, Computational Geosciences 7 (2003) 295{318.*

[29] *M. Blatt, A parallel algebraic multigrid method for elliptic problems with highly discontinuous coefficients, Ph.D. thesis, Ruprecht-Karls-Universitat Heidelberg (2010).*

[30] *M. Blatt, P. Bastian, The Iterative Solver Template Library, in: B. Kågström, E. Elmroth, J. Dongarra, J. Waśniewski (Eds.), Applied Parallel Computing. State of the Art in Scientific Computing, Vol. 4699 of Lecture Notes in Computer Science, Springer, 2007, pp. 666{675*

[31] *M. Blatt, P. Bastian, On the generic parallelisation of iterative solvers for the finite element method, Int. J. Comput. Sci. Engrg. 4 (1) (2008) 56-69. doi:10.1504/IJCSE.2008.021112.*

1.7 FORMATION DAMAGE MODELS

1.7.1 INCOMPRESSIBLE EXTERNAL FILTER CAKE MODEL – LINEAR GEOMETRY

This simple model assumes that all of the filtrate is captured on the formation surface to form a filter cake of constant porosity and permeability. Hence, the filter cake will simply grow in thickness as a function of the volume of deposited material, and a simple volume balance assuming linear flow yields,

$$A_f l_c(t)(1-\phi_c) = f_s \int_0^t q(\tau)\, d\tau = f_s Q_c(t) \tag{1.36}$$

Here A_f is the area of flow, l_c is the filter cake thickness, ϕ_c is the filter cake porosity, f_s is the volume concentration of suspended material, q is the injection rate, Q_c is the cumulative injected volume, and t is time. The filter cake thickness is hence,

$$l_c(t) = \frac{f_s Q_c(t)}{A_f(1-\phi_c)}. \tag{1.36}$$

Assuming a constant filter cake permeability k_c, and viscosity μ, Darcy's law for the pressure drop across the filtrate gives the damage pressure drop ΔP_d as,

$$\Delta P_d(t) = \frac{q\mu f_s Q_c(t)}{k_c A_f^2(1-\phi_c)} \tag{1.36}$$

Skin factor,

$$S = \frac{f_s}{1-\phi_c}\frac{k}{k_c}\frac{Q_c(t)}{A_f r_w} = \frac{l_c}{r_w}\frac{k}{k_c} \tag{1.36}$$

Here k is the rock permeability. The linear flow assumption is valid for flow near a fracture surface, and it is also a decent approximation for radial flow assuming that the filter cake is thin compared to the well radius.

1.7.2 INCOMPRESSIBLE EXTERNAL FILTER CAKE MODEL – RADIAL GEOMETRY

Deposition Inside the Wellbore

Simple volume balance on deposited filtrate inside a well in radial geometry yields, for a 2D cross section perpendicular to the well,

$$\pi r_w^2 - \pi r_c^2 = \frac{f_s}{1-\phi_c}\int_0^t q_h(t)\, dt = \frac{f_s}{1-\phi_c} Q_h(t) \tag{1.36}$$

Here q_h and Q_h are rate and cumulative injected volume per length. This may be solved for the filtrate inner radius r_c to compute the cake thickness $r_w - r_c$ as,

$$\Delta r_c(t) = r_w - r_c = r_w - \sqrt{r_w^2 - \frac{f_s Q_h(t)}{\pi(1-\phi_c)}} \tag{1.36}$$

The pressure drop across the filter cake is then given as,

$$\Delta P_c = \frac{q\mu}{2\pi k_c} \ln \frac{r_c}{r_w} \tag{1.36}$$

The skin is simply,

$$S = \frac{k}{k_c} \ln \frac{r_w}{r_c} \tag{1.36}$$

Deposition in the Reservoir

If the filtrate is instead deposited in the near-well reservoir (i.e., outside the well), we will have $r_c > r_w$, and the skin factor will be,

$$S = \frac{k}{k_c} \ln \frac{r_c}{r_w} \tag{1.36}$$

For implementation convenience and consistency with the linear geometry case the filter cake thickness may be expressed in terms of a connection area, $A_f = 2\pi r_w h$, so that with Q being the cumulative injected volume, the filter cake radius follows from,

$$r_c^2 = r_w^2 + \frac{2 r_w f_s Q(t)}{A_f(1-\phi_c)} \tag{1.36}$$

CHAPTER 2: INSTALLING AND RUNNING FLOW

2.1 INSTALLING OPM FLOW

OPM Flow can be installed and used in a variety of ways. Perhaps the simplest way is to use the operating system's package manager to install the OPM modules as binary packages, including OPM Flow and other OPM programs. That way all prerequisite libraries will automatically be installed on the system. This is supported on Ubuntu Linux version 20.04 and Red Hat Enterprise (or CentOS) version 6 or 7.

OPM Flow can also be used via a Docker container. This allows OPM Flow to run on any system that supports Docker, including Microsoft Windows, without installing prerequisite libraries or affecting any other software on your system.

Finally, OPM Flow can be installed by compiling from source on Linux or macOS systems.

2.1.1 UBUNTU LINUX 20.04 (64-BIT VERSION ONLY) AND HIGHER

The easiest way to install the Ubuntu packages is to first add the OPM personal package archive (ppa). In order to do that we need to install the apt-add-repository command.

```
sudo apt-get update
sudo apt-get install software-properties-common
```

Then we add the repository, and run update again:

```
sudo apt-add-repository ppa:opm/ppa
sudo apt-get update
```

At this point, all the OPM modules should be available to install. To see a list of (for example) the opm-simulators packages:

```
apt-cache search opm-simulators
```

Then to install the opm-simulators programs (including OPM Flow) and their dependencies:

```
sudo apt-get install mpi-default-bin
sudo apt-get install libopm-simulators-bin
```

To optionally install the OPM Python bindings use:

```
sudo apt install python3-opm-simulators
```

Notes

1) The mpi install above is required for the mpi libraries which the OPM executables are linked to, even those that are not intended to be run with mpirun. The reason the OPM packages do not depend on that package is that by Debian policy the user should be allowed to choose which MPI implementation to use (above we picked the default set by Ubuntu). Also, note that due to some problems with the OpenMPI packages in Ubuntu 18.04, OPM has rolled out its own custom MPI packages to prevent this bug for this release of the operating system.

2) If an old versions of prerequisite libraries have already been installed (for example from installing a previous release of OPM Flow) it might be necessary to upgrade them via the following commands:

```
sudo apt-get update

sudo apt-get upgrade
```

3) Some users have experienced trouble with OPM Flow simply aborting with no error message. In some cases that can be related to locale issues. Check by running the "locale" command. OPM Flow requires the "C" or an English locale to run correctly. One way to fix this can be to put the following in your ".bash_profile" (note the initial period or full stop in the filename), and open a new terminal (command) window:

```
LANG="en_US.UTF-8"

export LANG

LC_ALL="en_US.UTF-8"

export LC_ALL
```

2.1.2 RED HAT ENTERPRISE OR CENTOS (VERSION 6 OR 7)

First add the OPM package repository:

```
sudo yum-config-manager --add-repo \
http://www.opm-project.org/package/opm.repo
```

The OPM software is split in several packages. To list all available OPM packages one can use:

```
sudo yum search opm-
```

For example you can install the opm-simulators binary package to get access to the OPM Flow reservoir simulator:

```
sudo yum install opm-simulators-bin
```

Development packages are available using the -devel suffix. For example, to install the opm-upscaling development package use:

```
sudo yum install opm-upscaling-devel
```

Notes

1) If you wish to install the MPI parallel version of Flow, you must choose an MPI implementation, either mpich or openmpi, and instead use the following command:

```
sudo yum install opm-simulators-openmpi-bin
```

2) For mpich, just replace openmpi with mpich in the above command. In order to run Flow in parallel using mpirun you must also load the required module first:

```
module load mpi/openmpi-x86_64
```

The above command works for RHEL 7, for RHEL 6 you must drop the "mpi/" part. To use mpich instead, replace openmpi with mpich in the above command.

2.1.3 WINDOWS 7 AND 10 USING VIRTUALBOX

Oracle VM VirtualBox's (formerly Sun VirtualBox, Sun xVM VirtualBox and Innotek VirtualBox) is a free and open source hosted hypervisor for x86 virtualization, developed by Oracle Corporation. The software was originally created by Innotek, who were acquired by Sun Microsystems in 2008, which was in turn acquired by Oracle in 2010. VirtualBox can be installed on Windows, macOS, Linux, Solaris and Open Solaris operating systems. The software supports the creation and management of guest virtual machines running Windows, Linux, BSD, OS/2, Solaris, Haiku, and OSx86, as well as limited virtualization of macOS guests on Apple hardware For some guest operating systems, a "Guest Additions" package of device drivers and system applications is available, which typically improves performance, especially that of graphics[32].

The OPM web site outlines the install process using VirtualBox, Vagrant[33] and a Vagrant file to automatically set up a virtual environment – see *https://opm-project.org/?page_id=294&page=2* for details on this approach.

Alternatively, one can use the more standard approach of installing virtual machines using VirtualBox. The basic outline for this method to use VirtualBox to run a guest Linux distribution and install OPM Flow in the guest Linux virtual machine. Note only the Linux distributions mentioned previously should be used as a guest virtual machine, although any of the Ubuntu "flavors" (Kubuntu, Lubuntu, Ubuntu Mate etc.) will work as well. The steps are:

1) Installing VirtualBox[34]
   - First download the latest VirtualBox software from Oracle's web site at *https://www.virtualbox.org/* for Windows host machines.
   - Install the software via double clicking on the executable and following the on screen instructions.
2) Create a Guest Linux Virtual Machine.
   - Download a suitable Linux distribution in ISO format that supports running OPM Flow, for example Ubuntu 20.04 LTS (64-bit version only) from *https://ubuntu.com/download/desktop* or CentOS Linux (version 6 or 7) from *https://www.centos.org/download/*.
   - Start VirtualBox, which should display a screen similar to Figure 2.1 without all the guest virtual machines.

[32] *After* https://en.wikipedia.org/wiki/VirtualBox.

[33] *See* https://www.vagrantup.com/ *for details*

[34] *See also* https://www.virtualbox.org/wiki/Documentation *for further and more detailed information.*

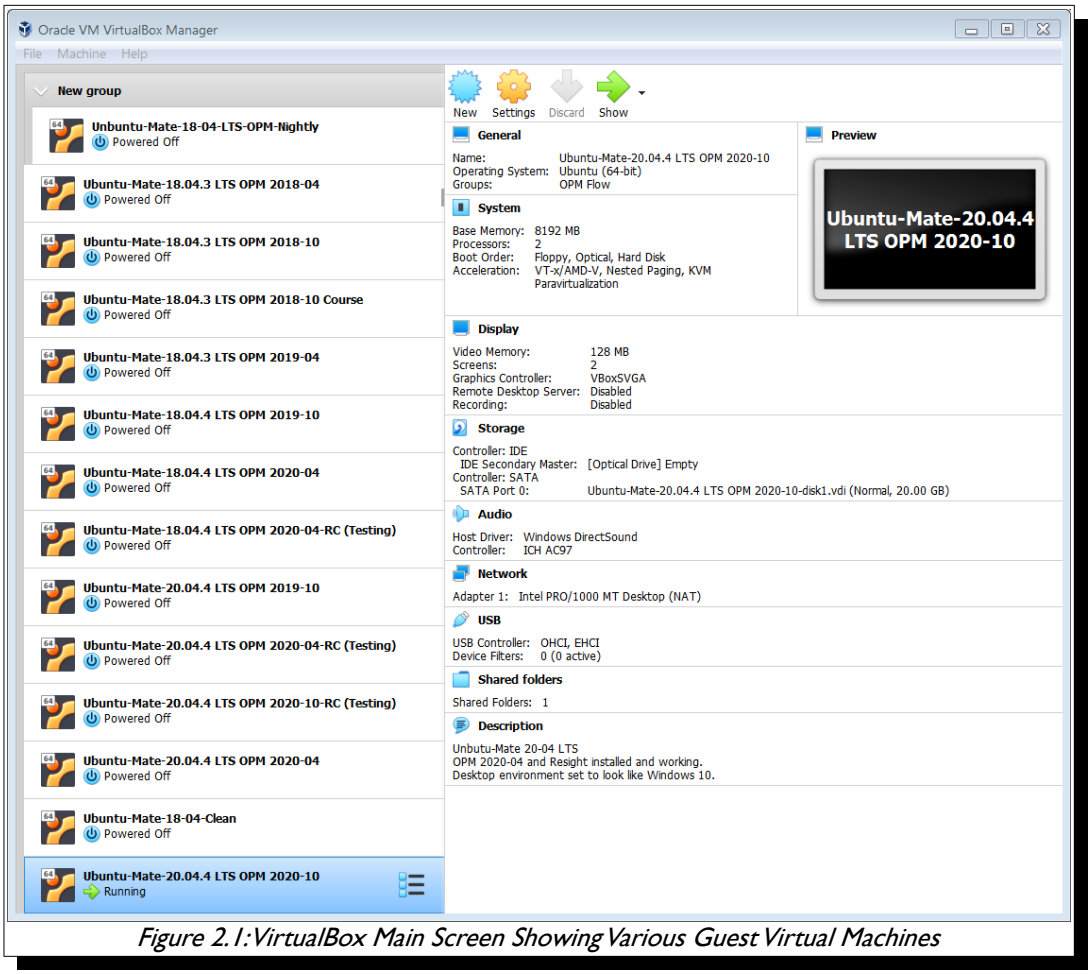


Figure 2.1: VirtualBox Main Screen Showing Various Guest Virtual Machines

- Then click the "New" icon button to install a new guest virtual machine.
- Identify the operating system type (Linux) and the Linux distribution that one wishes to install (Ubuntu 64-bit or RedHat for one of the CentOS distributions).
- Set the amount of RAM, should be 4GB or more; note the guest machine properties can subsequently be changed after the guest machine has been created.
- Create a virtual hard drive, the default virtual disk of VDI is normally used, and select either the dynamic or the fixed option. Next select the size of the virtual disk, the size should be sufficient to allow for the operating system and any other software, including OPM Flow, to be installed. For Ubuntu 20.04 LTS the operating system needs at least 10 GB. VirtualBox will then create the guest virtual machine.
- The next step is to start the guest virtual machine and installing the Linux operating system. This can be done by double clicking on the guest virtual machine and then selecting the previously downloaded IOS file. For Ubuntu 20.04 LTS the machine will boot into a live distribution, that is the guest virtual machine is running from a “live CD”, shown below.

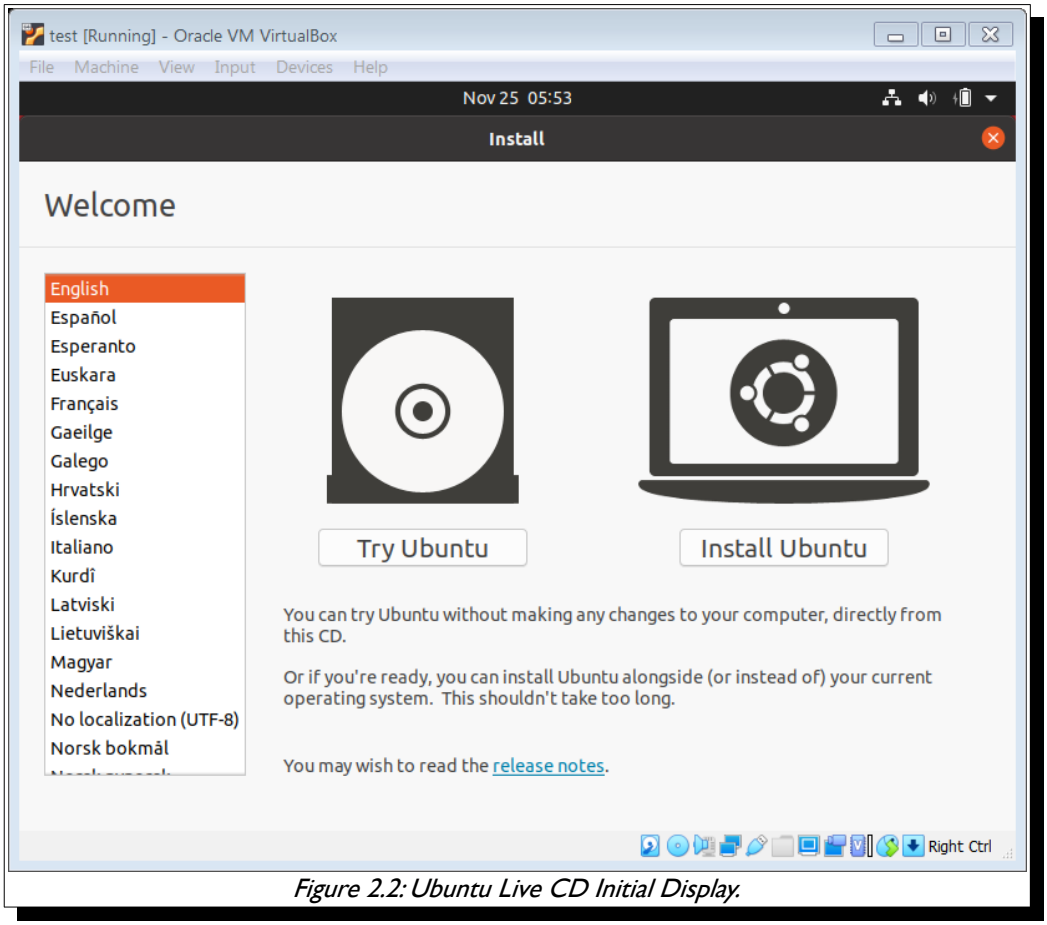


Figure 2.2: Ubuntu Live CD Initial Display.

- Select the "Install Ubuntu" option to install the operating system. The "Try Ubuntu" option means the guest machine will run from a "Live CD".
- Follow the on screen instructions to install the operating system, together with any updates suggested by the installer. Once the installation is complete the guest machine will re-boot as a normal guest Linux machine.

3) Installing OPM Flow in the Guest Virtual Machine.

- Boot up your virtual machine by selecting the machine and selecting the "Start" icon on the main VirtualBox screen (Figure 2.1). One should then be presented with the newly created guest virtual machine, similar to one shown in Figure 2.3, that displays the standard Ubuntu desktop.

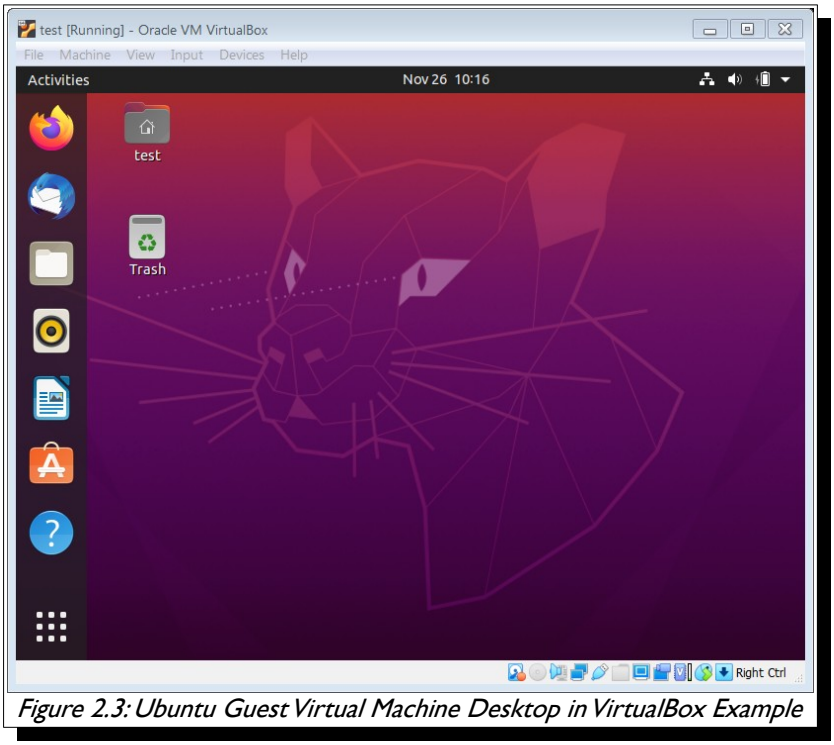


Figure 2.3: Ubuntu Guest Virtual Machine Desktop in VirtualBox Example

- In order to install OPM Flow in the guest virtual machine one first needs to open a command terminal using ALT+CNTL+T key sequence (Figure 2.4).

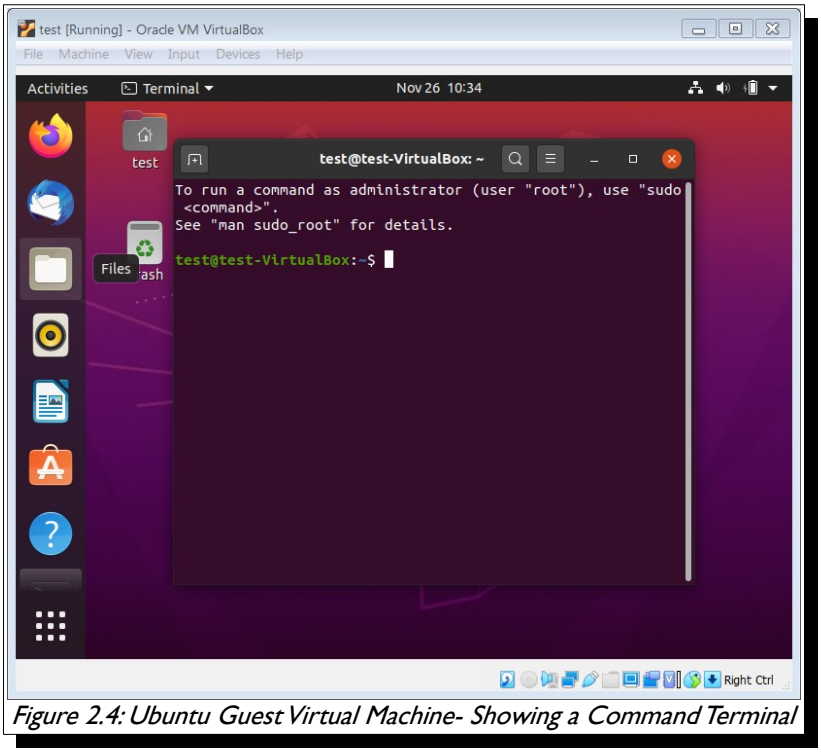


Figure 2.4: Ubuntu Guest Virtual Machine- Showing a Command Terminal

- One can then use the same commands as for a host machine, as described earlier in Sections *2.1.1 Ubuntu Linux 20.04 (64-bit version only) and Higher* and *2.1.2 Red Hat Enterprise or CentOS (version 6 or 7)* to install the software. To check if OPM Flow is installed, execute the following command in the terminal window:

```
flow --help
```

Which should then display the command line for the installed version of OPM Flow, as depicted in Figure 2.5

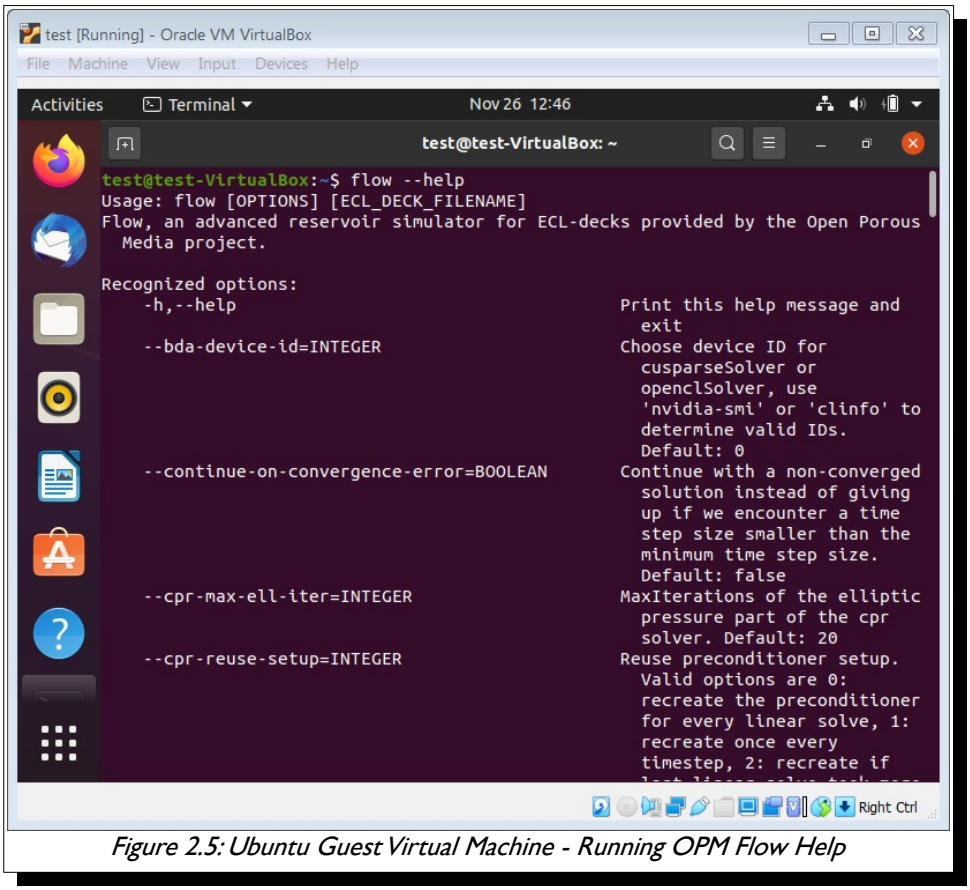


Figure 2.5: Ubuntu Guest Virtual Machine - Running OPM Flow Help

4) Using OPM Flow in a Linux Virtual Machine.

- The guest virtual machine only has access to files within the guest machine. To enable the guest to have access to the host machine files; on the main VirtualBox screen (Figure 2.1), select the "Setting" icon and then the "Shared Folders" option to set a share directory between the guest and the host machines. One can then have all the input and output files on the host machine and run OPM Flow in the guest virtual machine.
- Note one should shut down the guest virtual machine in the normal way to avoid potential file corruptions.
- One may also take snapshots of the current state of the virtual machine, as a form of backup if one changes the configuration of the virtual machine, or if one updates to the latest release of OPM Flow.

Historically this has been one of the preferred methods to run OPM Flow on Windows machines and still has several advantages, including being able to export and import various guest virtual machines for testing various versions of the simulator. However, Microsoft, with the introduction of Windows Subsystem for Linux ("WSL") have create a more seamless method for running OPM Flow on Windows 10 machines. This option is outlined in the next section.

2.1.4 WINDOWS 10 – WINDOWS SUBSYSTEM FOR LINUX

The first release of WSL was on August 2nd, 2016 and there has been various updates to this version which is known as WSL 1. On May 6th, 2019 WSL 2 was announced, that was a complete re-write of the Windows Subsystem for Linux, and offers significant performance improvements over WSL 1. Note that WSL 2 requires Windows 10 version 1903 or higher, with Build 18362 or higher, for x64 systems. WSL 2 is based on a virtual machine with an actual Linux Kernel that can immediately react to system calls. Such an architecture enables Full System call capabilities.

To run OPM Flow under WSL 1 or WSL 2 one has to first activate WSL, then install a Linux distribution from the Application store, and then finally install OPM Flow. The detail instruction are as follows:

1) Activating Windows Subsystem for Linux.

- On the Windows 10 machine, Click Start and navigate to Control Panel, then Select the Programs Category. Under Programs and Features, click on Turn Windows Features on or off and click the check box next to the Windows Subsystem for Linux option, to enable WSL.

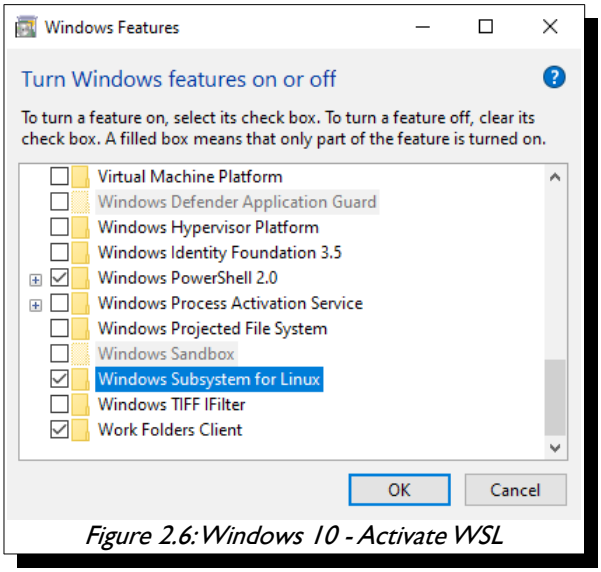


Figure 2.6: Windows 10 - Activate WSL

You may also have to ensure that CPU virtualization. has been activated in your computer's BIOS.

2) Installing a Linux Distribution.

- To install the Linux distribution in Windows 10, click Start and then select the Microsoft Store.
- Using the search box available in the Microsoft Store, pick one of the distributions that is compatible with OPM Flow (Figure 2.7).

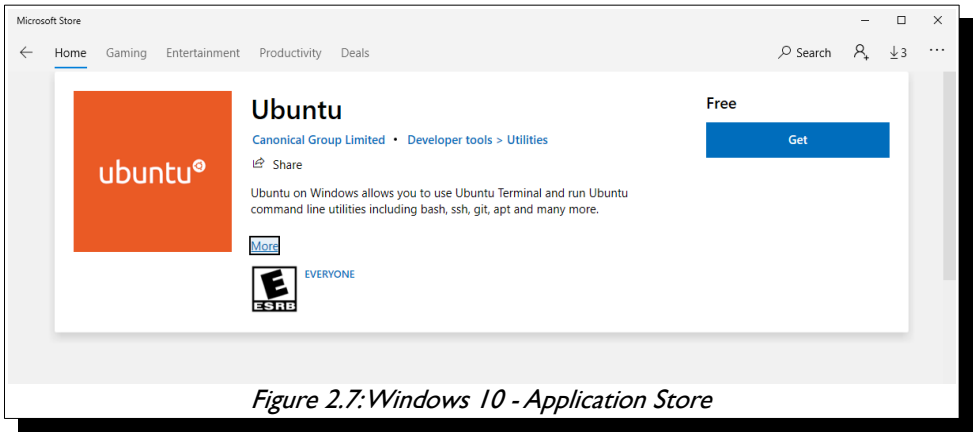


Figure 2.7: Windows 10 - Application Store

- After selecting the distribution, click Install or Get to install the Linux Distribution. Note the installer will ask you to sign into your account, but you can close the pop-up window if you don't want to sign in at this time.
- On completion of the installation process the Linux distribution will be installed on Windows 10 and ready for use. Furthermore, the distribution will be available directly in the Start Menu itself in the form of a standard everyday application (Figure 2.8).

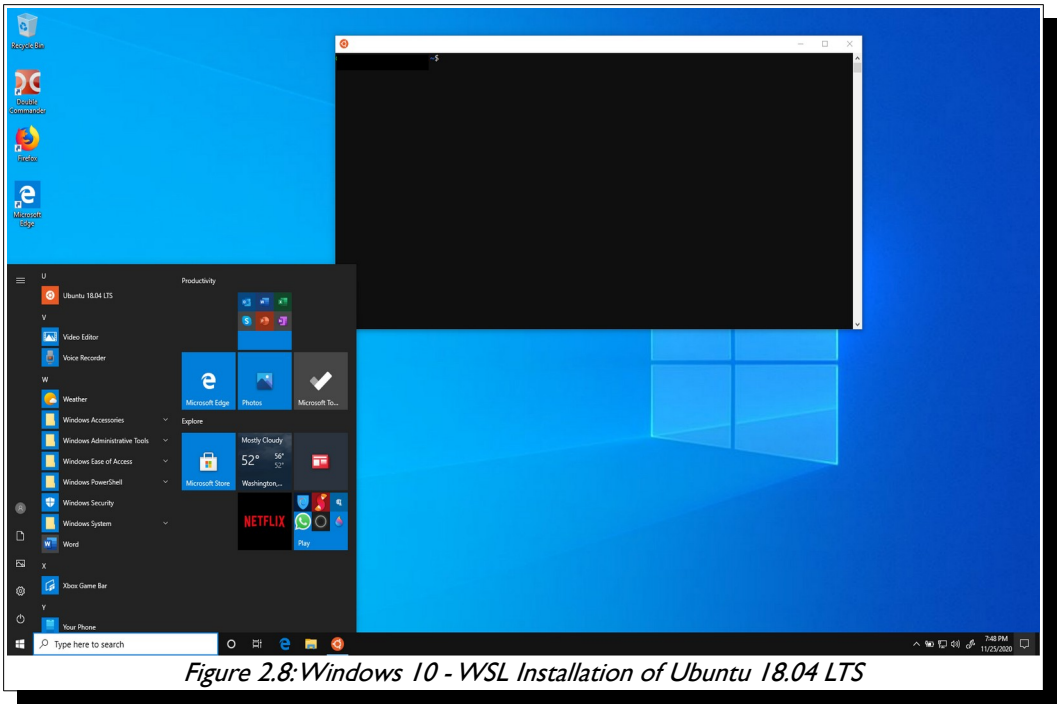


Figure 2.8: Windows 10 - WSL Installation of Ubuntu 18.04 LTS

- When starting the Linux Distribution for the first time, a brief setup process will take place. Basically, it will ask for the username and password you want to set for the Linux environment, a similar process as when you install a Linux distribution via VirtualBox.
- To check the version of WSL that will be used to run the Linux distribution, in PowerShell use the following command:

```
wsl -l -v
```

- To set the WSL for a given distribution, for example Ubuntu-22-04 to WSL 2, in PowerShell use the following command:

```
wsl --set-version Ubuntu-22-04 2
```

- And finally to set the default version for WSL for all Linux distributions, use the following PowerShell command:

```
wsl --set-default-version 2
```

3) Installing OPM Flow in a WSL Linux Distribution:

- Click on installed Linux distribution in the Start Menu to open a Linux terminal session, and then execute the Linux commands as described earlier in Sections *2.1.1 Ubuntu Linux 20.04 (64-bit version only) and Higher* and *2.1.2 Red Hat Enterprise or CentOS (version 6 or 7)* to install the software.

- To check if OPM Flow is installed, execute the following command in the Linux terminal window:

```
flow --help
```

Which should then display the command line for the installed version of OPM Flow, as depicted in Figure 2.9.

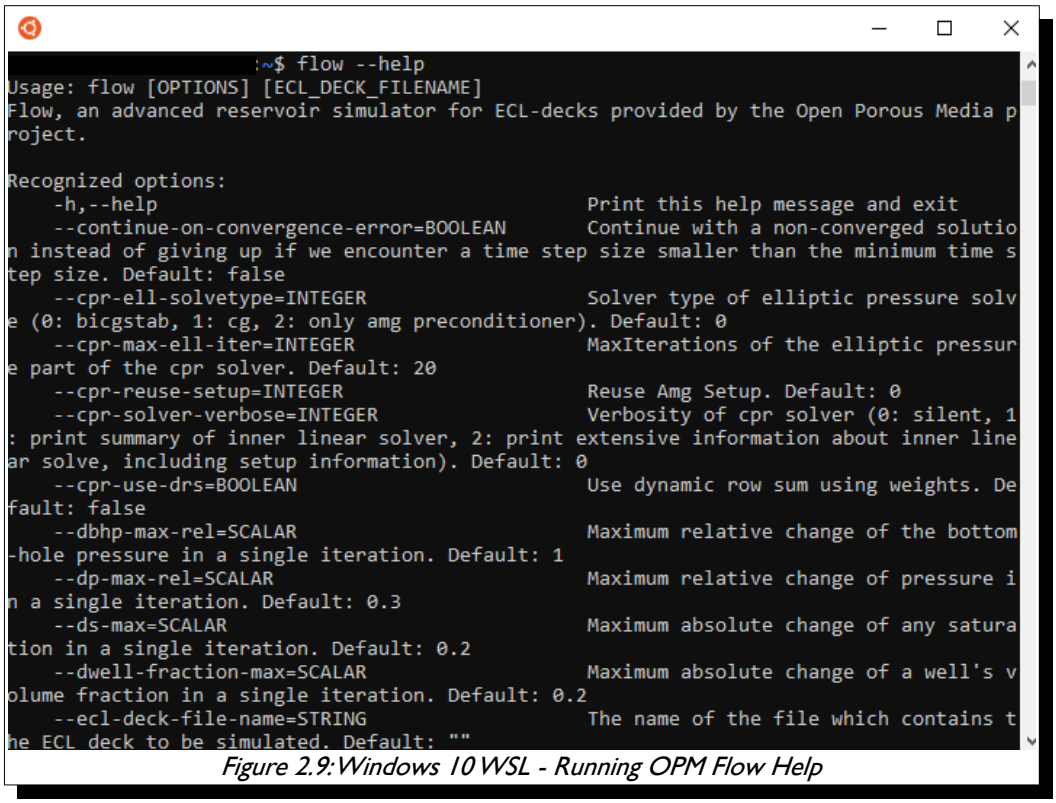


Figure 2.9: Windows 10 WSL - Running OPM Flow Help

One can also use the Windows command terminal, instead of the WSL Linux Terminal, to do the same thing by adding WSL character string before the command, that is:

```
wsl flow --help
```

4) Using OPM Flow via the Windows 10 WSL Linux Virtual Machine.

- Similar to running OPM Flow using a Linux distribution installed in VirtualBox, the WSL Linux virtual machine only has access to files within WSL virtual machine. To access the Windows 10 file system, one has to execute the Linux CD command in the WSL Linux Terminal, the command has the form:

```
cd /mnt/<drive letter>
```

For example, to access the Windows 10 system drive C one would use the following command:

```
cd /mnt/c
```

One can then run OPM Flow from the desired directory in the WSL Linux virtual machine and use OPM ResInsight under Windows 10 to view the results of the simulation run.

User experience indicates this approach of running OPM Flow under Windows 10 is the most integrated and convenient method, compared to the VirtualBox approach, especially for the less technically advance users.

2.1.5 USING A DOCKER CONTAINER

See the tutorial on the OPM website *Running Flow in Docker.*

2.1.6 INSTALLING FROM SOURCE

See instructions on the OPM website *Building from Source*.

2.2 RUNNING OPM FLOW 2023-10 FROM THE COMMAND LINE

This section describes how to run the OPM Flow simulator and the various command line options associated with this release, the command line syntax for running OPM Flow is:

```
flow [OPTIONS] CASENAME
```

and typing the following command from your terminal:

```
flow CASENAME
```

will start the simulator and run the case specified by CASENAME.DATA. Alternatively, you can type the path to the binary and the *.DATA file, as per the example below:

```
path_to_binary/flow path_to_data/CASENAME
```

The simulator can also be run using command line parameters for example to run the previous CASENAME one could use:

```
flow --ecl-deck-file-name=path_to_data/CASENAME
```

Note that there must be no spaces around the equals sign and there should be no spaces in CASENAME. If there are spaces in CASENAME, which can typically occur on Windows based systems, then the CASENAME should be enclosed in quotes; thus instead of CASE NAME use "CASE NAME".

By default output files are generated in the same folder as the *.DATA file. To override this one can use the --output-dir command line parameter, for instance:

```
flow --output-dir=foo CASENAME
```

will send the output files to the foo directory.

It is also possible to put multiple command line options together in a parameter file. To do so, put one option on each line of the file with ".param" as the extension and pass that filename as a command line parameter to OPM Flow using:

```
flow --parameter-file=CASENAME.PARAM
```

to instruct OPM Flow to read the parameter file. Note, it is good practice to name both the *.DATA and parameter file with the same CASENAME, to avoid confusion. However, if one wishes to use the same parameter file for various cases one can use the same parameter file followed by the DATA file, for example:

flow –-parameter-file=path_to_data/CASENAME.param CASENAME.DATA

If OPM Flow is installed with parallel capabilities then:

```
mpirun -np 4 flow --parameter-file=path_to_data/CASENAME.param
```

will start OPM Flow on four nodes etc.

Note

On Red Hat based distributions, including centOS distributions, one needs to install the openmpi version of the simulator, opm-simulators-openmpi-bin, via:

sudo yum install opm-simulators-openmpi-bin

and then set the MPI version to use. The way this is done is by first making the module command available, by running the following command:

. /usr/share/Modules/init/bash

Then query for the installed modules with:

module avail

And lastly, you tell the system it to use openmpi with:

module add mpi/openmpi-x86_64

On Ubuntu based distributions, there is no need for this as openmpi is installed and mpirun just works.

A complete list of the active user facing command line options and their function is given in the Table 2.1 for the current release. These commands are the options listed with command line --help option, there is also a --help-all option that lists all the command line options included in the release, including experimental, obsolete, hidden and deprecated options.

OPM Flow release 2018-10 and beyond now use the eWoms/ebos[35] command line interface; whereas, previous releases used the OPM Flow specific command line parameters and are documented in section *APPENDIX E: RUNNING PREVIOUS RELEASES OF OPM FLOW* for reference.

Additional tutorials for running OPM Flow is available on OPM website in the *Tutorials* section.

OPM Flow 2023-10 Command Line Options

No.	Variable Name	Description	Default
1	-h or --help	A character string that causes OPM Flow to print a help message that gives a brief description of the available command line parameters. **This now only prints the active user facing command line options.**	N/A
2	--help-all	Prints all the command line options included in the release, including obsolete, hidden and deprecated options.	N/A
3	--accelerator-mode	Choose a linear solver, usage: '—accelerator-mode=[none | cusparse | opencl | amgcl | rocalution]'.	"none"
4	--allow-distributed-wells	Allow the perforations of a well to be distributed to interior of multiple processes.	false
5	--alternative-well-rate-init	Use alternative well rate initialization procedure.	true
6	--bda-device-id	Choose device ID for cusparseSolver or	0

[35] *eWorms is a C++ software framework concerned with fully-implicit numerical models for flow and transport in porous media. It it is developed as an integral part of the Open Porous Media (OPM) initiative.*

OPM Flow 2023-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
		openclSolver, use 'nvidia-smi' or 'clinfo' commands to determine valid IDs.	
7	--continue-on-convergence-error	Continue with a non-converged solution instead of giving up if we encounter a time step size smaller than the minimum time step size.	false
8	--cpr-reuse-interval	A positive integer that sets the reuse preconditioner interval. Used when *--cpr-reuse-setup* is set to 4, then the preconditioner will be fully recreated instead of reused every N linear solve, where N is this parameter.	30
9	--cpr-reuse-setup	A positive integer that defines if the CPR solver should re-use the AMG setup. Valid options are: 0 : Recreate the preconditioner for every linear solve. 1 : Recreate once every time step. 2 : Recreate if last linear solve took more than 10 iterations. 3 : Never recreate. 4 : Recreate every N linear solves, where N is the parameter *--cpr-reuse-interval*. Changed the default value from three to four.	4
10	--dbph-max-rel	A real positive value that sets maximum relative change of the bottom-hole pressure in a single iteration	1.0
11	--dp-max-rel	A real positive double precision value that sets the maximum allowed relative change of pressure per iteration.	0.3
12	--ds-max	A real positive double precision value that sets the maximum allowed change in saturation per iteration.	0.2
13	--dwell-fraction-max	A real positive value that sets the maximum allowed change in well's volume fraction per iteration.	0.2
14	--ecl-deck-file-name	A character string that defines the name of the OPM Flow input file which contains the simulator's ECLIPSE formatted input deck to be simulated. Note that there must be no spaces in the filename. If there are spaces, which can typically occur on Windows based systems, then the filename should be enclosed in quotes; thus instead of FILE NAME use	""

OPM Flow 2023-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
		"FILE NAME".	
15	--ecl-enable-drift-compensation	A Boolean value set to true or false that enables (true) or disables (false) partial compensation of systematic mass losses via the source term of the next time step.	true
16	--ecl-enable-tuning	Honor some aspects of the TUNING keyword from the ECL deck.	false
17	--ecl-newton-relaxed-tolerance	The maximum error which the volumetric residual may exhibit if it is in a 'relaxed' region during a strict iteration.	1e+09
18	--ecl-newton-relaxed-volume-fraction	The fraction of the pore volume of the reservoir where the volumetric error may be voilated during strict Newton iterations.	0.03
19	--ecl-newton-strict-iterations	The number of Newton iterations where the volumetric error is considered.	8
20	--ecl-newton-sum-tolerance	The maximum error tolerated by the Newton method for considering a solution to be converged.	0.0001
21	--ecl-newton-sum-tolerance-exponent	The the exponent used to scale the sum tolerance by the total pore volume of the reservoir.	0.333333
22	--ecl-output-double-precision	A Boolean value set to true or false that switches on (true) or off (false) double precision in restart files. Useful for 'perfect' restarts.	false
23	--ecl-output-interval	An integer value defining the number of report steps that ought to be skipped between two writes of restart files results.	-1
24	--edge-weights-method	A defined positive integer value that defines the edge-weighing strategy: 0 : For uniform, 1 : for trans, or 2 : for log(trans).	1
25	--enable-adaptive-time-stepping	A Boolean value set to true or false that turns on (true) or off (false) adaptive time stepping. If set to false the report time steps provided by the input deck are used.	true
26	--enable-async-ecl-output	A Boolean value set to true or false that sets the output mode to be asynchronously (true), letting the simulator continue computing the next time step while writing results to restart and summary files in the commercial simulators format.	true
27	--enable-async-vtk-output	A Boolean value set to true or false that sets the output mode to be asynchronously	true

OPM Flow 2023-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
		(true), letting the simulator continue computing the next time step while writing results to the VTK output files.	
28	--enable-dry-run	A Boolean value set to true or false that specifies if the simulation should actually run, (true) or just check the input deck (false). This option is equivalent to activating the NOSIM keyword in the RUNSPEC section of the input deck (see section *5.3.97 NOSIM – Activate the No Simulation Mode for Data File Checking*). The default value of "auto' will use whatever is stipulated in the input deck via the NOSIM keyword.	"auto"
29	--enable-ecl-output	A Boolean value set to true or false that specifies if the binary output files (restart, summary files, etc.) should be written in the commercial simulator's format (true), or OPM Flow's format (false).	true
30	--enable-esmry	A Boolean value that switches on (true) or off (false) the output SUMMARY vectors to the ESMRY file for fast loading of summary data.	false
31	--enable-gravity	A Boolean value that switches on (true) or off (false) the use of the gravity correction for the pressure gradients.	true
32	--enable-grid-adaptation	A Boolean value that enables (true) or disables (false) adaptive grid refinement/coarsening.	false
33	--enable-intensive-quantity-cache	A Boolean value that switches on (true) or off (false) the caching of intensive quantities.	true
34	--enable-logging-fallout-warning	A Boolean value set to true or false that sets the developer option to see whether logging was on non-root processors (true). If set to true output will be appended to the *.DBG or *.PRT files.	false
35	--enable-opm-rst-file	A Boolean value set to true or false to write OPM specific data sets to the commercial simulators restart file to enable restart of an OPM Flow runs (true) by OPM Flow, or not to write the data (false).	false
36	--enable-storage-cache	A Boolean value set to true or false that turns on (true) or off (false) storing previous storage terms and avoid re-calculating them.	true
37	--enable-terminal-output	A Boolean value set to true or false that	true

OPM Flow 2023-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
		turns on (true) or off (false) high-level information about the simulation's progress to the terminal	
38	--enable-thermodynamic-hints	A Boolean value that enables (true) or disables (false) thermodynamic hints.	false
39	--enable-tuning	A Boolean value set to true or false that instructs OPM Flow to read the time stepping parameters from the *TUNING - Numerical Tuning Control* keyword in the input deck, if set to true. Note that only the first record of the TUNING keyword is processed.	false
40	--enable-vtk-output	A Boolean value set to true or false that turns on (true) or off (false) a global switch for writing VTK files.	false
41	--enable-well-operability-check	A Boolean value set to true or false that enables (true) checking of a well's operating status, or disables (false) the checking.	true
42	--enable-well-operability-check-iter	A Boolean value set to true or false that enables (true) checking of a well's operating status during iterations, or disables (false) the checking during iterations.	false
43	--enable-write-all-solutions	A Boolean value set to true or false that turns on (true) or off (false) the writing of all solutions to disk instead of only the ones for the report steps.	false
44	--end-time	The simulation time at which the simulation is finished [s].	1e+100
45	--force-disable-fluid-in-place-output	A Boolean value set to true or false that instructs OPM Flow not to print the Fluid In-Place report after each report time step (true) or not (false). Note this parameter will override the print request in the input deck.	false
46	--force-disable-resv-fluid-in-place-output	A Boolean value set to true or false that instructs OPM Flow to not to print the Reservoir Volume Fluid In-Place report after each report time step (true) or not (false). Note this parameter will override the print request in the input deck.	false
47	--full-time-step-initially	A Boolean value set to true or false that instructs OPM Flow to always attempt to finish a report step using a single time step (true) or not (false).	false

OPM Flow 2023-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
48	--ignore-keywords	A character string that defines a list of keywords which should be ignored by OPM Flow. The keyword string should be enclosed in quotes and each keyword separated by a colon, that is ':' .	""
	ILU Preconditioner Parameters		
49	--ilu-fillin-level	A positive integer value that sets the fill in level for the ILU preconditioner.	0
50	--ilu-redblack	A Boolean value set to true or false that instructs OPM Flow to use red-black partitioning for the ILU preconditioner. (true) or not (false).	false
51	--ilu-relaxation	A real positive double precision value that sets the relaxation factor of the linear solver's ILU preconditioner	0.9
52	--ilu-reorder-spheres	A Boolean value set to true or false that specifies OPM Flow to reorder the entries of the matrix in the red-black ILU preconditioner in spheres starting at an edge (true) or not (false) If false the original ordering is preserved in each color. Otherwise try to ensure D4 ordering (in a 2D structured grid, the diagonal elements are consecutive).	false
53	--initial-time-step-in-days	A real double precision value that sets the size of initial time step in days.	1.0
54	--initial-time-step-size	A real double precision value that sets the size of initial time step in seconds.	86400
	Linear Solver Parameters		
55	--linear-solver	A defined quoted character string that sets the configuration of the linear solver, valid values are: 1) ilu0 (default), 2) cprw or cpr (an alias for cprw),[36] [37] and [38] 3) cpr_quasiimpes, 4) cpr_trueimpes, 5) amg,[39] 6) hybrid (experimental), or	"ilu0"

[36] *Wallis, J. R., Little, T. E., and Nolen, J. S.: "Constrained Residual Acceleration of Conjugate Residual Methods," paper SPE 13536 presented at the SPE Reservoir Simulation Symposium, Dallas, Texas, USA (February 10-13, 1985).*

[37] *R. Scheichl, M. Roland, J. Wendebourg, Decoupling and block preconditioning for sedimentary basin simulations, Computational Geosciences 7 (2003) 295{318.*

[38] *Klemetsdal, Ø.S., Møyner, O. & Lie, KA. Accelerating multiscale simulation of complex geomodels by use of dynamically adapted basis functions. Comput Geosci 24, 459–476 (2020). https://doi.org/10.1007/s10596-019-9827-z.*

OPM Flow 2023-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
		7) a file name that has the extension ".json", that contains the linear solver configuration parameters. Option (2) extends the existing Constrained Pressure Residual ("CPR") preconditioner to include wells. This option can also be invoked via the CPR keyword in the RUNSPEC section; however, the command line parameter takes precedence. For option (7) one enters a character string enclosed in quotes that defines the filename of a JSON configuration file for the flexible linear solver system. In this case the file extension should be '.json'. Note that the *.PRT file contains the "Property tree for the linear solver" listing, which is the JSON specification of the current case, and can be used to configure a user specific linear solver JSON file. The option "cpr" now is an alias for "cprw" instead of "cpr_trueimpes".	
56	--linear-solver-ignore-convergence-failure	A Boolean value set to true or false that if set to true convergences failures in the linear solver are ignored. This option should be used with care, as the results might be unreliable.	false
57	--linear-solver-max-iter	A positive integer value that defines the maximum number of linear iterations.	200
58	--linear-solver-print-json-definition	Write the JSON definition of the linear solver setup to the DBG file.	true
59	--linear-solver-reduction	A real positive double precision value that sets the minimum reduction of the residual for the linear solver. The linear solver convergences when the residual is reduced sufficiently. **The simulator now overrides the default reduction to be 0.005 instead of 0.01, if the linear solver has been set to one of the cpr options, in a similar manner as how the default maximum number of linear iterations for the cpr and cprw options are changed to 20 instead of 100, unless specified by the command line option by the user.**	0.01
60	--linear-solver-restart	A positive integer value that sets the number of iterations after which GMRES is restarted.	40

[39] *M. Blatt, A parallel algebraic multigrid method for elliptic problems with highly discontinuous coefficients, Ph.D. thesis, Ruprecht-Karls-Universität Heidelberg (2010).*

OPM Flow 2023-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
61	--linear-solver-verbosity	A positive integer value that defines the output from linear solver: 0 : No extra output, 1 : output per solution iteration, or 2 : output per iteration.	0
62	--load-file	A character string enclosed in quotes that defines the FileName for the .OPMRST file used to load the serialized state. If empty, CASENAME.OPMRST is used.	""
63	--load-step	An integer value that determines if the simulator should load the serialized state from OPM Flow's version of the restart file. and should be set to either a specific report step, or 0 to load the last stored report step. The default value of -1 does not load the data from the OPM Flow specific restart file. OPM Flow's version of the restart file, is written using the *--save-step=N* option.	-1
64	--local-domains-ordering-measure	A character string enclosed in quotes that defines the subdomain ordering measure. Allowed values are 'pressure' and 'residual'.	"pressure"
65	--local-domains-partitioning-imbalance	A real positive double precision value that sets the subdomain partitioning imbalance tolerance. 1.03 is 3 percent imbalance.	1.03
66	--local-domains-partitioning-method	A character string enclosed in quotes that defines the subdomain partitioning method. Allowed values are 'zoltan', 'simple', and the name of a partition file ending with '.partition'.	"zoltan"
67	--local-solve-approach	A character string enclosed in quotes that defines the local solve approach. Valid choices are 'jacobi' and 'gauss-seidel'.	"jacobi"
68	--local-tolerance-scaling-cnv	A real positive double precision value that sets the convergence tolerance for local solves. Set to lower than 1.0 to use stricter convergence tolerance for local solves.	0.01
69	--local-tolerance-scaling-mb	A real positive double precision value that sets the convergence tolerance for local solves. Set to lower than 1.0 to use stricter convergence tolerance for local solves.	1
70	--local-well-solve-control-switching	A boolean value that allows (true) or disallows (false) control switching during local well solutions.	false
71	--matrix-add-well-contributions	A Boolean value set to true or false that if set to true explicitly specifies the influences	false

OPM Flow 2023-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
		of wells between cells in the Jacobian and preconditioner matrices.	
72	--max-inner-iter-ms-wells	A positive integer value that defines the maximum number of inner iterations for multi-segment wells.	100
73	--max-inner-iter-wells	A positive integer value that defines the maximum number of inner iterations for standard wells.	50
74	--max-local-solve-iterations	A positive integer value that defines the maximum number of iterations for local solves with NLDD nonlinear solver.	20
75	--max-newton-iterations-with-inner-well-iterations	A positive integer that specifies the maximum newton iterations with inner well iterations.	8
76	--max-pressure-change-ms-wells	A real positive value that defines the maximum relative pressure change for a single iteration of the multi-segment well model.	1.0×10^6
77	--max-residual-allowed	A real positive value that sets the absolute maximum tolerance for residuals without cutting the time step size.	1.0×10^7
78	--max-single-precision-days	A real positive value that sets the maximum time step size where single precision floating point arithmetic can be used for solving the linear systems of equations.	20.0
79	--max-temperature-change	A real positive value that stipulates the maximum absolute change of temperature in a single iteration.	5
80	--max-time-step-divisions	A positive integer that specifies the maximum number of divisions by two of the timestep size before the simulation bails out.	10
81	--max-time-step-size	A real positive value that defines the maximum size to which all time steps are limited to in seconds.	inf
82	--max-welleq-iter	A positive integer that defines the maximum number of iterations to determine the solution to the well equations.	30
83	--maximum-number-of-well-switches	A positive integer values that stipulates the maximum number of times a well can switch to the same control.	3
84	--maximum-water-saturation	A real positive value that defines the maximum water saturation.	1.0
85	--milu-variant	A defined character string that specifies	"ILU"

OPM Flow 2023-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
		which variant of the modified ILU preconditioner ought to be used. Possible variants are: 1) ILU (default, plain ILU), 2) MILU_1 (lump diagonal with dropped row entries), 3) MILU_2 (lump diagonal with the sum of the absolute values of the dropped row entries), 4) MILU_3 (if diagonal is positive add sum of dropped row entries. Otherwise subtract them), 5) MILU_4 (if diagonal is positive add sum of dropped row entries. Otherwise do nothing). The default is "ILU"	
86	--min-strict-cnv-ite	A positive integer that sets the minimum number of Newton iterations before relaxed tolerances are used for the CNV convergence criterion.	0
87	--min-time-step-based-on-newton-iterations	A real positive value that sets the minimum time step size (in days for field and metric unit and hours for lab unit) can be reduced to based on Newton iteration counts.	0.0
88	--min-time-step-before-shutting-problematic-wells-in-days	A real positive value that sets the minimum time step size in days for which problematic wells are not shut. Time steps below this value will result in problematic wells being shut-in by the simulator.	0.01
89	--min-time-step-size	A real positive value that sets the minimum size to which all time steps are limited to in seconds.	0.0
90	--network-max-iterations	A positive integer that sets the maximum number of iterations in the network solver before giving up.	200
91	--network-max-strict-iterations	A positive integer that sets the maximum iterations in network solver before relaxing tolerance.	100
92	--newton-max-error	A real positive value that sets the maximum error tolerated by the Newton method to which does not cause an abort.	1e+10
	Newton Solver Parameters		
93	--newton-max-iterations	A positive integer that defines the maximum number of Newton iterations per time step used by the simulator.	20
94	--newton-max-relax	A real positive value that sets the maximum	0.5

OPM Flow 2023-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
		relaxation factor of a Newton iteration used by the simulator.	
95	--newton-min-iterations	A positive integer that sets the minimum number of Newton iterations per time step used by the simulator. The default value of one ensures that at least one Newton iteration is performed after the previous time step.	1
96	--newton-relaxation-type	A character string that defines the type of relaxation used in Newton's method, the default is dampen.	"dampen"
97	--newton-target-iterations	A positive integer that sets the 'optimum' number of Newton iterations per time step.	6
98	--newton-tolerance	A real positive value that sets the maximum raw error tolerated by the Newtonmethod for considering a solution to be converged.	0.01
99	--newton-verbose	A boolean value that specifies whether the Newton method should inform the user about its progress (true) or not (false).	true
100	--newton-write-convergence	A boolean value that specifies whether to write the convergence behaviour of the Newton method to a VTK file (true) or not (false).	false
101	--nldd-num-initial-newton-iter	A positive integer that sets the number of initial global Newton iterations when running the NLDD nonlinear solver.	1
102	--nonlinear-solver	A character string that specifies the nonlinear solver. Valid choices are newton or nldd.	"newton"
103	--num-local-domains	A positive integer that sets the number of local domains for NLDD nonlinear solver. Note this is an experimental feature in the current release 2023.10 that is expected to be more complete and tested by the 2024.10 release.	0
104	--num-pressure-points-equil	A positive integer that sets the number of pressure points (in each direction) in tables used for equilibration.	2000
105	--opencl-ilu-parallel	A Boolean value set to true or false that if set to true then parallelize the ILU decomposition and application on GPU, or not (false).	true
106	--opencl-platform-id	A positive integer that specifies the platform identification ("ID") for the	0

OPM Flow 2023-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
		openSolver. Use the "clinfo" command to determine valid IDs.	
107	--output-dir	A character string that defines the directory to which OPM Flow is to write the commercial simulator compatible output files (restart, summary files etc.). The default value results in the files be written to the same directory as the input file.	""
108	--output-extra-convergence-info	A defined character string that specifies whether to provide additional convergence output to separate files for diagnostic purposes. The available options are: 1) "none" results in no extra output and overrides all other options. 2) "steps" writes out convergence information per time step, to a CASENAME.INFOSTEP file. The file is useful for identifying numerical issues. 3) "iterations" writes out non-linear convergence metrics, i.e., the MB and CNV values, per phase, for each non-linear iteration in each time step, to a CASENAME.*INFOITER* file. Options can be combined with commas, e.g."steps,iterations" for multiple outputs. The default value of "none" prevents the two files from being written out, for better compatibility with the commercial simulator.	"none"
109	--output-interval	A positive integer that specifies the number of report steps between two consecutive writes of restart data.	1

110	--output-mode	A defined character string that defines the output to the *.PRT and *.DBG files:
		1) "none": No output to the files.
		2) "log" or false: Output logging information only.
		3) "all" or true: Output everything.
		For example to just output logging information use:
		`--output-mode="log"`
		or
		`--output-mode=false`
			"all"

111	--owner-cells-first	A Boolean value set to true or false that determines if the cells owned by a rank should be ordered before ghost/overlapping the cells (true), or not (false).	true

OPM Flow 2023-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
112	--parameter-file	A character string that defines the name of a parameter file which contains the simulator's set of run-time parameters, as listed in this table.	""
113	--parsing-strictness	Set strictness of parsing process. Available options are: 1) "normal": stop for critical errors, and for unsupported keywords that would would change the simulator results if supported. 2) "high": stop for all errors, that is even for unsupported keywords that do not effect the results, for example ECHO and NOECHO. 3) "low": same as normal, except do not stop due to unsupported keywords that would change the simulator results if supported, and even if marked critical. Default: "normal"	"normal"
114	--predetermined-time-steps-file	A file with a list of predetermined time step sizes (one time step per line).	""
115	--pri-var-oscilation-threshold	A real positive value that defines the threshold value for the primary variable switching conditions after its meaning has switched to hinder oscillations.	1.0 x 10-5
116	--print-parameters	A positive integer value that request that the run time parameters be printed at the start of the run: 0 : No output to the files. 1 : Output *.DBG file. 2 : Output to *.DBG and *.PRT files (default).	2
117	--print-properties	A positive integer value that request that the compile time parameters be printed at the start of the run: 0 : No output to the files. 1 : Output *.DBG file. 2 : Output to *.DBG and *.PRT files (default).	2
118	--project-saturations	A Boolean value set to true or false that determines if all the fluid saturations should be scaled to ensure the values are in the interval (0, 1), including runs that use solvents (true), or not (false).	false
119	--regularization-factor-wells	A real positive value that defines the "regularization factor" for wells.	100

OPM Flow 2023-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
120	--relaxed-linear-solver-reduction	A real positive value that defines the minimum reduction of the residual which the linear solver need to achieve for the solution to be accepted.	0.01
121	--relaxed-max-pv-fraction	A real positive value that defines The fraction of the pore volume of the reservoir where the volumetric error (CNV) might be violated during strict Newton iterations.	0.03
122	--relaxed-pressure-tol-msw	A real positive value that sets the relaxation tolerance for the multi-segment well pressure solution in Pascals.	10000
123	--relaxed-well-flow-tol	A real positive value that sets the relaxation tolerance for the well flow residual in reservoir cubic metres (rm3).	0.001
124	--restart-time	A real positive value that sets the simulation time at which a restart should be attempted [s].	-1e+35
125	--restart-writing-interval	A positive integer value that sets the frequency at which time steps are serialized to disk.	16777215
126	--save-file	A character string that specifies the FileName for .OPMRST file used for saving serialized state. If empty, CASENAME.OPMRST is used.	""
127	--save-step	A character string that determines if the simulator should save the serialized state of the OPM Flow simulator at one or more report steps to a special *.OPMRST file. This is in addition to the normal restart files written, and consumes significantly more space than the normal restart files, but restarting OPM Flow from this file using the *--load-step* option deviates less from the original run, compared to restarting using the normal restart features. The files produced are not compatible with other simulators, and also will not be compatible between different releases of OPM Flow The parameter should be set to one of the following: 1) "all" to save all report steps, 2) ":x" to save every x'th step, or 3) "x" to save a specif time step. The default value of "" does not write anything to the OPM Flow specific restart file.	""
128	--scale-linear-system	A Boolean value set to true or false that	false

OPM Flow 2023-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
		switches on (true) or off (false) the scaling of linear system of equations according to the equation scale and primary variable types.	
129	--sched-restart	A Boolean value that determines for a restart run if the case should initialize wells and groups from the historical SCHEDULE section (true), or from the well and group data on the restart file (false). Note that the commercial simulator always uses data from the restart file (false).	falsch
130	--serial-partitioning	A Boolean value that determines if partitioning for parallel runs on a single process (true), or not (false).	false
131	--shut-unsolvable-wells	A Boolean value that determines if the simulator should shut unsolvable wells (true), or not (false).	true
132	--solve-welleq-initially	A Boolean value set to true or false that determines if the simulator should fully solve the well equations before each iteration of the reservoir model (true), or not (false). Note that the well equations are always added to the full system and solved until converged.	true
	Solver Parameters		
133	--solver-continue-on-convergence-failure	A Boolean value that stipulates if the simulator should continue (true) instead of stopping (false) when the minimum solver time step is reached.	false
134	--solver-growth-factor	A real positive value that specifies the growth factor a time step can be increased by when recovering from one or more time step chops, subject to the maximum allowable time step size set by the *--solver-max-time-step-in-days* parameter. For example, if the current time step has converged at 5 days after a time step chop, and --solver-growth-factor is set to the default value of 2.0, then the next time step will be 2.0 * 5 days, that is at 10 days.	2.0
135	--solver-max-growth	A real positive value that specifies the maximum growth factor a time step can be increased by after a report time step. subject to the maximum allowable time step size set by the --solver-max-time-step-in-days parameter. Thus, if the current time step has converged at 5 days after at a report step,	3.0

OPM Flow 2023-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
		and --solver-*max*-growth is set to the default value of 3.0, then the next time step will be 3.0 * 5 days, that is at 15 days.	
136	--solver-max-restarts	A positive integer that defines the number of allowed consecutive restarts (or time step chops) before the simulation is terminated.	10
137	--solver-max-time-step-in-days	A real positive double precision value that specifies the maximum allowed time step size in days.	365
138	--solver-min-time-step	A real positive double precision value that specifies the minimum size of a time step in days for field and metric units, and hours for laboratory units If a time step cannot converge without getting cut below this time step size the simulator will stop.	1.0×10^{-12}
139	--solver-restart-factor	A real positive double precision value that sets the time step chop factor of the time step after a convergence failure. For example, if the current non-convergent time step is 30 days and *--solver-restart-factor* is set to the default value, then the time step will be repeated using 0.33 * 30 days as the time step, that is 9.9 days.	0.33
140	--solver-verbosity	A positive integer that specifies the "chattiness" of the non-linear solver.	1
141	--strict-inner-iter-wells	A positive integer that specifies the number of inner well iterations with strict tolerance.	40
142	--strict-outer-iter-wells	A positive integer that specifies the number of newton iterations for which wells are checked with strict tolerance.	6
143	--temperature-max	A real positive value that sets the maximum absolute temperature.	1.0×10^{9}
144	--temperature-min	A real positive value that sets the minimum absolute temperature.	0
145	--threads-per-process	A positive integer value that stipulates the maximum number of threads to be instantiated per process ('-1' means 'automatic').	-1
	Time Stepping Control Algorithm Parameters		
146	--time-step-after-event-in-days	A real double precision value that sets maximum allowed time step after an event; for example, when a well is open or closed etc.	-1

OPM Flow 2023-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
		The default value of -1 means that events to do effect the time stepping.	
147	--time-step-control	A defined character string that defines the time stepping control algorithm and is set to one of the following: 1) pid: PID controller based adaptive time step control as suggested by Turek and Kuzmin[40]. 2) pid+iteration: Use PID and linear iteration numbers to guide the time step. 3) pid+newtoniteration: Use PID and non-linear iterations numbers to guide the time step. 4) Hardcoded: Use time steps supplied by user via the time-step-control-filename parameter.	"pid+newtoniteration"
148	--time-step-control-decay-damping-factor	A real positive value that specifies the decay rate a time step can be decreased by after the number target iterations has been exceeded	1.0
149	--time-step-control-decay-rate	A real positive value that specifies the decay rate a time step size can be decreased by after the number of target iterations has been exceeded	0.75
150	--time-step-control-file-name	A character string enclosed in quotes that specifies a filename where time steps are specified. The default is the character string timesteps For instance time steps can be generated by the ecl_summary application in libecl as per the following UNIX command line: `path_to_libecl_applications/ ecl_summary DECK TIME > filename` Where: DECK is the name of the data deck you want to get the time steps from, TIME tells the application to return the timing for the run, and "filename" is the name of the file the times are piped to.	"timesteps"
151	--time-step-control-growth-damping-factor	A real positive value that specifies the growth rate of the time step increase when the number of target iterations are undercut.	3.2

[40] *Algebraic Flux Correction III. Incompressible Flow Problems. Uni Dortmund, Turek and Kuzmin, January, 2006 (DOI: 10.1007/3-540-27206-2_8)*

OPM Flow 2023-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
152	--time-step-control-growth-rate	A real positive value that specifies the growth rate of the time step size when the number of target iterations are undercut.	1.25
153	--time-step-control-target-iterations	A positive integer that defines the number of linear iterations which the time step control scheme should aim for (if applicable).	30
154	--time-step-control-target-newton-iterations	A positive integer that specifies the number of Newton iterations which the time step control scheme should aim for (if applicable).	8
155	--time-step-control-tolerance	A real double precision value that sets the tolerance for PID (only used with the pid and pid+ options defined by the --time-step-control option).	0.1
156	--time-step-verbosity	A positive integer that specifies the "chattiness" during the time integration.	1
	Convergence and Material Balance Tolerance Parameters		
157	--tolerance-cnv	A real positive double precision value that specifies the maximum non-linear tolerance error. This is the local convergence tolerance (maximum of local saturation errors).	0.01
158	--tolerance-cnv-relaxed	A real positive value that defines the relaxed local convergence tolerance that applies for iterations after the iterations with the strict tolerance.	1.0
159	--tolerance-mb	A real positive double precision value that sets the maximum mass balance error, that is the tolerated mass balance error relative to total mass present.	1.0 x 10-6
160	--tolerance-pressure-ms-wells	A real positive double precision value that specifies the tolerance for the pressure equations for multi-segment wells.	1000
161	--tolerance-well-control	A real positive double precision value that sets the maximum tolerance for the well control equations.	1.0×10^{-7}
162	--tolerance-wells	A real positive double precision value that defines the maximum non-linear error for the well equations.	1.0×10^{-4}
163	-update-equations-scaling	A Boolean value that switches on (true) or off (false) the updating of the scaling factors for mass balance equations during the simulation.	false

OPM Flow 2023-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
164	--use-average-density-ms-wells	A Boolean value that specifies whether to approximate segment densitities by averaging over the segment and its outlet (true) or not (false).	false
165	--use-gmres	A Boolean value that when set to true OPM Flow will use Generalized Minimal Residual ("GMRES")[41] and [42] solver instead of Biconjugate Gradient Stabilized ("BiCGSTAB")[43] and [44] as the linear solver within the Newton iterations.	false
166	--use-multisegment-well	A Boolean value that when set to true the simulator will use the well model for multi-segment wells instead of the one for single-segment wells.	true
167	--use-update-stabilization	A Boolean value set to true or false that switches on (true) or off (false) the stabilized Newton option, that attempts to detect and correct oscillations or stagnation during the Newton iterations. This option may improve convergence for some cases.	true
168	--water-only-threshold	A real positive value that defines the saturation threshold, for which cells with water saturations above or equal to this threshold are considered one-phase water only.	1.0
169	--zoltan-imbalance-tol	A real positive value that defines the tolerable imbalance of the load balancing provided by Zoltan package.	1.1
170	--zoltan-params	A character string that specifies the configuration of the Zoltan partitioner. Valid options are: graph, hypergraph or scotch. Alternatively, you can request a configuration to be read from a JSON file by giving the filename here, ending with '.json.' See https://sandialabs.github.io/Zoltan/ug_html/ug.html for available Zoltan options.	"graph"
Notes: 1) Cells colored green in the No. column are new command line parameters for this release. Similarly for the			

[41] *Y. Saad, A flexible inner-outer preconditioned GMRES algorithm, SIAM J. Sci. Statist. Comput., 14, (1993).*

[42] *Y. Saad and M.H. Schultz, GMRES: A generalized minimum residual algorithm for solving nonsymmetric linear systems, SIAM J. Sci. Statist. Comput., 7 (1986), pp. 856{869.*

[43] *Van der Vorst, H. A. (1992). "Bi-CGSTAB: A Fast and Smoothly Converging Variant of Bi-CG for the Solution of Nonsymmetric Linear Systems". SIAM J. Sci. Stat. Comput. 13 (2): 631–644. doi:10.1137/0913035. hdl:10338.dmlcz/104566*

[44] *Sleijpen, G. L. G.; Fokkema, D. R. (November 1993). "BiCGstab(l) for linear equations involving unsymmetric matrices with complex spectrum" (PDF). Electronic Transactions on Numerical Analysis. Kent, OH: Kent State University. 1: 11–32. ISSN 1068-9613.*

OPM Flow 2023-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**

Default column, cells colored green indicate the default value has changed from the previous release.

2) Items shaded in gray are considered to be developer options that should be used with caution as the values associated with these command line parameters are subject to change.

3) VTK Graphics Command Line Parameters[45] set of commands are no longer listed via the basic help command line command:

```
flow --help
```

instead use:

```
flow --help-all
```

to get a list of supported command line Parameters.

4) The –enable-vtk-output option above, if set to "true" will write out the data as stipulated by the compile options. One may need to compile the source code to obtain the desired output.

5) As per all UNIX and LINUX based system the input is case dependent.

6) If a parameter file is used to set one of the above parameters, the leading "--" should not be incorporated in the parameter file.

Table 2.1: OPM Flow 2023-10 Command Line Options

As per all UNIX and LINUX based systems the input is case dependent. The command line parameters must either be entered as lower case or CamelCase, for example:

```
flow --enable-dry-run=false CASENAME.DATA
```

or:

```
flow --EnableDryRun=false CASENAME.DATA
```

If a parameter file is used to set one of the above parameters, the leading "--" should not be incorporated in the parameter file, for example on the command line use

```
flow --enable-dry-run=false CASENAME.DATA
```

but in the parameter file use:

```
ecl-deck-file-name=CASENAME.DATA

enable-dry-run=false
```

or:

```
EclDeckFileName=CASENAME.DATA

EnableDryRun=false
```

OPM Flow prints out the command line run time and compile time parameters in CamelCase in both the *.PRT and *.DBG files for reference, one can therefore use this as basis for creating additional run specific parameter files.

Note

It is worth considering having a parameter file for each run of the form CASENAME.param, in order to re-run a case and to use the same parameters for other cases.

[45] *Virtualization ToolKit ("VTK") files used to visualize the 3D results from the model using ParaView, an open-source, multi-platform general purpose data analysis and visualization application (https://www.paraview.org/overview/). For the commercial simulator's compatible output files OPM's ResInsight 3D visualization package can be used instead and is specifically tailored for OPM Flow and the commercial simulator.*

Example

The following example is taken for the Norne project and the comments in the example (preceded with "/#") explain the parameter setting used in the file.

```
# ------------------------------------------------------------------------
# INPUT AND OUTPUT OPTIONS
# ------------------------------------------------------------------------
#
# Input File
#
ecl-deck-file-name=NORNE_ATW2013.DATA
#
# Output and Output Directory
#
output-dir='\$HOME\OPM\NORNE\'
# ------------------------------------------------------------------------
# NEWTON SOLVER PARAMETER
# ------------------------------------------------------------------------
#
# Define Numerical Tolerances
#
tolerance-cnv=1e-2
tolerance-mb=1e-5
tolerance-wells=1e-2
#
# Set Min Newton Solver iterations to 1 and Max to 15
#
newton-min-iterations=1
newton-max-iterations=15
#
# ------------------------------------------------------------------------
```

Notice that the leading "--" have not be incorporated in the parameter file, as per the notes in Table 2.1.

In order to use the above parameter file called one would use the following format:

```
flow --parameter-file=CASENAME.PARAM
```

If the above parameter file was called NORNE_ATW2013.PARAM, then the command would be:

```
flow --parameter-file=NORNE_ATW2013.PARAM
```

or:

```
flow --ParameterFile=NORNE_ATW2013.PARAM
```

2.3 RUNNING OPM FLOW USING OPMRUN

This section briefly describes how to run the OPM Flow simulator using the Graphical User Interface ("GUI") program called OPMRUN, a more detailed description is given in *APPENDIX C: OPMRUN – FLOW JOB SCHEDULER*, that also describes the various tools included with OPMRUN. The program is written in Python 3 and has been tested under Ubuntu-Mate 20.04 TLS. The software can be downloaded from the following link:

https://github.com/OPM/opm-utilities/tree/master/opmrun

The intent is to develop a graphical user interface to OPM Flow that has similar functionality to the commercial simulator's program, with the targeted audience being Reservoir Engineers in a production environment. Developers and experienced Linux users will already have compatible work flows. OPMRUN enables the editing and management of OPM Flow's run time parameters, setting up job queues to run a series of simulation jobs sequentially, as well as the management of the job queues. Figure 2.10 shows the initial display.

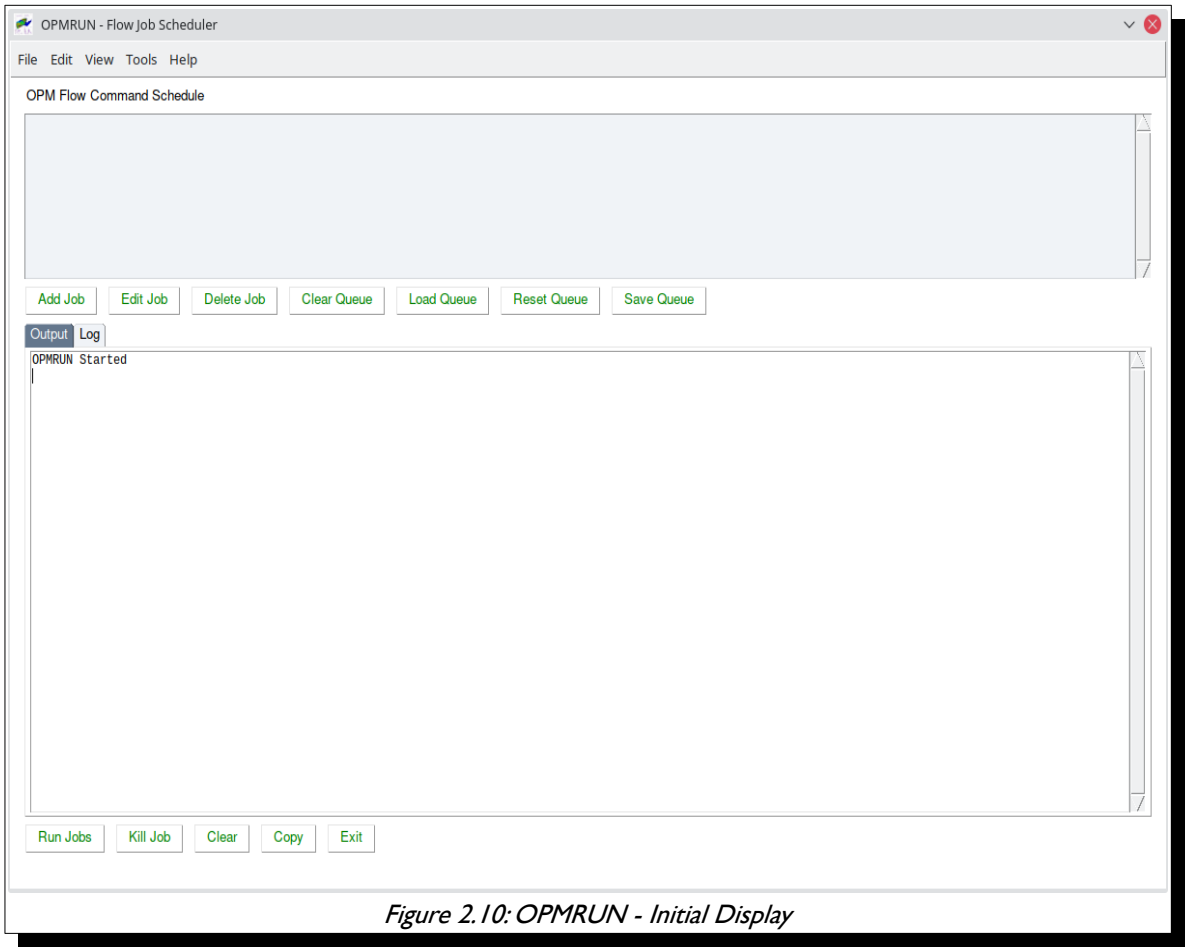


Figure 2.10: OPMRUN - Initial Display

Upon launch the program runs OPM Flow to get a list of command line parameters from the current version of OPM Flow. These default parameters can be edited for each case, or alternative default parameter sets can be loaded from an existing parameter file from another job, or a *.PRT file from a completed simulation.

As can be seen in Figure 2.11 the program has upper and lower display elements. The upper element shows a list of simulation jobs that are in the job queue and the lower element consists of two elements, one for the OPM Flow Output (the terminal output from OPM Flow) and a second element (OPM Run Log) that is a session log of the jobs run by OPMRUN. Clicking the OPM Flow Output and OPM Run Log tabs switches the display on the lowered element between two display types.

Figure 2.11: OPMRUN - Display Elements

To add jobs to the queue use the Add Job button or load an existing job queue using the Load Queue button. Jobs can be edited or deleted from the queue using the Edit Job and Delete Job buttons, and a series of jobs can be saved as a job queue by using the Save Queue button. The Clear Queue button deletes all jobs from the queue.

Pressing the Add Job button will display the following dialogue box:

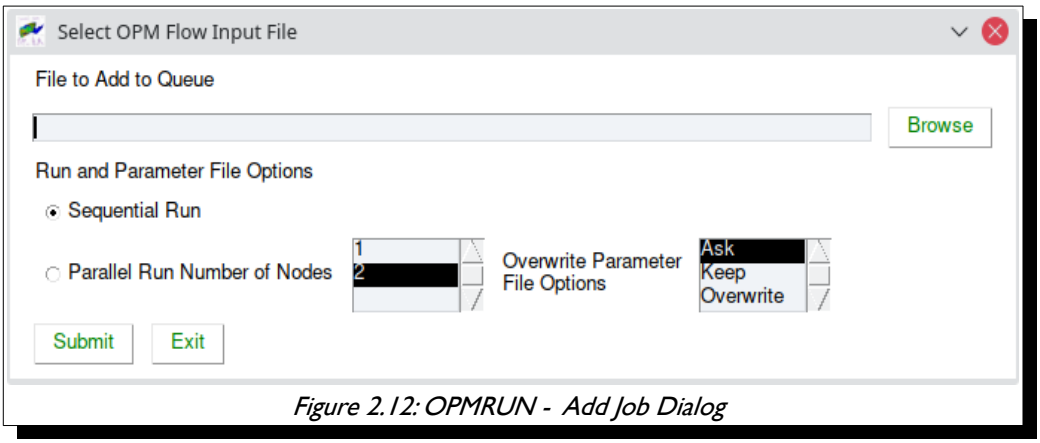


Figure 2.12: OPMRUN - Add Job Dialog

Use the *Browse* button to select the input file to add to the queue, then select the Run Parameters for this input file, then press the Submit button to add the input file to the job queue.

The Overwrite Parameter File Options allow for different default treatments of existing *.PARAM files, which is particularly useful when adding multiple jobs at the same time. The reason for this is because different versions of OPM Flow have different parameter sets and if a newer version of OPM Flow runs with a previous version's *.PARAM file then the simulator will stop with an error message for the various invalid

parameters for the current version of the simulator. To load a previously saved job queue, press the *Load Queue* button this will display a dialog box allowing the user to select a queue file (*.que), after pressing the *OK* button the jobs will be displayed in the Job List Element as illustrated in Figure 2.13.

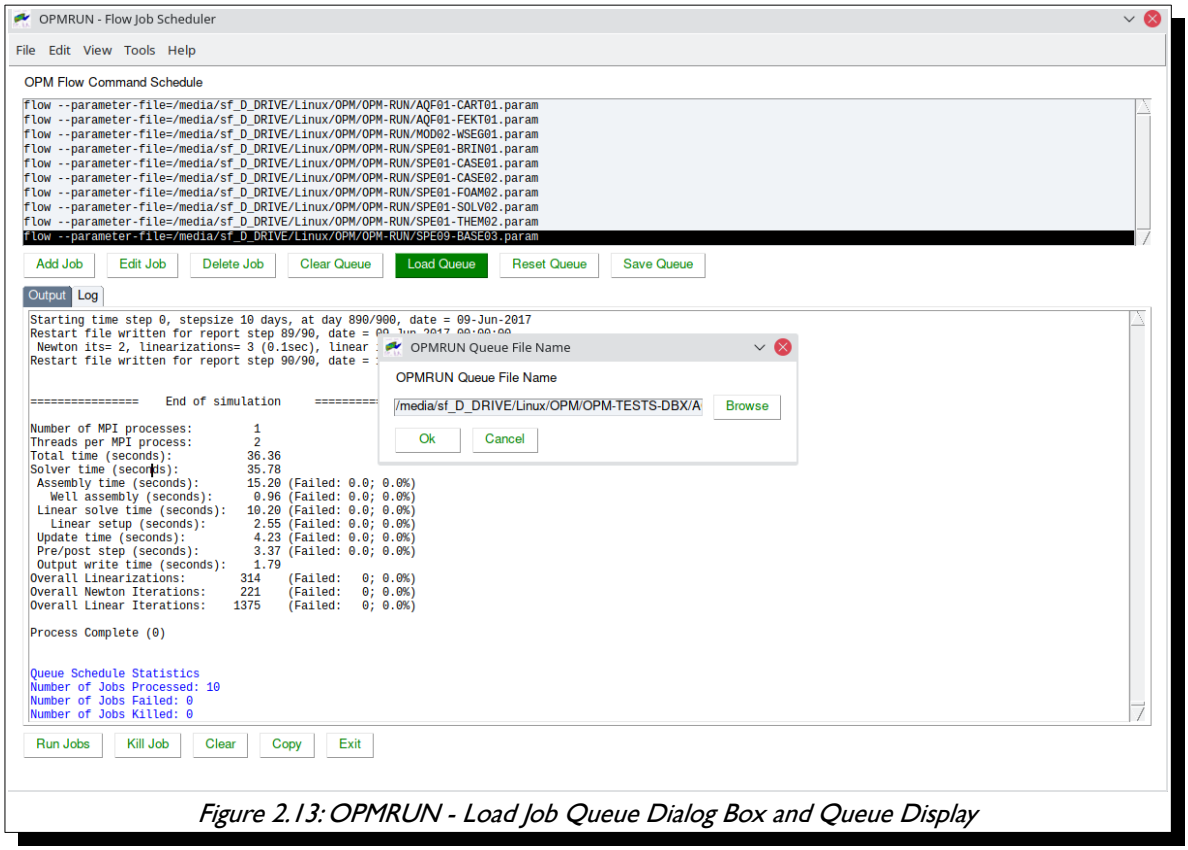


Figure 2.13: OPMRUN - Load Job Queue Dialog Box and Queue Display

Jobs in the queue can be edited by selecting the *Edit Job* button that will display two options: one to edit the input file using the defined editor and the second to edit the OPM Flow Parameter File. If the second option is selected OPMRUN will display a dialog box that shows a list of the OPM Flow command line parameters together with the parameter help information (Figure 2.14). Selecting a parameter from the list and selecting the Edit button will display the setting for the selected parameter (alternatively one can double click the required entry). One can then edit the parameter as required. Use the Save button to save the change and use the Exit but to save all the changes to the parameter file. The Cancel button will cancel all changes to the parameter file.

Alternatively one can use the:

1) Edit OPM Flow Parameter menu option to edit the parameter file for a job.
2) List OPM Flow Parameters menu option to list the commands in the parameter file for a job.
3) Set OPM Flow Default Parameters to set the default parameters for all subsequent jobs added to the queue. This option allows the user to load a default set of parameters from (1) OPM Flow, (2) an OPM Flow Parameter File, or (3) an OPM Flow print file (*.PRT).

One can also right-click on a job in the *Job List Element* and select one of the available options.

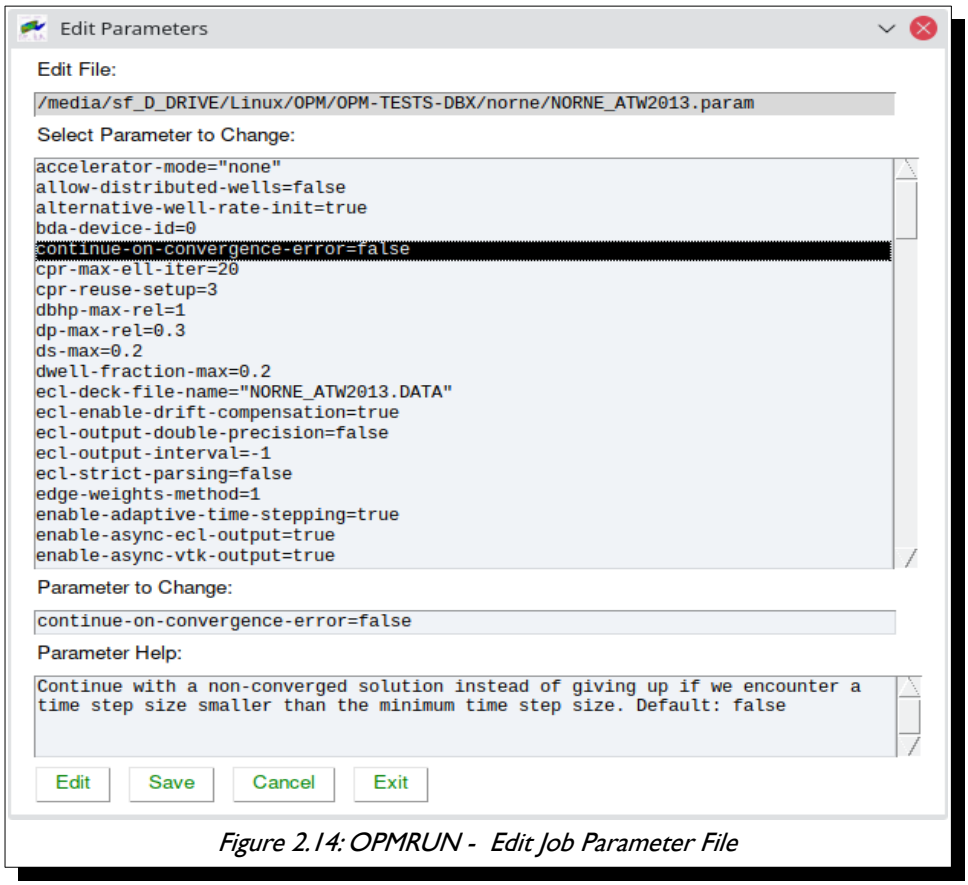


Figure 2.14: OPMRUN - Edit Job Parameter File

Selecting the *Run Jobs* button displays the Select Run Option dialog box shown in Figure 2.15.

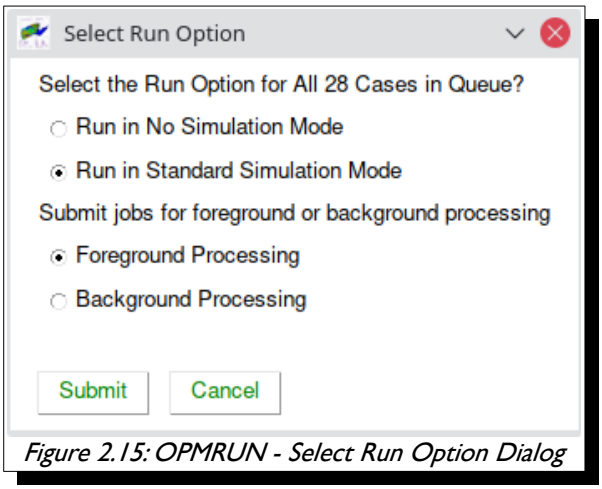


Figure 2.15: OPMRUN - Select Run Option Dialog

The Run in No Simulation Mode option is equivalent to setting the NOSIM option in the input deck for all jobs in the queue (see see section *5.3.97 NOSIM – Activate the No Simulation Mode for Data File Checking* and the –enable-dry-run command line parameter in Table 2.1 in section *2.2 Running OPM Flow 2019-04 From The Command Line*. This allows for checking all the jobs at once.

Selecting Run in Standard Simulation Mode will run all the jobs in the queue sequentially, with the OPM Flow terminal output directed to OPM Flow Output Element, as shown in Figure 2.16. The terminal output is also directed to a *.LOG file as well, similar to what the commercial simulator does.

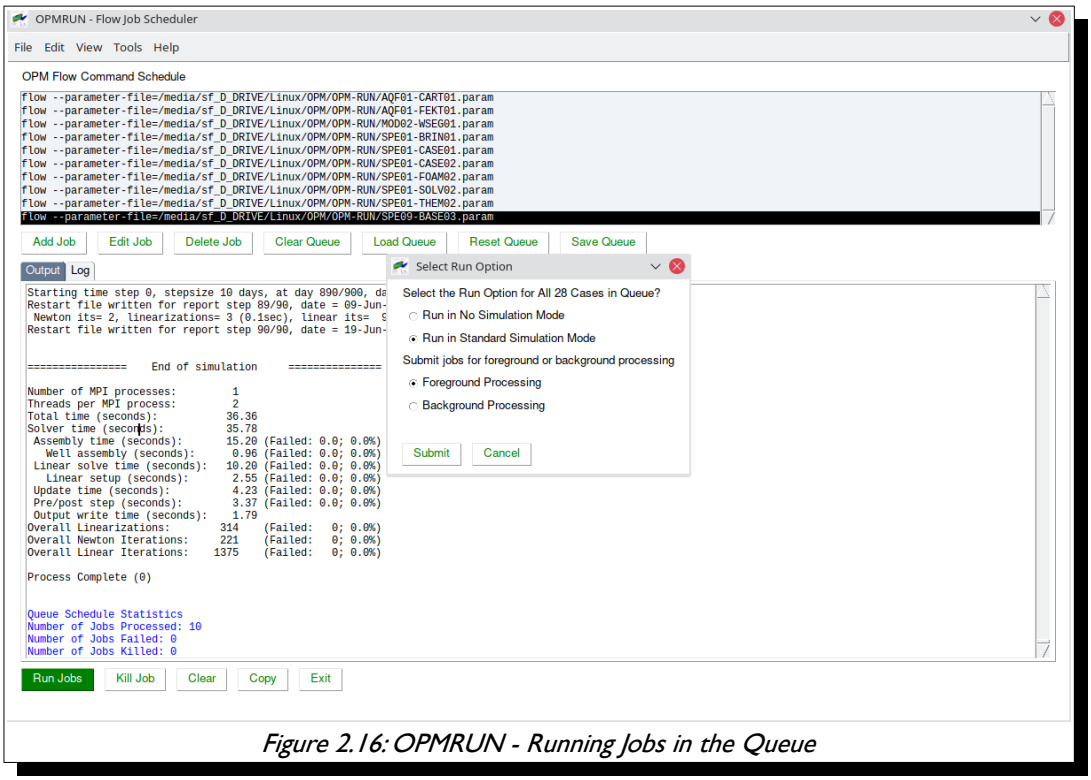


Figure 2.16: OPMRUN - Running Jobs in the Queue

Clicking the OPM Run Log tab displays the OPMRUN's session log file that records the time and date of the major events that have occurred, including the start and end times of each run. Notice also how OPMRUN deletes all the existing output files for a given job, if they exists, before running OPM Flow.

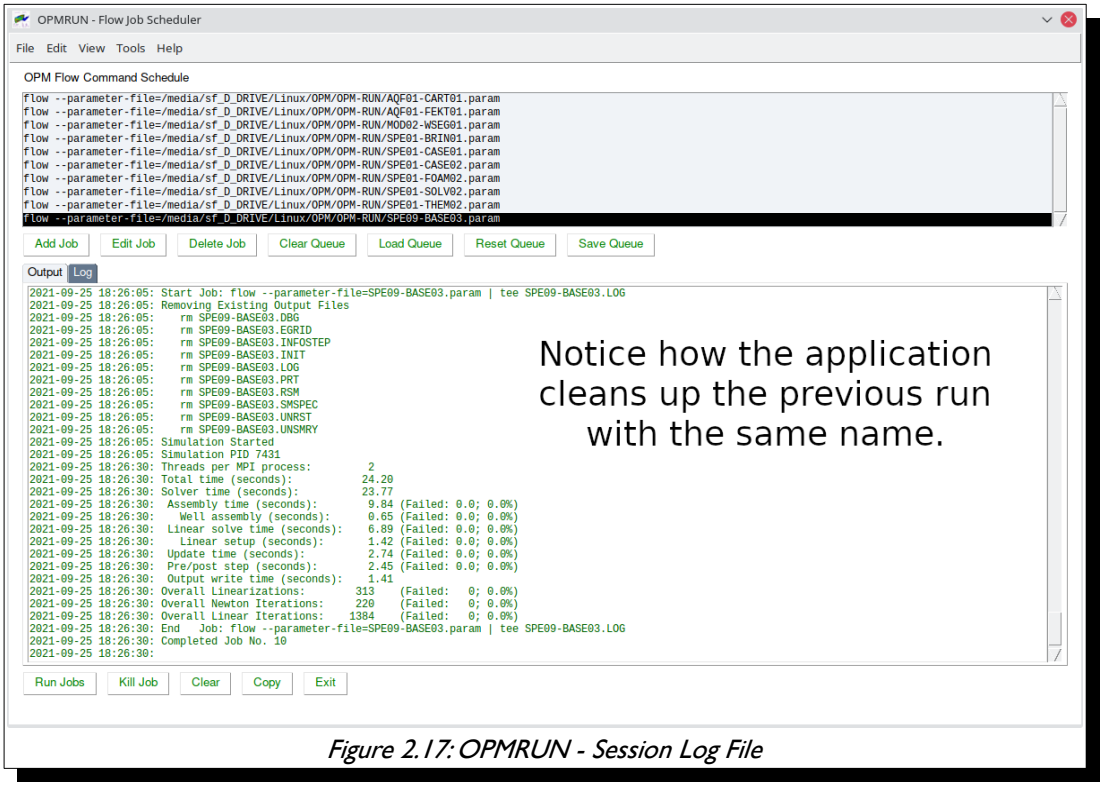


Figure 2.17: OPMRUN - Session Log File

The Kill button will ask the user if the current running job should be killed, and if the job is to be killed, the application will prompt as whether or not all the jobs in the queue should be killed.

The *Clear* clears the OPM Flow Output Element from the currently displayed tab (*Output* or *Log*) and the *Copy* button copies the data to the clipboard.

Finally, an example *.param file generated by OPMRUN for the 2022-10 release is shown below and on the following pages for reference:

```
#
# OPMRUN Parameter File
#
# File Name   : "/media/Fields/Norne/Model/NOR-BASE-A00.param"
# Created By  : opmuser
# Date Created: 2022-11-01 13:12:44
#
accelerator-mode="none"
allow-distributed-wells=false
alternative-well-rate-init=true
bda-device-id=0
continue-on-convergence-error=false
cpr-max-ell-iter=20
cpr-reuse-interval=10
cpr-reuse-setup=3
dbhp-max-rel=1
dp-max-rel=0.3
ds-max=0.2
dwell-fraction-max=0.2
ecl-deck-file-name="NOR-BASE-A00.DATA"
ecl-enable-drift-compensation=true
ecl-output-double-precision=false
ecl-output-interval=-1
ecl-strict-parsing=false
edge-weights-method=1
enable-adaptive-time-stepping=true
enable-async-ecl-output=true
enable-async-vtk-output=true
enable-dry-run="auto"
enable-ecl-output=true
enable-esmry=false
enable-logging-fallout-warning=false
enable-opm-rst-file=false
enable-storage-cache=true
enable-terminal-output=true
enable-tuning=false
enable-vtk-output=false
enable-well-operability-check=true
enable-well-operability-check-iter=false
enable-write-all-solutions=false
force-disable-fluid-in-place-output=false
force-disable-resv-fluid-in-place-output=false
fpga-bitstream=""
full-time-step-initially=false
ignore-keywords=""
ilu-fillin-level=0
ilu-redblack=false
ilu-relaxation=0.9
ilu-reorder-spheres=false
initial-time-step-in-days=1
linear-solver="ilu0"
linear-solver-ignore-convergence-failure=false
linear-solver-max-iter=200
linear-solver-reduction=0.01
linear-solver-require-full-sparsity-pattern=false
linear-solver-restart=40
```

```
linear-solver-verbosity=0
matrix-add-well-contributions=false
max-inner-iter-ms-wells=100
max-inner-iter-wells=50
max-newton-iterations-with-inner-well-iterations=8
max-pressure-change-ms-wells=1e+06
max-residual-allowed=1e+07
max-single-precision-days=20
max-temperature-change=5
max-welleq-iter=30
maximum-number-of-well-switches=3
milu-variant="ILU"
min-strict-cnv-iter=0
min-time-step-based-on-newton-iterations=0
min-time-step-before-shutting-problematic-wells-in-days=0.01
newton-max-iterations=20
newton-max-relax=0.5
newton-min-iterations=1
newton-relaxation-type="dampen"
opencl-ilu-parallel=true
opencl-platform-id=0
output-dir=""
output-interval=1
output-mode="all"
owner-cells-first=true
parameter-file=""
pri-var-oscilation-threshold=1e-05
print-parameters=2
print-properties=2
project-saturations=false
regularization-factor-wells=100
relaxed-max-pv-fraction=0.03
relaxed-pressure-tol-msw=10000
relaxed-well-flow-tol=0.001
scale-linear-system=false
sched-restart=false
serial-partitioning=false
shut-unsolvable-wells=true
solve-welleq-initially=true
solver-continue-on-convergence-failure=false
solver-growth-factor=2
solver-max-growth=3
solver-max-restarts=10
solver-max-time-step-in-days=365
solver-min-time-step=1e-12
solver-restart-factor=0.33
solver-verbosity=1
strict-inner-iter-wells=40
strict-outer-iter-wells=6
temperature-max=1e+09
temperature-min=0
threads-per-process=-1
time-step-after-event-in-days=-1
time-step-control="pid+newtoniteration"
time-step-control-decay-damping-factor=1
time-step-control-decay-rate=0.75
time-step-control-file-name="timesteps"
time-step-control-growth-damping-factor=3.2
time-step-control-growth-rate=1.25
time-step-control-target-iterations=30
time-step-control-target-newton-iterations=8
time-step-control-tolerance=0.1
time-step-verbosity=1
tolerance-cnv=0.01
tolerance-cnv-relaxed=1
```

```
tolerance-mb=1e-06
tolerance-pressure-ms-wells=1000
tolerance-well-control=1e-07
tolerance-wells=0.0001
update-equations-scaling=false
use-gmres=false
use-multisegment-well=true
use-update-stabilization=true
zoltan-imbalance-tol=1.1
#
# End of Parameter File
```

In the above example, all the default parameters have be used.

In addition to the functionality described in this section, the application has various additional features accessible through the application's menus including:

1) Configuring OPMRUN various options, default editor, default terminal console for background jobs, setting default project directories, etc.

2) Various additional simulation input generation and conversion utilities are available including:

   - Compressing a job to save space (DATA, and all OPM Flow output files) and uncompressing previously compressed jobs.

   - Keywords, a keyword generator based on the Apache Velocity Template Language ("VTL"). The templates can therefore also be used with any editor that supports VTL, jEdit for example. There is one template per keyword, with the formatting the same as the OPM Flow manual. Over 450 templates are currently implemented. One can also customize the existing templates as well as creating User defined templates. The keywords are examples, one still has to edit the resulting deck with the actual required data, but the format with comments should make this a straight forward process.

   - A Production Schedule application that takes a comma delimited CSV file containing historical production and injection data and converts the data to an OPM Flow SCHEDULE file using the WCONHIST series of keywords. Currently only production data is supported.

   - Sensitivities application that generates sensitivity cases based on a "Base" case file. The Base file contains "Factors" (variable names), $X01, $X02, etc., that are substituted with user defined values using the data entered and the type of Sensitivity Scenario selected.

   - A Well Specification application that uses the standard well export files from OPM ResInsight to reformat the data in a more user-friendly manner for the WELSPECS and COMPDAT keywords. Optionally, the application can generate the COMPLUMP keyword based on the OPM ResInsight layers file, with one completion per defined reservoir layer.

   - Calling OPM ResInsight and loading the currently selected job into OPM ResInsight for viewing.

   - A Well Trajectory Conversion application that converts a Schlumberger Petrel exported well trajectory file into an OPM ResInsight file, containing all the wells.

All the aforementioned options and tools are described in detail in APPENDIX C:OPMRUN – FLOW JOB SCHEDULER,

Note

OPMRUN can be run as either a Linux or a Windows application. If the software is installed and run under Windows, then the program automatically handles the file naming conventions of the two operating systems. That is under Windows, the application uses the Windows file name to select the files, and then automatically converts the file name to the equivalent Linux file name to run under WSL.

2.4 IMPROVING SIMULATOR CONVERGENCE AND NUMERICAL PERFORMANCE

In general, most simulator performance issues are caused by the data used by the simulator, rather than the numerical algorithms used to solve the equations presented in sections *1.2 Black-Oil Model Equations* through *1.4 Multi-Segment Well Model*. Thus, it is important to verify all warning messages and to address the issues raised by these warnings. Naturally, some warnings can be safely ignored, while others should not. Just because the simulator runs with warning messages does not mean that the results are reasonable, or that numerical performance has not been impacted.

This section is therefore broken into two parts, the first part discusses potential data errors and how to address them, and the second section outlines the numerical controls available to improve the simulator's performance. The latter should only be done after the data quality issues have been rectified, although, it is not uncommon for users to skip the data quality step altogether, and jump to "tuning" the model's numerical controls - to be clear, skipping the data quality step is strongly not recommended.

2.4.1 DATA QUALITY AND ASSURANCE

GRID Section

Grid Configuration

In full field models grid distortions may occur due to faulting and the structural complexity of the geological model, and ideally one wishes to have all the cells to be orthogonal. Grid cells are orthogonal if all grid lines intersect at a right angles, that is at 90°. This is because orthogonal grids offer significant advantages in solving a system of partial differential equations used computational fluid dynamics. In addition, they ensure that the fluid flow is not being unduly influenced by the grid orientation effects, as cell to cell flow only occurs across connected cell faces.

Most earth modeling software can produce a histogram of the cell angles as an aide to checking the orthogonality of the grid. A common cause for cells not being orthogonal, as well as creating other grid artifacts, is the grid cells following actual the faulting traces, as oppose to using "zig-zag" fault traces that follow the grid. A typical fault trace grid is illustrated in Figure 2.18.

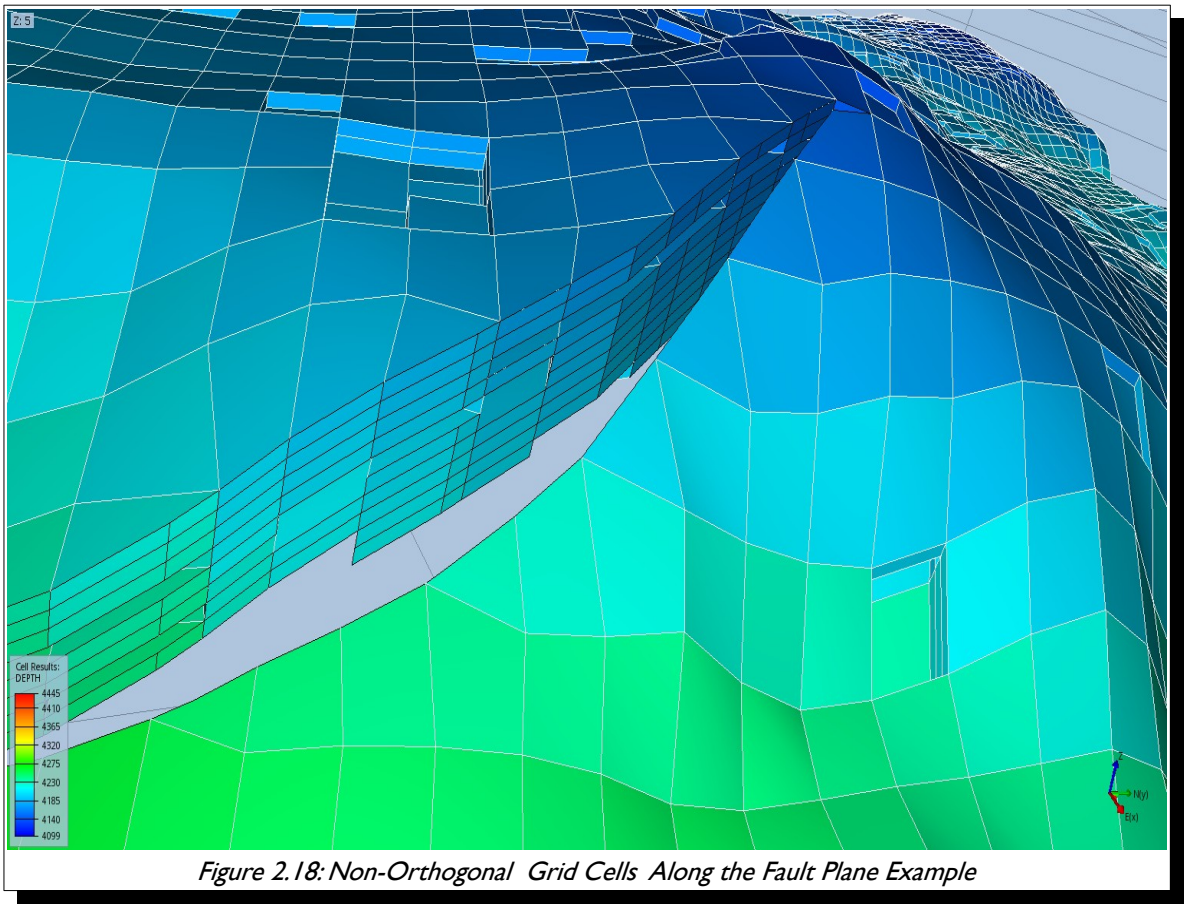


Figure 2.18: Non-Orthogonal Grid Cells Along the Fault Plane Example

Notice in Figure 2.18 how the cells along the fault plane are severely distorted, with some cells having only three faces instead of four in the x-y plane. Now compare this to the Norne model shown in Figure 2.19, which has more complex faulting, and yet the grid along the fault planes is mainly orthogonal.

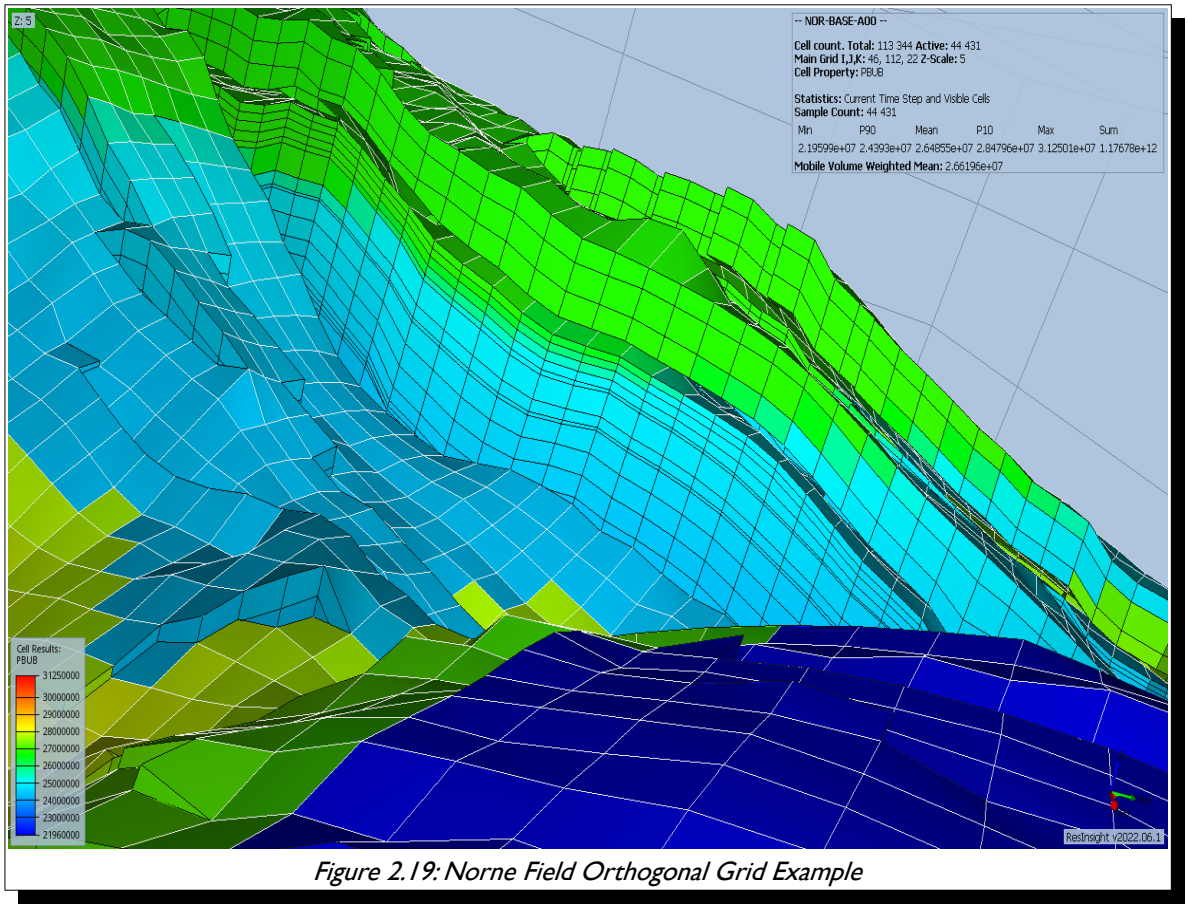


Figure 2.19: Norne Field Orthogonal Grid Example

Grid Property Data and Active Cells

There is a current trend in the industry for not applying cut-offs to the petrophysical parameters used to generate the grid property data (porosity, net-to-gross, etc.). The philosophy behind this approach is that the simulator should determine what is reservoir and is capable of contributing to flow within the reservoir, and what is non-reservoir. At first glance this would appear reasonable, but unfortunately the inclusion of low pore volume and low permeability grid cells will have a dramatic negative impact on the model's numerical convergence and run times. This is because one ends up with many more active cells than otherwise, plus the fact that low pore volumes grid blocks will be connected to high pore volume cells. Because of this inter-connectedness, a change in a large pore volume grid block will have disproportional change on a low pore volume cell, which will result in smaller time step sizes in order to satisfy the numerical controls.[46] That is, the low pore volume cell throughput will influence the simulator's selected time step size disproportionately.

Nevertheless, even if the more conventional approach to the model build is undertaken by applying some form of petrophysical cut-offs, there may well still be grid blocks containing small pore volumes. In this case the MINPV keyword in the GRID section can be used to set a minimum pore volume threshold in which cells below the threshold value will be automatically made inactive. It is not possible to give absolute guidance for a suitable pore volume value to be used with MINPV, the best approach is to run some sensitivities to investigate the impact of different pore volume thresholds on the reduction in the number of cells and the consequent change in the fluid in-place volumes. One should also be mindful not to remove thin high permeability cells that have small pore volumes due to their size, as this will effect the model's flow behavior.

Finally, if the PINCH keyword in the GRID section, is also in the input deck, then one must be cognizant of the interaction between the two keywords.

[46] *The large pore volume differences make the solution of the linear system of equations more challenging, due to the residual errors in certain grid cells or areas within the grid, that are much harder to reduce with iterative solvers. This results in more convergence failures, smaller time steps, etc. Similarly, for highly varying cell sizes.*

Grid Coarsening

As the number of active grid blocks directly effect a model's run time, reducing the number of active cells by coarsening areas of model that are of secondary importance, can be beneficial to the overall run time of a model. An example of where this approach should be considered is the modeling of aquifers. It is not necessary to fully model an aquifer, by having the model cells cover the whole area of the aquifer. Instead, incorporating either analytical aquifers via the *AQUCT – Define Carter-Tracy Analytical Aquifers* and *AQUFETP – Define Fetkovich Analytical Aquifers* keywords, or numerical aquifers using the *AQUNUM – Define Numerical Aquifer Properties* keyword, can effectively re-produce the aquifer response. Both types of aquifers have advantages and disadvantages.

Analytical aquifers have the advantage that their response and behavior can be matched by using material balance techniques to quantify the aquifer properties, by history matching the measured pressure response in the hydrocarbon zones. This is an efficient analytical modeling work flow to identify the aquifer response, the properties derived can therefore be more or less directly incorporated into the dynamic model. The dis-advantage is that analytical aquifers are not directly comparable with geological interpretations, which may, or may not, be important, depending upon the circumstances.

The opposite is true for numerical aquifers in that an approximate aquifer volume and property data can be matched with current geological interpretation, but cannot be history matched in most material balance software, as numerical aquifers in general are not supported.

Nevertheless, no mater which type of aquifer is employed, they should not be directly connected to hydrocarbon bearing cells, ideally there should be between three to five water bearing cells between the aquifer connected cells and the hydrocarbon cells. However, the water cells may vary in size, such that water cells connected to the hydrocarbon zone are of similar size, and then increase in size towards the edge of the model. The reason for this is that we wish to limit "small" hydrocarbon cells being directly coupled to "large" water cells, that will hinder the length of time steps selected by the simulator due to the convergence criteria.

Grid Isolated Cells

Simulation models should only include grid blocks that influence the flow and recovery in the model. One common issue with some of the geo-statistical formulations used to generate the grid and the associated grid property data, is the tendency to create isolated grid blocks. These isolated grid blocks are not connected to the rest of the model and therefore do not influence the behavior of the model. Figure 2.20 shows a typical example, with the cells circled in red being the isolated cells.

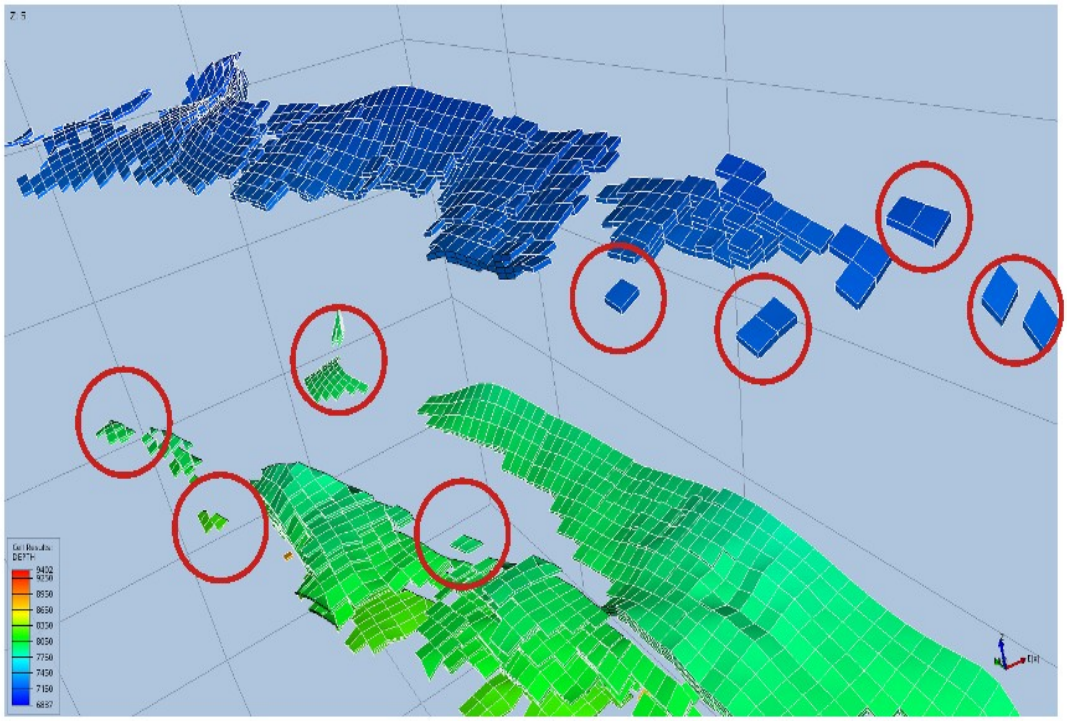


Figure 2.20: Isolated Cells Example

Unfortunately, these cells will be included in all the calculations, and may cause numerical issues due to their unchanging state, both pressure and saturation. They will also unnecessarily add to the active cell count, and therefore effect the simulation run time. In addition, they will give a false impression of the average reservoir pressure, as the isolated cell pressures will remain at initial conditions throughout the simulation run. If model convergence is proving difficult, then this may be a potential cause of the problem. In any case, it is recommended that the identified isolated cells be removed by making them inactive via the ACTNUM keyword in the GRID section.

If there are only a few isolated cells then they can manually be made inactive; however, in many cases this not possible due to the number of isolated grid blocks. In this case, it is better to revert back to the static model and fix the issue in the earth modeling software. Indeed, if there are numerous isolated grid blocks this may be indicative that an inappropriate approach has been used to generate the static model, although it may be also due to the depositional environment as well.

PROPS Section

This section describes potential issues associated with the property data and is split into two separate sub-sections:

1) The first sub-section covers the rock properties, and in particular the saturation functions (relative permeability curves), that can have a dramatic impact on reservoir performance, especially for the non-linear solver. As will be shown later, it is important that the derivatives of the relative permeability curves are smooth and monotonically increasing or decreasing depending on the phase of the relative permeability fluid. Particular care must be taken if end-point scaling is being used, as it is not uncommon for the resulting de-normalized curves being inappropriate, even though the normalized curves are reasonable and correct.

2) The second sub-section addresses issues that may arise with the fluid properties, and inconsistent PVT data that may result in the simulation struggling to converge in the linear solver.

PROPS Section: Saturation Function (Derivative)

In general, one should not enter laboratory derived relative permeability data directly into the simulator. This is because it is dubious that laboratory measurements on 1½ inch core plugs can represent the flow in 100 x 100 x 1.0 m grid cell in the model. There are various upscaling techniques that can account for the scaling issue, using various forms of relative permeability curves, including the laboratory curves, piston-like curves and thickness average curves etc. However, the resulting pseudo relative permeability curves tend to not have smooth derivatives, which will have a detrimental impact on convergence efficiency. Thus, it is common practice to use the laboratory derived curves, either in an upscaling workflow or as part of the input workflow, and to fit a Corey[48] type curve to the "rock" curves in order to smooth out any discontinuities.

Unfortunately, neither OPM Flow nor the commercial simulator, unlike some other simulators, support the direct entry of Corey type curves; however, OPM Flow does support the more advance and flexible LET family of models instead. See section *8.2.6Saturation Table Generation - LET Functions* and Lomeland et al[47] for further information on the model, as well as the *SGOFLET– Gas-Oil LET Relative Permeability Functions*, *SGWFLET – Gas-Water LET Relative Permeability Functions*, and the *SWOFLET – Water-Oil LET Relative Permeability Functions* keywords in the PROPS section.

Corey[48] combined the work of Purcell[49] and Burdine[50] that was widely accepted for its simplicity. His original equations were developed for the drainage cycle in water-wet sandstones, but have also been used in carbonate formations. Corey's original water-oil equations were as follows:

$$k_{ro}\left(S_w\right) = \left[\frac{1 - S_w}{1 - S_{wcr}}\right]^{n_o} \tag{2.1}$$

$$k_{rw}\left(S_w\right) = \left[\frac{S_w - S_{wcr}}{1 - S_{wcr}}\right]^{n_w} \tag{2.2}$$

Where:

$k_{ro}(S_w)$	= relative permeability to oil,
$k_{rw}(S_w)$	= relative permeability to water,
n_o	= Corey oil exponent, set to four in the original paper,
n_w	= Corey water exponent, set to four in the original paper,

[47] *Lomeland F., Ebeltoft E. and Thomas W.H., 2005. A New Versatile Relative Permeability Correlation. Paper SCA2005-32 presented at the International Symposium of the Society of Core Analysts held in Toronto, Canada, 21-25 August, 2005.*

[48] *Corey, A. T.: "The Interrelation Between gas and Oil Relative Permeabilities", Production Mon., 19. 38. (1954).*

[49] *Purcell, W. R., "Capillary Pressures- Their Measurement Using Mercury and the Calculation of Permeability Therefrom", Transactions AIME, 186, 39 (1949).*

[50] *Burdine, N. T., "Relative Permeability Calculations from Pore Size Distribution Data", , Transactions AIME, 198, 71 (1953).*

S_w = water saturation, and
S_{wc} = critical water saturation.

The denominator in equations (2.1) and (2.2) scales the water saturation to the mobile water phase. There are several forms of these equations, with the most common normalizing the saturation over the mobile hydrocarbon phase, as depicted in equations (2.3) and (2.4).

$$k_{ro}(S_w) = k_{row}\left[\frac{1 - S_w - S_{orw}}{1 - S_{orw} - S_{wcr}}\right]^{n_o} \quad (2.3)$$

$$k_{rw}(S_w) = k_{rww}\left[\frac{S_w - S_{wc}}{1 - S_{orw} - S_{wcr}}\right]^{n_w} \quad (2.4)$$

Where,

k_{row} = maximum oil relative permeability at Swc,
k_{rw} = maximum water relative permeability st Sorw,
S_{wc} = critical water saturation, and
S_{orw} = residual oil saturation under a water flood (SOWCR).

Figure 2.21 show a typical Corey plot, with the plot at the top of the figure showing the Corey plot and the bottom plot showing the derivative plot, the data is also tabulated in Table 2.2. Notice in Figure 2.21 that the water curve does not extend to 100% water saturation, this is an error as the water below the oil-water contact would have restrictive water flow.

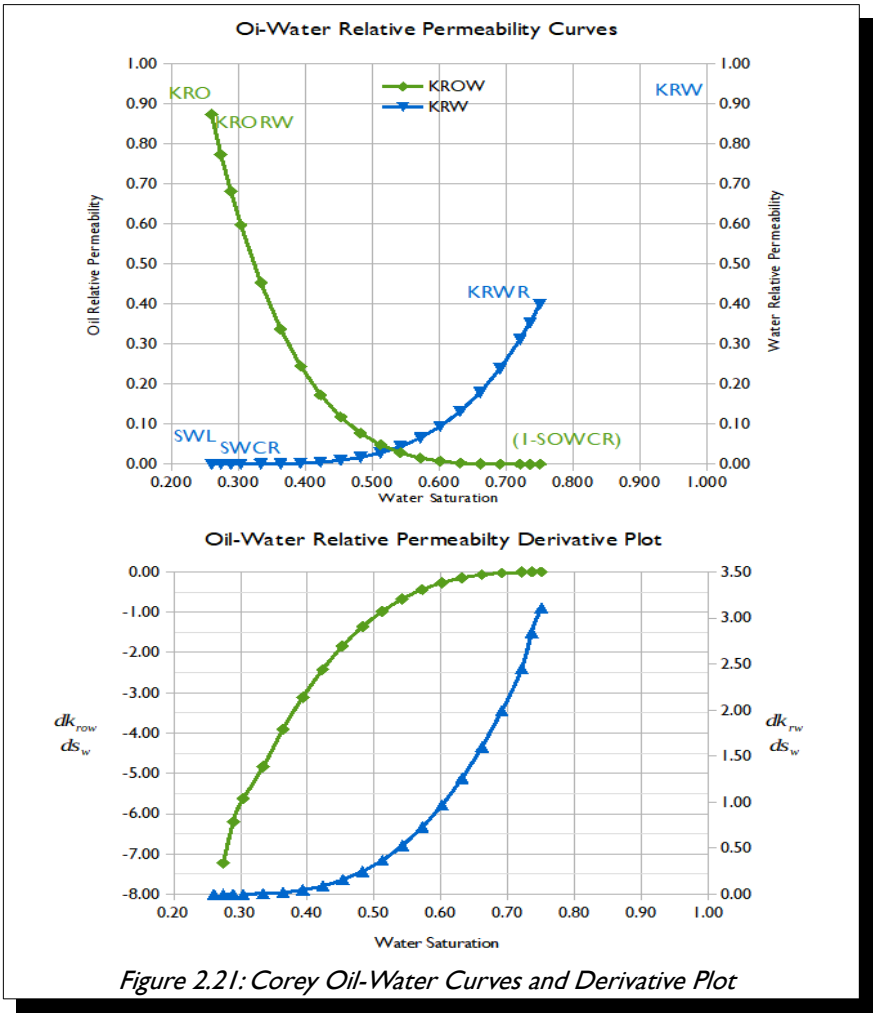


Figure 2.21: Corey Oil-Water Curves and Derivative Plot

Oil-Water **Saturation Function** and Derivative **Table**						
Sw	**Krw**	**Krow**	**Pcow (psia)**	$\frac{dk_{rw}}{dS_w}$	$\frac{dk_{row}}{dS_w}$	$\frac{dP_{cow}}{dS_w}$
0.2600	0.0000	0.8750	16.4040	0,00	0,00	0,00
0.2740	0.0000	0.7737	13.4170	0,00	-7,23	-213,13
0.2889	0.0000	0.6814	11.0910	0,00	-6,20	-156,36
0.3038	0.0000	0.5976	9.2570	0,00	-5,63	-123,34
0.3335	0.0002	0.4535	6.6120	0,01	-4,84	-88,90
0.3633	0.0008	0.3372	4.8610	0,02	-3,91	-58,87
0.3930	0.0022	0.2448	3.6610	0,05	-3,11	-40,33
0.4228	0.0049	0.1728	2.8150	0,09	-2,42	-28,44
0.4525	0.0096	0.1181	2.2030	0,16	-1,84	-20,55
0.4823	0.0171	0.0775	1.7520	0,25	-1,36	-15,18
0.5120	0.0282	0.0484	1.4120	0,37	-0,98	-11,42
0.5418	0.0440	0.0283	1.1520	0,53	-0,67	-8,74
0.5715	0.0656	0.0153	0.9500	0,73	-0,44	-6,78
0.6013	0.0944	0.0074	0.7920	0,97	-0,27	-5,34
0.6310	0.1318	0.0030	0.6650	1,26	-0,15	-4,25
0.6608	0.1793	0.0010	0.5640	1,60	-0,07	-3,42
0.6905	0.2386	0.0002	0.4810	1,99	-0,03	-2,78
0.7203	0.3115	0.0000	0.4130	2,45	-0,01	-2,28
0.7351	0.3537	0.0000	0.3840	2,84	0,00	-1,97
0.7500	0.4000	0.0000	0.3570	3,11	0,00	-1,80
1.0000	0.9000	0.0000	0.1270	2,00	0,00	-0,92

Table 2.2: Oil-Water Saturation Function and Derivative Table

If we correct the data shown in Figure 2.21 by extending the water curve to KRW, then we obtain the result shown in Figure 2.22, but notice what has happens to the water derivative plot, which now has a discontinuity because of the extension. Thus, small changes to the relative permeability curves can gave an undue influence on the simulator's performance. In this particular case, the effect is minor as it is only one point that has of issue, but it is not uncommon to have multiple discontinuities in numerous curves that will have an impact on convergence.

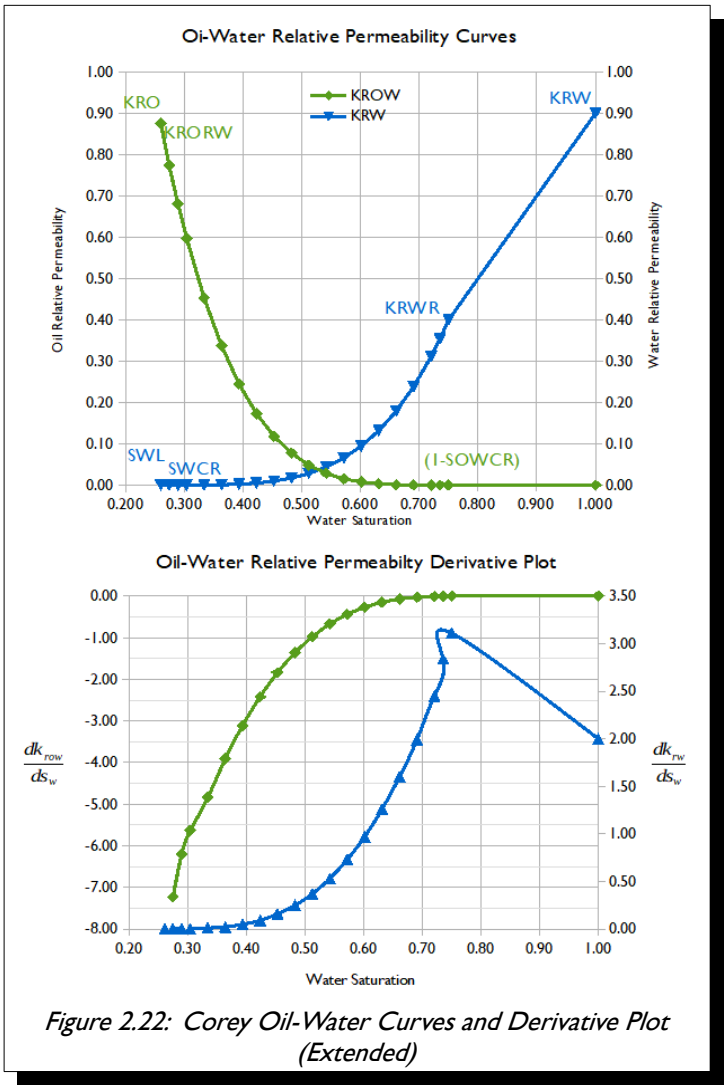


Figure 2.22: Corey Oil-Water Curves and Derivative Plot (Extended)

To summarize, if convergence is an issue verify that both the entered relative permeability data and capillary pressure do not have numerous discontinuities in the derivative plots.

The next section discusses the impact and relationship lowest saturation and critical saturation, in terms of pressure depletion. However, again, care should be taken to avoid any potential discontinuities in the derivatives between these to data points.

PROPS Section: Saturation Function (Lowest Saturation and Critical Saturation)

In most cases time step chops caused by saturation changes, may be reduced by avoiding having the lowest and critical saturations being the same. For example for the water phase, this would mean having different values for SWL and SWCR. The reason for this due to the compressibility difference between the gas and liquids (oil and water), as illustrated Figure 2.23. Here we assume that initially 80% of cell's volume of 100 m3 is occupied by gas and the remaining 20% by water.

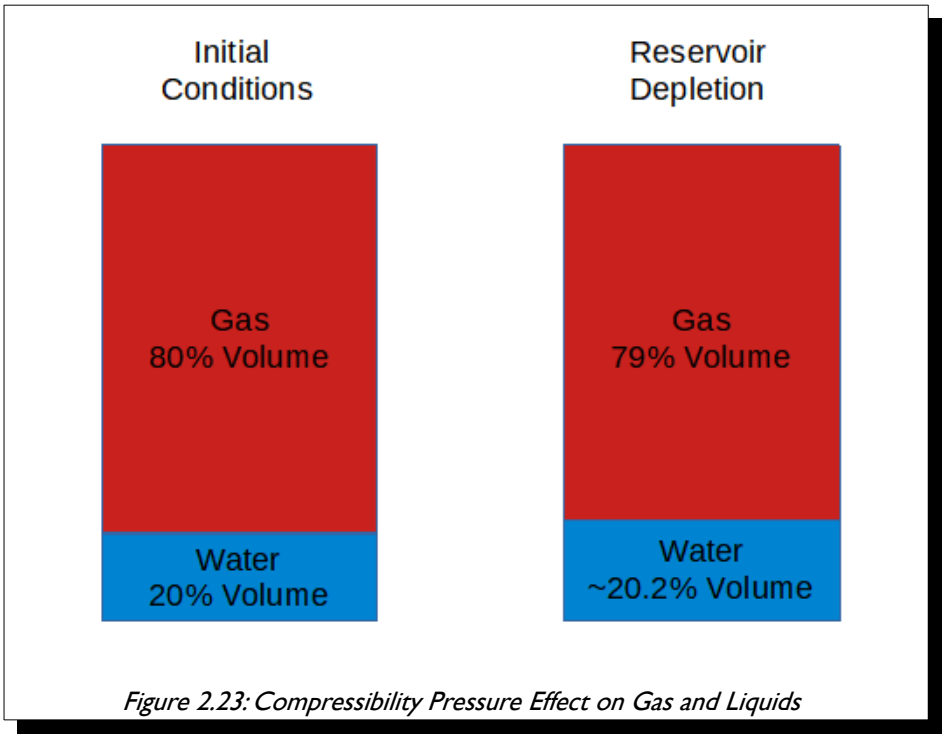


Figure 2.23: Compressibility Pressure Effect on Gas and Liquids

Assuming there is no flow in or out of the grid block and the cell pressure decreases due to production elsewhere in the model, that results in a small reduction (1.0%) in the cell's pore volume. And since gas is much more compressible than oil and water, then the gas volume will decrease by approximately1.0%; whereas the water will increase slightly to 20.2%[51]. Thus, if SWL was set to 20%, then the 0.2% increase in water saturation would become mobile and start to flow to the other connect grid blocks. This can be a cause of instability.

The solution of cause is to enter values for SWL and SWCR, such that there is a small difference between them, around 2% to 5% for example, instead of this:

```
--
--        WATER RELATIVE PERMEABILITY TABLES (SWFN)
--
SWFN
--        SWAT        KRW         PCOW
--        FRAC        FRAC        PSIA
--        --------    --------    -------
           0.30       0.0000      0.6190
           0.35       0.0005      0.3891
           0.40       0.0028      0.2279
           0.50       0.0237      0.0602
           0.60       0.0931      0.0077
           0.80       0.5000      0.0000
           0.90       0.7000      0.0000
           1.00       0.9000      0.0000                    / TABLE NO. 1
```

[51] *Assuming that the initial pore volume of the grid block was 100 m^3, then a 1.0% reduction would result in a pore volume 99 m^3, and 20 m^3 of water in 99 m^3 pore volume results in 20.2% of the grid block being occupied by water.*

Amend the table as follows:

```
--
--       WATER RELATIVE PERMEABILITY TABLES (SWFN)
--
SWFN
--       SWAT        KRW         PCOW
--       FRAC        FRAC        PSIA
--       --------    --------    -------
          0.30       0.0000      0.6190
          0.31       0.0000      1*
          0.35       0.0005      0.3891
          0.40       0.0028      0.2279
          0.50       0.0237      0.0602
          0.60       0.0931      0.0077
          0.80       0.5000      0.0000
          0.90       0.7000      0.0000
          1.00       0.9000      0.0000                    / TABLE NO. 1
```

The incorporation of the of Sw equal to 0.31should assist convergence and will not influence the oil in-place, although it will slightly reduce the water mobility between the water saturations of 0.30 and 0.31. Secondly, as one can see from Table 2.3 the additional water saturation value did not cause any derivative discontinuities.

Oil-Water **Saturation Function** and Derivative **Table**						
Sw	**Krw**	**Krow**	**Pcow (psia)**	$\frac{dk_{rw}}{dS_w}$	$\frac{dk_{row}}{dS_w}$	$\frac{dP_{cow}}{dS_w}$
0,30	0.0000	0.6190	9.7247			
0,31	0.0000	0.5918	9.1501	0,00	-5,44	-114,92
0,35	0.0005	0.3891	5.6425	0,01	-4,50	-77,95
0,40	0.0028	0.2279	3.4623	0,05	-3,22	-43,60
0,50	0.0237	0.0602	1.5494	0,21	-1,68	-19,13
0,60	0.0931	0.0077	0.7989	0,69	-0,52	-7,50
0,80	0.5000	0.0000	0.3110	2,03	-0,04	-2,44
0,90	0.7000	0.0000	0.2190	2,00	0,00	-0,92
1,00	0.9000	0.0000	0.1270	2,00	0,00	-0,92

Table 2.3: Oil-Water Saturation Function and Derivative Table (Modified)

Note also that same approach can also be applied to the oil phase.

PROPS Section: Saturation Table End-point Scaling

The end-point scaling functionality in OPM Flow gives tremendous flexibility, allowing one to use a limited set of relative permeability functions by accounting for rock property variation, such that each cell in the model has its own relative permeability function. However, both the original normalized functions and the de-normalized grid cells functions should be check for consistency. In general, it is best to avoid entering the original curves as normalized curves, in the classical sense, but instead enter a standard curve or a more correct normalized curve that takes into all the end-points. The reason for is shown in Figure 2.24 and Figure 2.25.

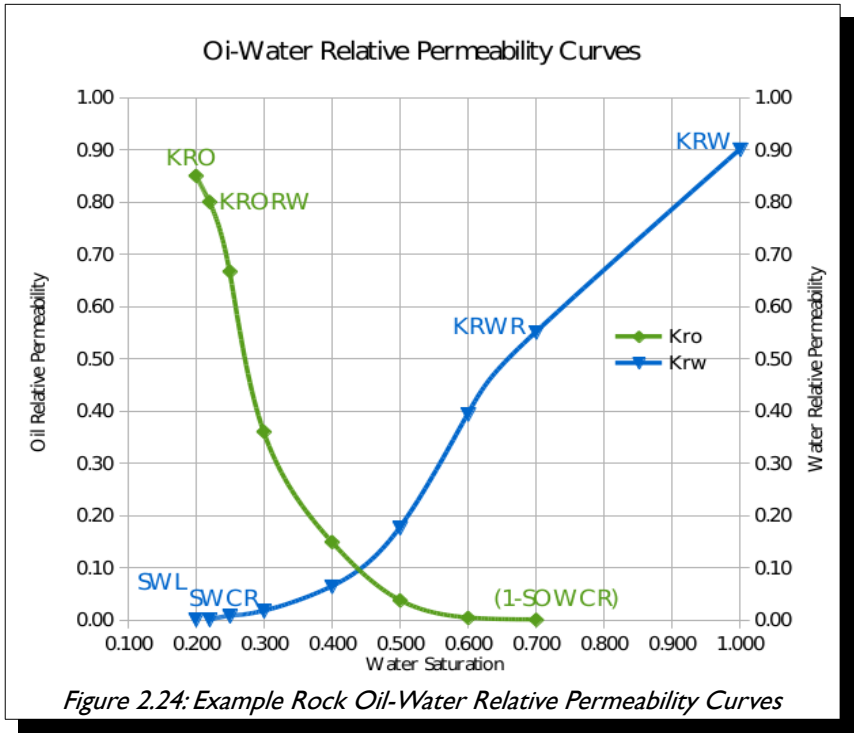


Figure 2.24: Example Rock Oil-Water Relative Permeability Curves

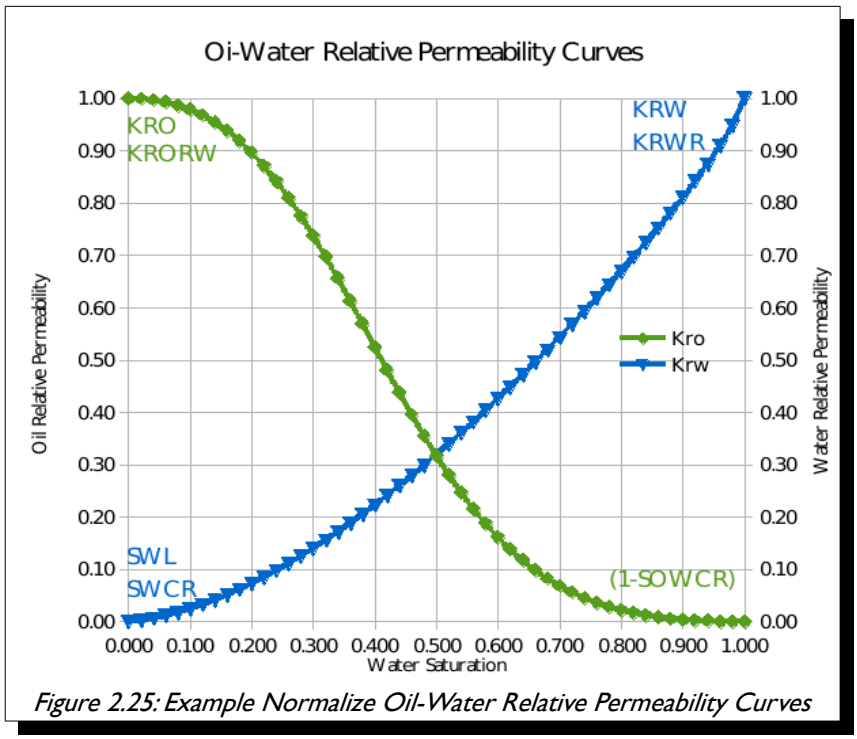


Figure 2.25: Example Normalize Oil-Water Relative Permeability Curves

Notice how the distinction between KRO and KROW is no longer present in the normalized curve in Figure 2.25, and similarly for KRW and KRWR, and SWL and SWCR. Thus, the simulator will ignore one of each pair of end-points - this can lead to unexpected results for the grid cell based saturation functions. An example is shown in Figure 2.26 for a gas-water saturation table. Notice how the de-normalized end-point water relative permeability, KRW, is only 0.20 versus 1.00 for the normalized value of KRW.

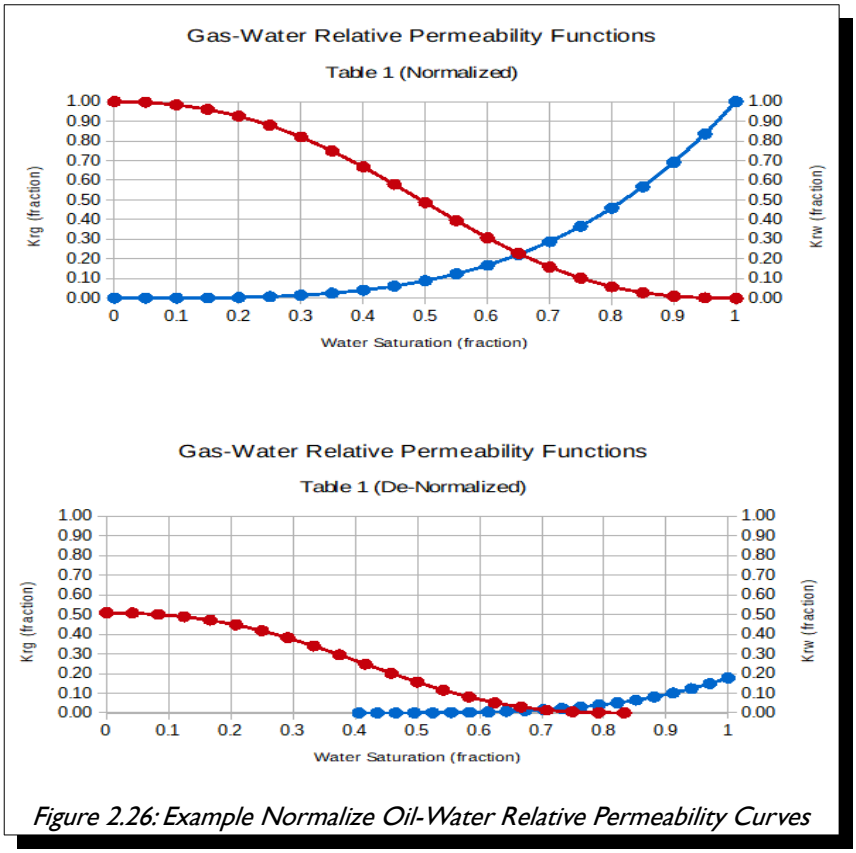


Figure 2.26: Example Normalize Oil-Water Relative Permeability Curves

Consequently, water influx will be inhibited leading to unreliable results in terms of water production, and potentially gas production if aquifers are present in the model.

As mentioned previously, entering a correct original curve that is not normalized will automatically resolve these type of issues. If a normalized curve is entered, then ensure that simulator can distinguish between SWL and SWCR etc., based on the variable definitions provided in Error: Reference source not found, Error: Reference source not found, and Error: Reference source not found.

Finally, see the SCALCERS keyword in the PROPS section, that defines the options used in the re-scaling process, including the options for two point and three point scaling. Whatever option is selected, always verify that the de-normalized curves that will be used in the model are correct and reasonable.

PROPS Section: Fluid Properties

It is important that the fluid properties are consistent to avoid convergence issues, and thus the simulator performs a series of validation checks to ensure that data is consistent. The following example indicates the oil formation volume factor is is not monotonically increasing for increasing GOR.

```
--
--       OIL PVT TABLE FOR LIVE OIL
--
PVTO
--       RS        PSAT       BO         VISC
--       MSCF/STB  PSIA       RB/STB     CPOISE
--       --------  --------   -------    ------
.............................................
                                                    /  TABLE NO. 43
         0.000    1.010      1.0388000  120924.000
                  2510.0     0.9801748  529773.445         /
         3.909      10.0     1.0563000   45850.000
                  2510.0     0.9969008  200315.440         /
         5.725      20.0     1.0617000   34198.000
                  2510.0     1.0022360  148947.825         /
         6.787      30.0     1.0648000   28808.000
                  2510.0     1.0054019  125083.718         /
         7.540      40.0     1.0671000   25508.000
                  2510.0     1.0078136  110411.450         /
         8.125      50.0     1.0688000   23210.000
                  2510.0     1.0096595  100151.779         /
         9.940     100.0     1.0742000   17312.000
                  2510.0     1.0159689   73535.291         /
        11.002     150.0     1.0774000   14583.000
                  2510.0     1.0202071   60960.876         /
        11.756     200.0     1.0797000   12912.000
                  2510.0     1.0235993   53105.652         /
        12.340     250.0     1.0814000   11749.000
                  2510.0     1.0264272   47530.728         /
        12.818     300.0     1.0829000   10877.000
                  2510.0     1.0290688   43270.164         /
        13.221     350.0     1.0841000   10191.000
                  2510.0     1.0314284   39854.509         /
        13.571     400.0     1.0851000    9631.000
                  2510.0     1.0336002   37015.563         /
        13.880     450.0     1.0861000    9163.000
                  2510.0     1.0357743   34599.472         /
        14.156     500.0     1.0869000    8763.000
                  2510.0     1.0377596   32498.634         /
        14.405     550.0     1.0877000    8417.000
                  2510.0     1.0397467   30648.326         /
        14.633     600.0     1.0884000    8113.000
                  2510.0     1.0416399   28994.746         /
        14.843     650.0     1.0890000    7843.000
                  2510.0     1.0434389   27501.353         /
        15.037     700.0     1.0896000    7601.000
                  2510.0     1.0452393   26140.639         /
        17.800    2500.0     1.0222000    4436.000
                  2510.0     1.0219701    4495.778         /
                                                    / TABLE NO. 44
```

The data is displayed above and the offending entries are shown in bold. These type of PVT warning messages are important and should be addressed before continuing with the simulation run.

SOLUTION Section: Initialization

The first quality control check that should be performed once the model has been initialized, is to check that the pressure distribution matches the observed data, this specifically important when there are multiple stack reservoirs in different hydraulic pressures zones, as per the likes of the oil and gas fields in the Malay Basin and the deep water fields in Indonesia and the Gulf of Mexico. One effective way to do this is to plot the initial pressure data versus depth from exploration wells, and compared this to the equivalent model pressures at the same depth, as shown in Figure 2.27. Here the measured data is from the RFT[52] ("Repeat Formation Tester") tool that measures the observed pressure profile in the open-hole well bore.

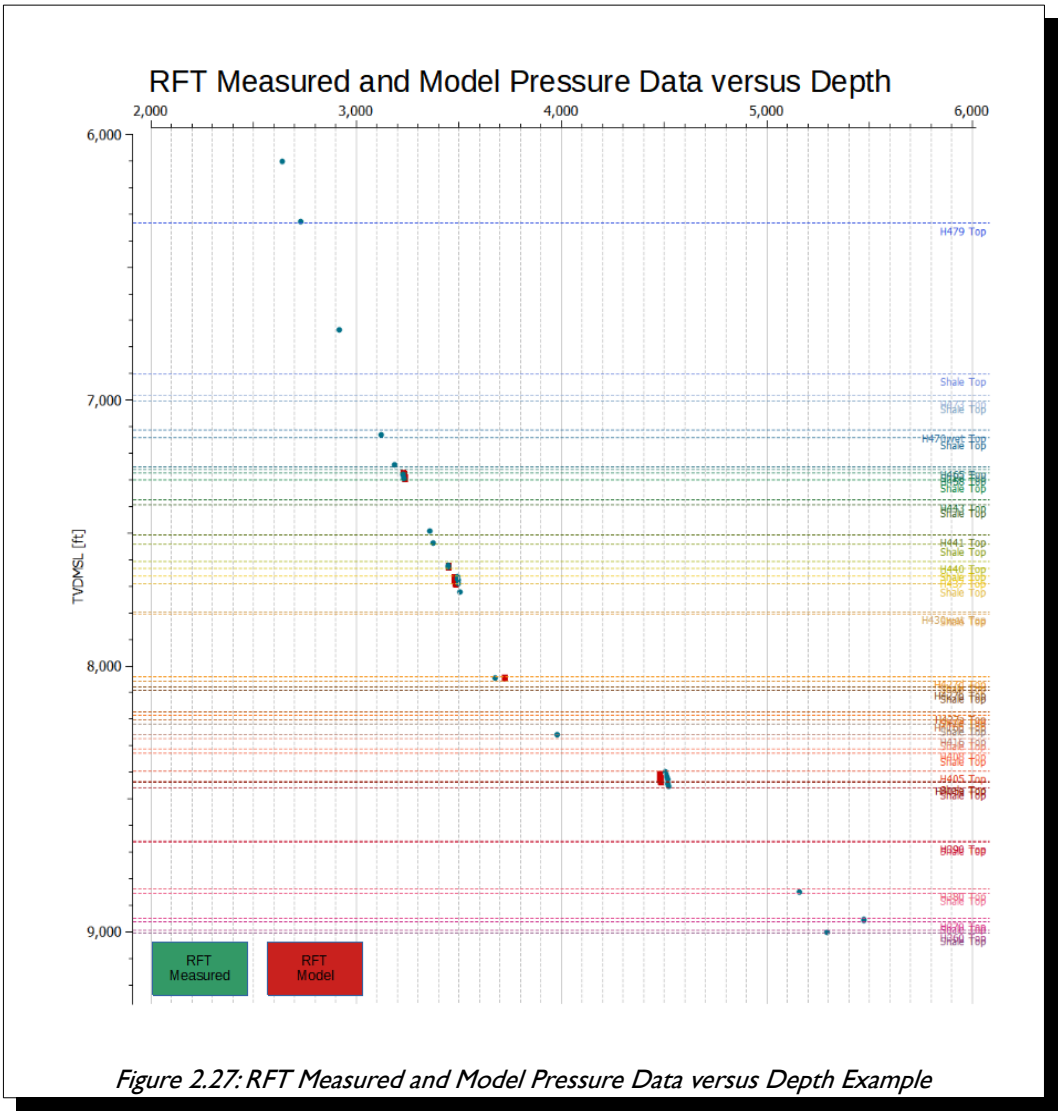


Figure 2.27: RFT Measured and Model Pressure Data versus Depth Example

Figure 2.27 was created in OPM ResInsight, and shows good agreement between the two data sets. This type of plot is commonly constructed when history matching field performance, but is also useful in verifying the initial pressure distribution. Note that for simple models with only a few reservoir zones, then this plot can be replaced by simply checking the three dimensional pressure distribution.

[52] *Repeat Formation Tester (RFT) tool is an open hole device which is an updated version of the Formation Interval Tester (FIT), both of which are run on wire line. Both tools take multiple pressure readings (at various depths) thus enabling a pressure depth profile to be obtained from the formation, and, in addition, they can also take fluid samples from the formation. The latest tool available is the Modular Formation Dynamics Tester (MDT), which, as its name suggests, is a modular tool that can be assembled in different configurations depending on what are the objectives for running the tool. Thus, the term RFT measurements applies to all tools that measure a pressure profiles versus depth (RFT/FIT/MDT etc.).*

If the SWATINIT keyword in the PROPS section is used to initialize the model by setting each grid block's initial water saturation ("Sw"), then OPM Flow will re-scale the water-oil capillary pressure curves entered via the SWFN saturation functions in the PROPS section, so that the resulting initialized Sw matches the values in the SWATINIT array.

Normally the SWATINIT array is generated in the static earth model when calculating the hydrocarbons in-place volumes using a Saturation Height Functions ("SHF") derived from capillary pressure functions. Static earth models do not directly use capillary pressure in these type of calculations as individual cell pressures are not available. There is therefore some potential for inconsistencies to arise between the two sets of formulations. This is normally manifested by extreme scaling in the scaled capillary pressure values calculated by the simulator. These scaled large values of capillary pressures can result in numerical issues, and should be addressed, by verifying that the SHF used in the earth model to derive the SWATINIT array and the unscaled capillary pressure values on the SWFN keyword are consistent. Note in general, the scaled capillary pressure values calculated by the simulator, should be roughly in the same range as the entered unscaled data, as shown in Figure 2.28 for the Norne field.

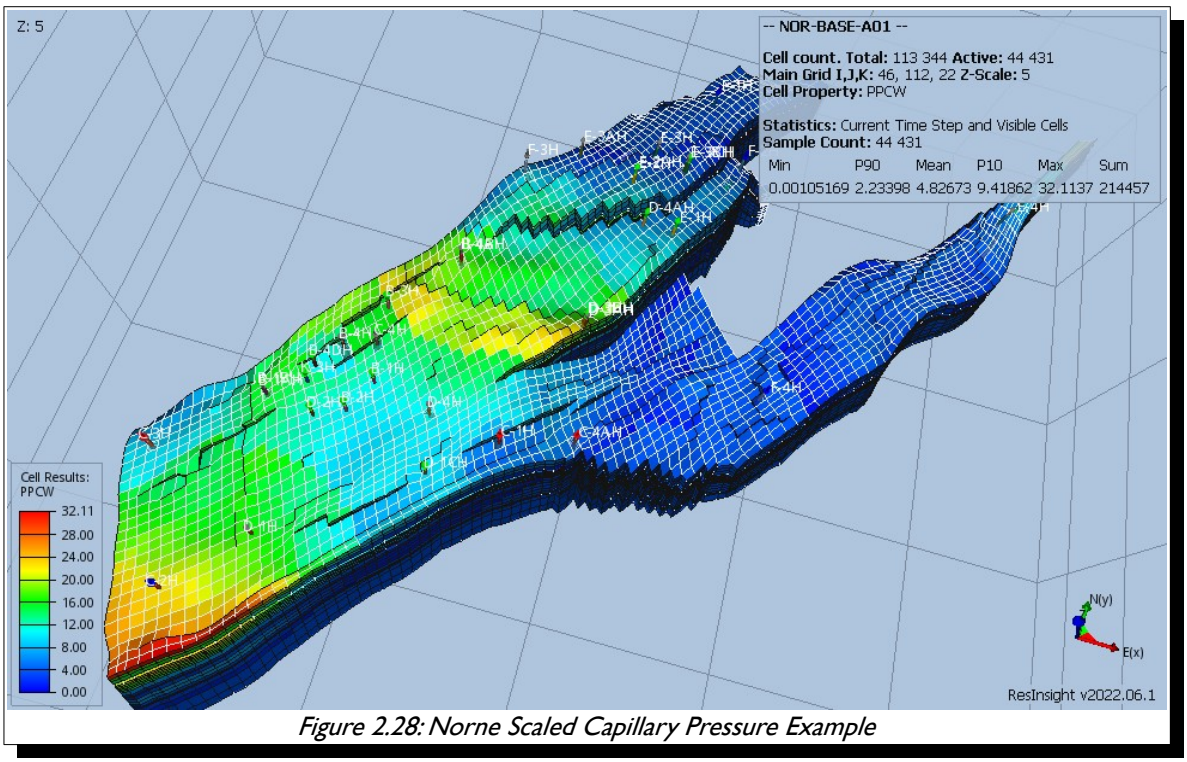


Figure 2.28: Norne Scaled Capillary Pressure Example

Note that although the PPCWMAX keyword can be used to set a maximum scaled capillary pressure value, a more technically sound approach is to resolve these inconsistencies before continuing with the model build.

Once a model is built and can be initialized, it is good practice to run the model with no production for several years to ensure that the initialization is stable. There should be under these circumstances no, or very limited fluid movement, and the pressure should remain stable, that is constant, although a small drift may be observable. An example for the Norne model is shown in Figure 2.29 on the field level, although it can, and should be performed on a region by region basis if the model is not stable.

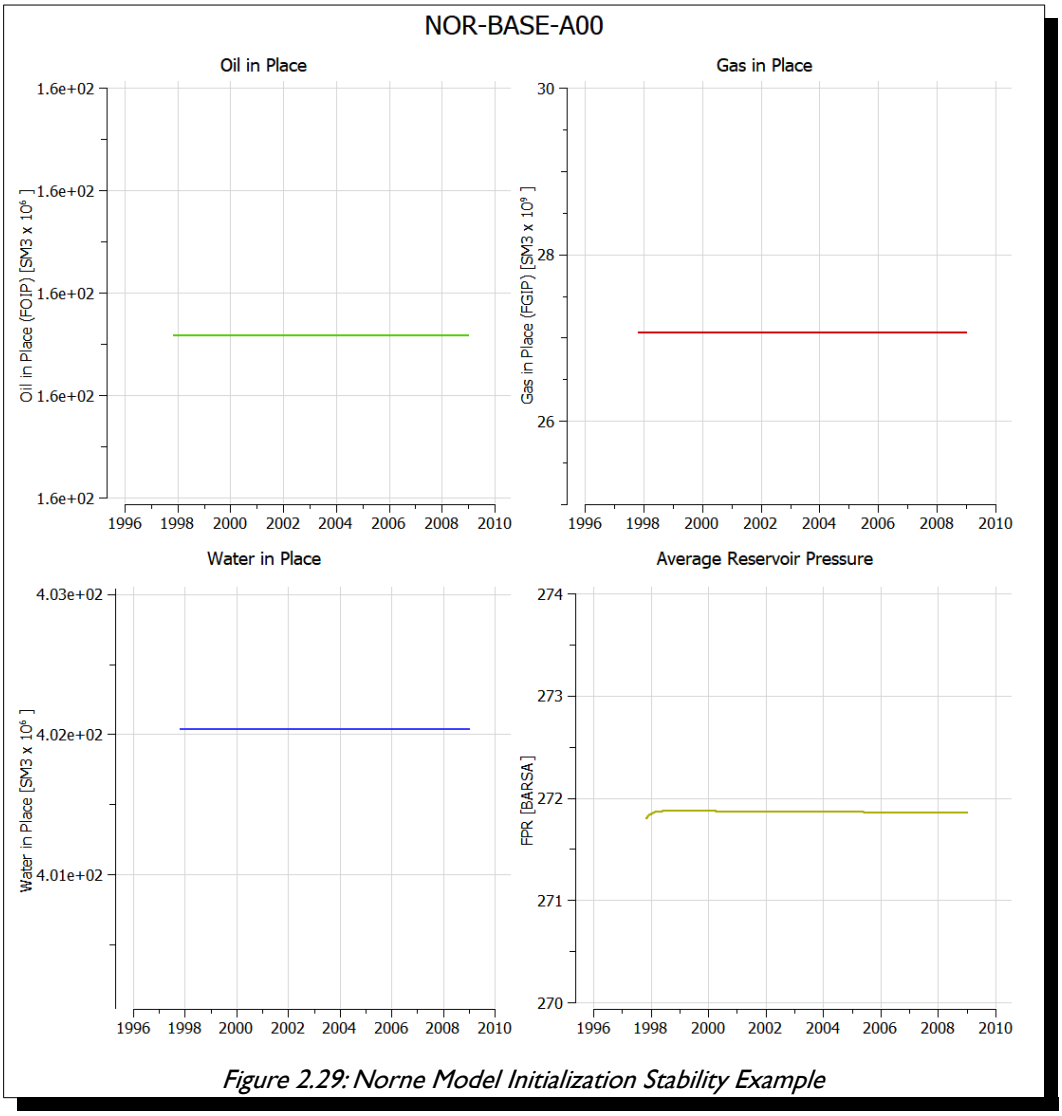


Figure 2.29: Norne Model Initialization Stability Example

Potential causes for the instability are:

1) Aquifer Not in Equilibrium with the Connected Hydrocarbon Zones. One can check for this by switching off all the aquifer connections and see if the instability is still present. If this corrects the issue then verify the aquifer connections by reviewing the AQUIFERA array in the *.INIT file in OPM ResInsight. OPM Flow writes out the AQUIFERA value to $2^{(AQUID-1)}$ for cells connected to aquifer AQUID. If a cell is connected to multiple analytical aquifers then AQUIFERA is summed for all aquifers connected to a cell. Note that connecting cells to multiple aquifers should be avoided.

   Note aquifers should only be connected to grid blocks containing 100% water, or at the very least any hydrocarbons in the connected cells should be immobile.

2) Different PVT Regions in Communication. Recall that fluid proprieties are associated with a grid cell, so if two different PVT regions are in communication, then a m^3 of fluid from PVT region one flowing into PVT region two, will automatically assume the properties of PVT region two. This creates a material balance error that will result in numerical difficulties. One may be able to identified this by reviewing the inter-region flows to identify the source of the problem, either via the *.PRT file or the summary file out. It typically occurs across fault planes, when one PVT region butts against a different PVT region along the fault face.

   There are several solutions to this, firstly, one can set the transmissibility along the fault plane to zero using the MULTFLT keyword in the GRID or EDIT sections.

   Secondly, create a MULTNUM array based on the PVTNUM array, as the PVTNUM keyword is not available in the GRID and EDIT sections. Then use the MULTREGT keyword with MULTNUM array in the GRID or EDIT sections to set the transmissibility between the offending regions to zero.

   Finally, one can use the THPRES keyword in the SOLUTION section combined with the EQLNUM array to isolate the offending regions. Normally, but not always, the PVTNUM and EQLNUM arrays are identical, if not, then it is relatively easy to copy the aforementioned MULTNUM array to the EQLNUM array, then add the additional equilibration data to the EQUIL keyword. One advantage of this option is that it will cater for the scenario for when the PVT regions are initially isolated, but after a certain production period, the faults break down and there is communication between the two regions. In this scenario, one still has the material balance error as different fluids inter-mix. If this is the case then the API Tracking option should be used to account for this behavior. See the API keyword in the RUNSPEC on how to activate this option.

3) Different Equilibration Regions in Communication. This is similar to (2) above and the remedies are also similar, so the solution is not repeated here.

4) PVT Data Not Consistent with Equilibration Data. In an oil-gas system the saturation pressure, that is the bubble-point pressure should be the pressure at the GOC, and similarly for the dew point. Conveniently, OPM ResInsight can produce such plot, as depicted in Figure 2.30. The figure illustrates that gas gradient (shown in red), and the oil gradient (shown in green) from the PVT data intersect at the GOC, which matches the initial pressure profile (shown in blue) from the grid cells. Thus, confirming the consistency of the two data sets.

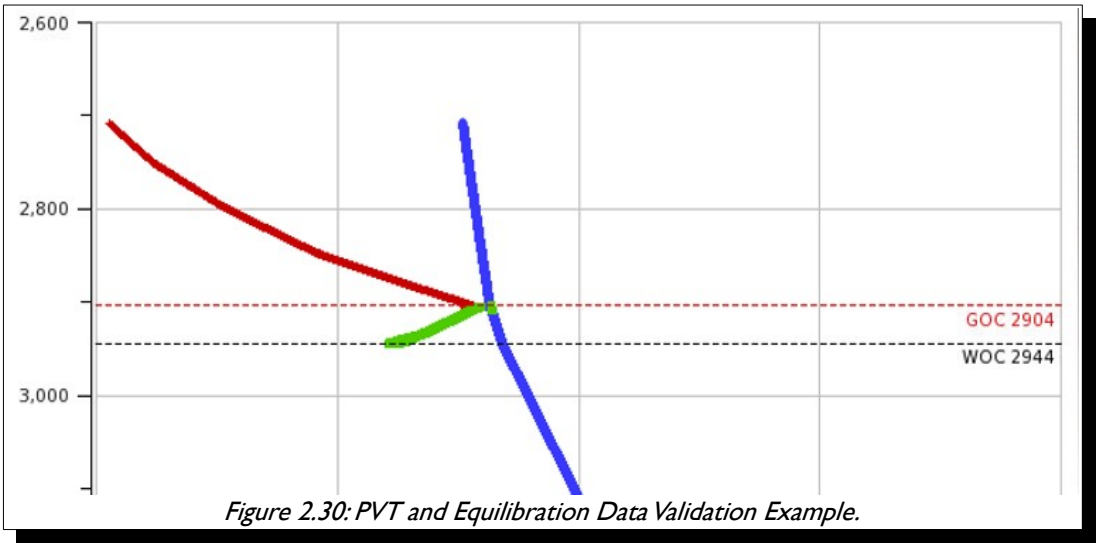


Figure 2.30: PVT and Equilibration Data Validation Example.

SCHEDULE Section

A common issue in most simulation models that employ Vertical Lift Performance ("VLP") tables used to relate a well's Tubing Head Pressure ("THP") to its Bottom-Hole Pressure ("BHP"), is the consistency of the data. Note that this is not just limited to wells, but also the network pipelines as well, that describe the pressure drop between the inlet node and the outlet node.

```
Warning: VFPPROD table 39 has 5 non-monotonic points of BHP(THP)
In /media/OPM/Fields/Norne/Model/INCLUDE/VFP/B3H.Ecl line 68
This may cause convergence issues due to switching between BHP and THP control.
```

Generally, the cause of this warning is because the VFP tables have been generated with too much fidelity or resolution. Users often consider that having more detailed and higher resolution VFP tables will give more accurate results. This is not always the case and in this particular example, will very well end up causing the wells using this particular VFP table oscillating between control modes. Figure 2.31 shows the VFP header section of the offending table, that declares the values used to generate the table.

```
VFPPROD

-- Table   Datum Depth   Rate Type   WFR Type   GFR Type   TAB Type
-- -----   -----------   ---------   --------   --------   --------
--    39       2625.13       'LIQ'      'WCT'      'GOR'           /


-- Prosper files are corrected from RKB to MSL depth. lmarr

-- Table   Datum Depth   Rate Type   WFR Type   GFR Type   TAB Type
-- -----   -----------   ---------   --------   --------   --------
      39       2603.13       'LIQ'      'WCT'      'GOR'           /

-- 'LIQ' units - SM3/DAY
  200.0   500.0  1000.0  1500.0  2000.0
 2500.0  3000.0  3500.0  4000.0  4500.0
 5000.0  5500.0  6000.0  6500.0  7000.0
 7500.0  8000.0 10000.0 14000.0 /

-- 'THP' units - BARSA
  20.00   50.00   60.00   80.00  100.00
 120.00  140.00  160.00  180.00  200.00 /

-- 'WCT' units - FRACTION
      0     0.1     0.2     0.3     0.4
    0.5     0.6     0.7     0.8       1 /

-- 'GOR' units - SM3/SM3
     90     100     150     300     500
   1000    2000 /

-- 'ALQ' units -
      0 /
```

Figure 2.31: VFP Table Header Example

At first glance this looks reasonable, but the problem is that the variables (LIQ, THP, WCT and GOR) are at a too high resolution, which causes some of the resulting curves to cross, due to the numerical accuracy when generating the curves. In addition, note that the maximum water cut is defined as 100%, which is unlikely to occur in practice, perhaps 98% would be more appropriate as the maximum value. OPM ResInsight is able to load VFPPROD and VFPINJ tables and to plot the data Here, Figure 2.32 shows a plot of

BHP versus THP with the various water cut values represented by the individual curves. The figure clearly illustrates the crossing of the water cut line of 80% over the 70% water cut curve, as well as the issue with the 100% water cut curve.

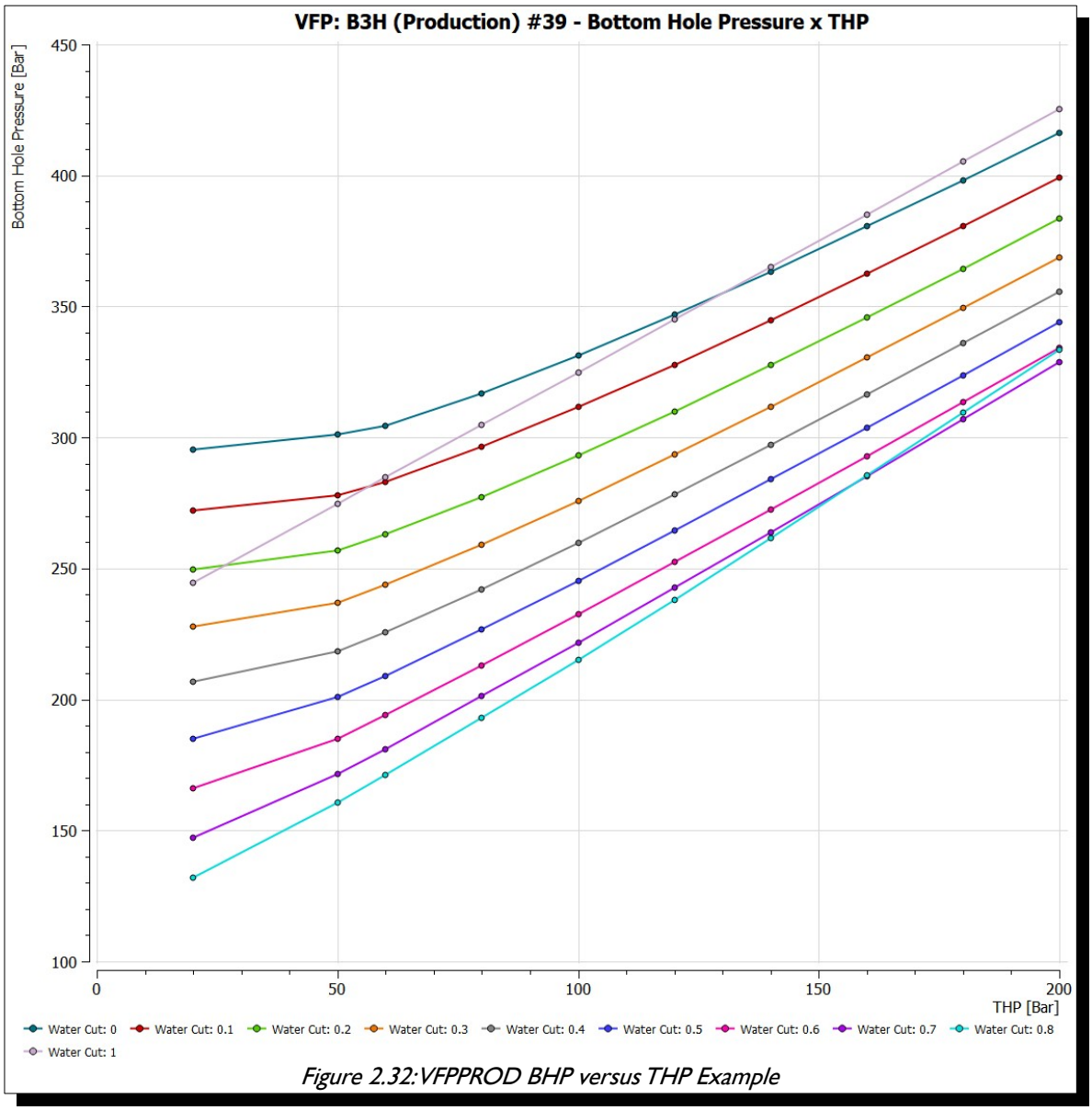


Figure 2.32: VFPPROD BHP versus THP Example

The solution is to re-generate the offending VFP tables using a coarser set of variable values. Petroleum Experts' PROSPER[53] program, as well as other nodal software programs, used to generate VFP tables for simulators, have options to generate the parameters based on the number of entries, and the minimum and maximum values for a variable, combined with various interval spacing options. Keeping the same range and selecting four or five entries and using the geometric spacing option normally resolves this type of issue. The simulator will then interpolate the coarser curves and avoid any issues with crossing curves. Note it is important to ensure that the variable ranges cover the full range of values expected to occur during the simulation, as extrapolation, unlike interpolation of the curves, is more prone to be problematic.

[53] *IPM Prosper Version 16.5 User Manual - November 2020, Petroleum Experts, Edinburgh, Scotland (www.petex.com).*

Note that this warning message can be safely ignored:

```
Warning: LIQ preferred phase not supported for well OP-01, using OIL instead
```

as it refers to the reference phase in a well's productivity index calculation. This is because OPM Flow only currently supports OIL, GAS and WATER well type fluids, as defined by the WELSPEC(TYPE) parameter in the SCHEDULE section, e.g., liquid (LIQ) is not supported. For producing, wells this mostly matters if one plots the WPI summary vector (productivity index for well's preferred phase). In the current treatment WPI will not have contributions from the water phase if the declared preferred phase is LIQ. Thus, if history matching a well's actual productivity index, this should be done on one of the recognized fluid well types. For injecting wells WELSPECS's preferred phase does not matter, since the preferred phase is (typically) reset to the injected phase via the WCONINJE and WCONINJH keywords.

2.4.2 CONVERGENCE AND NUMERICAL PERFORMANCE

Given that all the potential issues with the input data have been successfully resolved, there may still be problems with convergence and numerical performance, due to the nature of the problem the simulator is trying to solve. In Figure 2.33, at first glance, the non-linear iterations would appear to be reasonable, ranging from one to four, combined with linear iterations averaging around ten per time step. However, the time steps are very small resulting in nearly 12,500 Newton iterations and over 60,000 linear iterations in the commercial simulator, for this relatively simple problem. In fact, at the time of the study[54], the commercial simulator was unable to complete this simulation. Re-running the model with the latest version of the commercial simulator (Figure 2.33), still gives incorrect results compared to the published data.

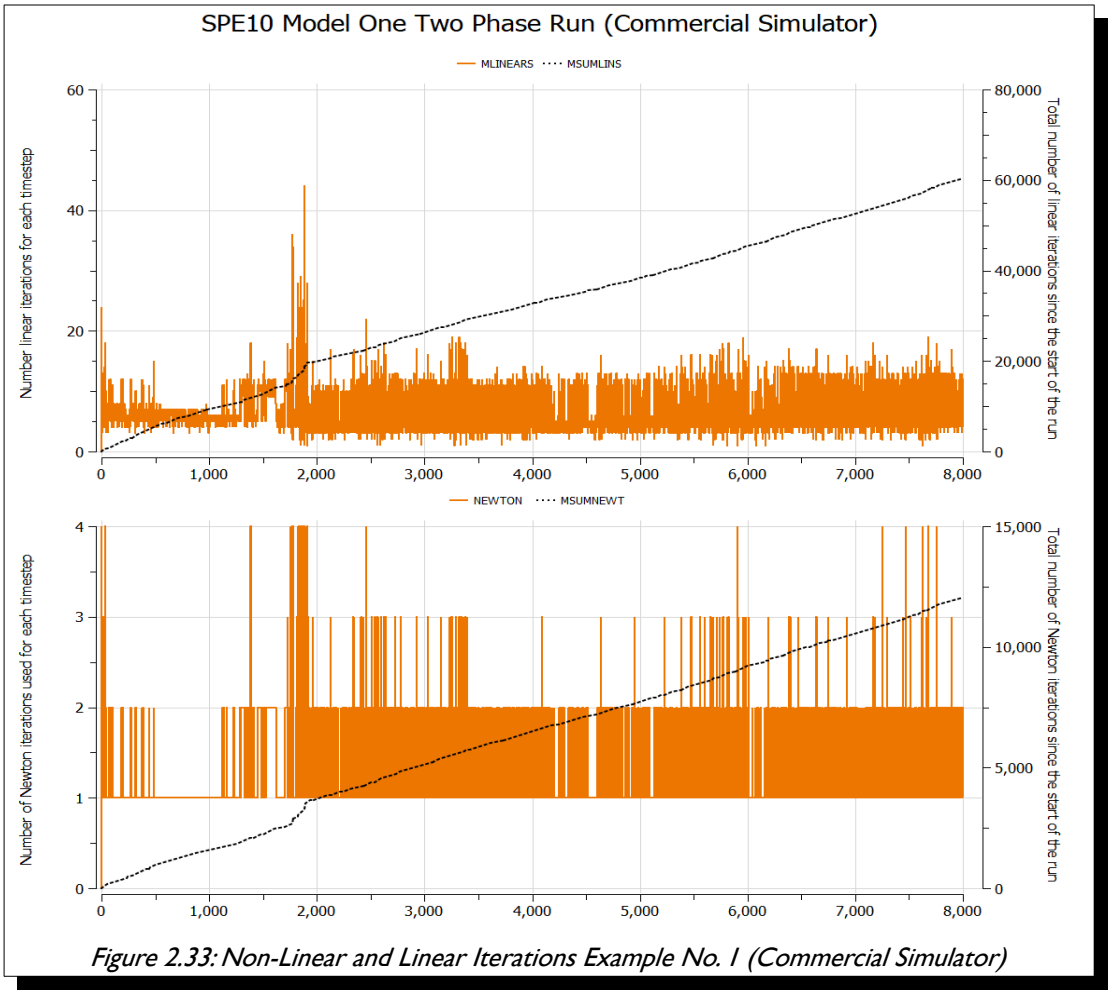


Figure 2.33: Non-Linear and Linear Iterations Example No. 1 (Commercial Simulator)

Fortunately, there are several ways to remedy poor performance, including:

1) The simplest and common approach is to manually restrict the time step size to reduce the number of time steps chops. Time step chops occurs when the simulator fails to converge at a given time step (t_{n+1}), and therefore restarts at the last successful time step (t_n) and advancing to the next time step using a smaller time step size (Δt_{n+1}) than before (Δt_n.). This was the approach used for the example in Figure 2.33, which although the model now runs, the results are incorrect.
2) Alternatively, one can change the numerical controls for the Newton-Raphson ('Newton" or non-linear solver) iteration, by changing the minimum and/or maximum number of Newton iterations. Although in general this may resolve some time step chops it can also result in a significant increase in run times, and should therefore be used with caution.

[54] *Christie, M.A., and M.J. Blunt. "Tenth SPE Comparative Solution Project:A Comparison of Upscaling Techniques." SPE Res Eval & Eng 4 (2001): 308–317. doi: https://doi.org/10.2118/72469-PA.*

3) The last option is to change the numerical parameters for the linear solver, which is generally preferable to modifying the Newton iteration parameters. OPM Flow contains various options in terms of the type of linear solver and the linear solver preconditioner to be employed, in addition to various standard linear solver parameters employed by the commercial simulator (minimum, maximum number of iterations etc.).

For example, Figure 2.34 illustrates the impact of selecting the more appropriate numerical controls. Here, the number of Newton iterations are greatly reduced to around 2,500 at the expense of increasing the number of linear iterations from ~60,000 to over 100,000. More importantly, the simulator results are correct, in that they are close to the published oil, gas and pressure values54.

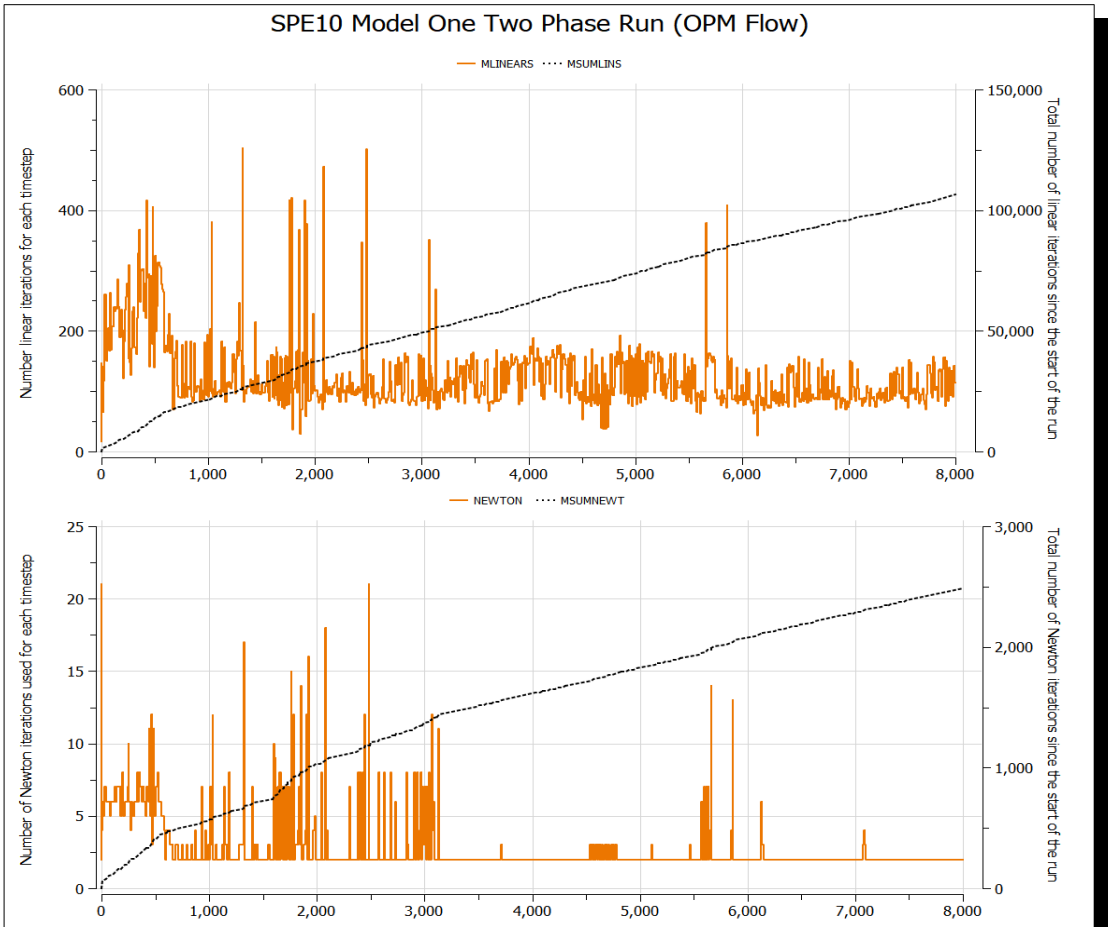


Figure 2.34: Non-Linear and Linear Iterations Example No. 2 (OPM Flow)

Note that the simulator has various other numerical performance parameters than can use used to identify the cause for under performance.

Note

To obtain the numerical performance data, the easiest way is to add the PERFORMA keyword in the SUMMARY section, that enables the writing out of a series of performance keywords to the summary file, which can then be plotted in OPM ResInsight as per Figure 2.33 and Figure 2.34.

In addition to the aforementioned PERFORMA keyword in the SUMMARY section, which is available for both OPM Flow and the commercial simulator, OPM Flow has two additional numerical performance reports that may help in identifying numerical issues. Both of these OPM Flow specific reports tabulate the numerical performance data available in the print file, in an easily readable format.

The first is the INFOSTEP report illustrated in Figure 2.35, which summarizes the numerical performance for the each time step. The Assembly, LSetup, LSolve, Update and Output header columns represent the time in seconds taken to complete the particular work item.

Time(day)	TStep(day)	Assembly	LSetup	LSolve	Update	Output	WellIt	Lins	NewtIt	LinIt	Conv
0	1	0.6386	0.1198	0.3741	0.2625	0.0000	0	3	2	7	1
1	3	0.3087	0.1457	0.5463	0.2510	0.0000	0	3	2	13	1
4	9	0.9498	0.5351	3.7574	0.9578	0.0000	0	9	8	106	1
13	4.254916684	0.3860	0.1832	1.6576	0.2900	0.0000	0	4	3	65	1
17.25491668	6.872541658	0.2838	0.1606	1.4545	0.2761	0.0000	0	5	4	71	1
24.12745834	6.872541658	0.2525	0.1586	1.6484	0.2652	0.0000	0	5	4	82	1
31	17.86860831	0.5594	0.4053	4.8491	0.6507	0.0000	0	11	10	246	1
48.86860831	10.13139169	0.2064	0.1194	1.7745	0.1998	0.0000	0	4	3	93	1
59	9	0.2306	0.1184	1.7544	0.2053	0.0000	0	4	3	92	1
68	20	0.5668	0.4067	3.2338	0.6546	0.0000	0	11	10	154	1
88	2	0.2300	0.1172	0.8394	0.2038	0.0000	0	4	3	39	1
90	6	0.2746	0.1595	1.5017	0.2674	0.0000	0	5	4	74	1
96	10.5	0.2075	0.1204	2.0434	0.2059	0.0000	0	4	3	108	1
106.5	10.5	0.2514	0.1579	2.2653	0.2758	0.0000	0	5	4	118	1
[...]											
821	3	0.1860	0.0903	0.6444	0.1439	0.0000	0	3	2	28	1
824	9	0.1844	0.0946	0.9104	0.1476	0.0000	0	3	2	42	1
833	11	0.1697	0.0897	0.7902	0.1406	0.0000	0	3	2	40	1
844	7	0.1916	0.0787	0.6669	0.1406	0.0000	0	3	2	33	1
851	1	0.2518	0.1192	0.6462	0.2108	0.0000	0	4	3	27	1
852	1.5	0.2534	0.1351	0.8762	0.2114	0.0000	0	4	3	37	1
853.5	1.5	0.2963	0.1737	1.3702	0.2844	0.0000	0	5	4	58	1
855	3.9	0.3004	0.1674	1.2472	0.2656	0.0000	0	5	4	57	1
858.9	6.1	0.1650	0.0812	0.9383	0.1401	0.0000	0	3	2	48	1
865	17	0.3542	0.2030	2.4171	0.3440	0.0000	0	6	5	117	1
882	26	0.2835	0.1740	2.2682	0.2740	0.0000	0	5	4	112	1
908	3	0.1963	0.0865	0.5798	0.1478	0.0000	0	3	2	26	1
911	1	0.1783	0.0824	0.5195	0.1393	0.0000	0	3	2	23	1

Figure 2.35: INFOSTEP Report Example

Here, the Assembly column represents the time taken to assemble the equations, *LSetup* is the time taken to setup the linear solver equations, *LSolve* is the linear solver time, *Update* is the time taken in updating the results, and *Output* is the time used for writing out the results, that is the output to the print, debug, summary, and restart files.

The *WellIt*, *Lins*, *NewtIt*, and *LinIt* column headers represent the number of iterations used to complete the time step for a given work item. Thus for time step one, the number of well iterations was zero (*WellIt*), the number of times the linear equations were setup was three (*Lins*), the number of Newton iterations used to complete the time step was two *(NewIt*), and the total number of linear iterations to completed the time step was seven (*LinIt*). The last column, *Conv*, indicates if the time step had successfully converged given the current convergence criteria, with a one indicating success and a zero indicating a failure.

By default the report is not generated to maintain compatibility with commercial simulator. To generate the report one can use the --output-extra-convergence-info command line parameter, for instance:

```
flow --output-extra-convergence-info="steps" CASENAME
```

will write the report to the CASENAME.INFOSTEP file in the same directory as the other output files.

The second report, INFOITER, contains more detailed information by reporting the numerical performance per iteration, as can be seen in Figure 2.36.

ReportStep	TimeStep	Time	Iteration	MB_Oil	CNV_Oil	MB_Water	CNV_Water	MB_Gas	CNV_Gas	WellStatus
82	0	9.1200e+02	0	2.3284e-05	4.5422e-02	1.1767e-05	2.5912e+01	9.7662e-06	7.4694e-02	CONV
82	0	9.1200e+02	1	2.9654e-07	3.8767e+00	2.1806e-07	4.1394e+00	5.2375e-08	2.9183e+01	CONV
82	0	9.1200e+02	2	2.2179e-07	6.8047e-01	3.0610e-07	1.1331e+00	9.1907e-08	3.8215e+00	CONV
82	0	9.1200e+02	3	7.9218e-08	2.1570e-01	1.0786e-07	9.9523e-01	3.6969e-08	2.5988e-01	CONV
82	0	9.1200e+02	4	1.0330e-09	1.3516e+00	3.9736e-09	7.8332e-02	1.6815e-09	4.7527e-01	CONV
82	0	9.1200e+02	5	2.0293e-10	8.8220e-03	6.3183e-10	8.3824e+00	2.2348e-10	2.3677e-02	CONV
82	0	9.1200e+02	6	3.4981e-11	3.0447e-03	7.7930e-11	7.1824e-01	5.5410e-11	1.0453e-03	CONV
82	0	9.1200e+02	7	2.5736e-12	1.2814e-03	2.4878e-12	6.9697e-02	3.7422e-11	5.9990e-04	CONV
82	0	9.1200e+02	8	9.4203e-13	1.5313e-04	1.7662e-12	6.2323e-03	2.7139e-12	5.2559e-05	CONV
[...]										
101	0	1.4610e+03	1	2.4697e-04	7.0125e+00	1.0699e-06	9.3247e+00	1.0367e-04	4.1111e+01	FAIL
101	0	1.4610e+03	2	2.3446e-04	7.6941e+00	1.1321e-06	4.3870e+00	9.9849e-05	3.2322e+00	CONV
101	0	1.4610e+03	3	2.4519e-04	2.4764e+00	1.1091e-07	1.1195e+00	1.0521e-04	1.8365e+01	FAIL
101	0	1.4610e+03	4	2.2569e-04	5.2145e+00	8.3692e-06	2.0409e-01	1.0160e-04	2.1750e+00	CONV
101	0	1.4610e+03	5	3.9522e-06	1.4566e+00	4.4403e-07	1.1981e+00	9.6223e-06	1.9711e+01	CONV
101	0	1.4610e+03	6	8.3787e-06	8.9540e+00	5.0083e-07	5.7035e-01	3.5820e-06	3.2819e+00	CONV
101	0	1.4610e+03	7	2.2669e-06	1.7800e+00	1.8097e-07	7.8105e-02	9.4668e-07	2.6049e+00	CONV
101	0	1.4610e+03	8	5.0458e-07	3.3687e-01	1.1915e-07	8.3545e-03	7.3566e-07	3.1211e+00	CONV
101	0	1.4610e+03	9	2.4234e-07	4.6142e-02	1.0972e-07	9.1612e-04	3.8631e-07	6.6698e-01	CONV
101	0	1.4610e+03	10	8.8876e-07	1.2618e-02	2.2543e-08	1.7938e-04	3.8122e-07	9.8301e-02	CONV
[...]										
113	0	1.8260e+03	0	3.9049e-04	3.4754e+00	1.6306e-04	2.5447e+00	1.9202e-04	1.5925e+00	CONV
113	0	1.8260e+03	1	2.5609e-04	1.8427e+01	1.1756e-04	3.4471e+01	7.9326e-05	2.6651e+02	FAIL
113	0	1.8260e+03	2	1.6074e-04	3.0898e+02	2.8540e-05	1.7710e+01	7.1727e-05	1.0152e+02	FAIL
113	0	1.8260e+03	3	2.7808e-05	5.5706e+00	1.0439e-05	1.6193e+01	1.3878e-05	1.9967e+01	FAIL
113	0	1.8260e+03	4	2.0310e-05	2.1713e+00	1.1343e-05	4.2916e+00	9.1809e-06	1.7611e+01	FAIL
113	0	1.8260e+03	5	1.2393e-05	6.9098e+00	7.1964e-06	9.3729e-01	9.0662e-07	6.3514e+00	FAIL
113	0	1.8260e+03	6	3.8059e-06	2.1091e+00	4.5883e-07	3.1448e-01	1.3182e-06	1.3689e+01	FAIL
113	0	1.8260e+03	7	1.0520e-07	6.2311e-01	2.2739e-07	2.2595e-01	1.4728e-07	3.4982e+00	FAIL
113	0	1.8260e+03	8	3.6760e-09	1.0626e-01	1.1834e-08	5.7336e-02	2.1837e-08	6.9609e-01	FAIL
113	0	1.8260e+03	9	1.4687e-08	1.8172e-02	5.5110e-08	2.2440e-03	7.8427e-09	8.4317e-03	FAIL
113	0	1.8260e+03	10	1.8306e-09	3.1723e-05	5.4198e-09	6.2984e-06	1.3070e-09	1.9392e-05	FAIL
113	0	1.8260e+03	11	3.8017e-12	7.4210e-09	3.4998e-12	1.7551e-08	1.0524e-12	5.5179e-09	FAIL
113	0	1.8260e+03	12	3.7820e-15	3.5240e-10	2.6918e-15	2.6820e-08	2.5483e-15	1.9588e-09	FAIL
113	0	1.8260e+03	13	3.2360e-16	2.8521e-10	6.4618e-16	1.2857e-08	1.8377e-16	9.8643e-10	FAIL
113	0	1.8260e+03	14	2.5688e-16	2.8569e-10	6.0128e-16	2.6820e-08	1.4191e-16	1.1094e-09	FAIL
113	0	1.8260e+03	15	1.9524e-16	3.4905e-10	5.1244e-16	2.6820e-08	1.1677e-16	1.1094e-09	FAIL
113	0	1.8260e+03	16	1.6093e-16	3.4858e-10	4.7598e-16	1.2601e-08	8.9227e-17	9.8642e-10	FAIL
113	0	1.8260e+03	17	1.0243e-16	2.8600e-10	4.0966e-16	2.6820e-08	6.6551e-17	8.6481e-10	FAIL
113	0	1.8260e+03	18	6.4405e-17	3.4864e-10	3.8223e-16	1.2993e-08	4.2039e-17	9.8642e-10	FAIL
113	0	1.8260e+03	19	5.5481e-17	2.8647e-10	3.2872e-16	1.2601e-08	3.5436e-17	8.6481e-10	FAIL
113	0	1.8260e+03	20	4.8745e-17	2.8681e-10	3.0035e-16	1.2601e-08	3.0282e-17	9.8642e-10	FAIL
[...]										
113	0	1.8260e+03	0	5.2653e-06	1.2490e-01	8.6797e-06	9.1448e-02	2.9507e-06	5.7228e-02	FAIL
113	0	1.8260e+03	1	8.7525e-06	9.3325e-02	2.8342e-06	1.0308e-01	3.8617e-06	1.7602e-01	CONV
113	0	1.8260e+03	2	2.3667e-08	1.2853e+00	8.1819e-09	6.4745e-01	6.3189e-08	4.6300e-01	CONV
113	0	1.8260e+03	3	4.8937e-09	1.8799e-02	3.8632e-10	3.6341e-02	4.9590e-09	5.1495e-03	CONV
113	0	1.8260e+03	4	2.0833e-11	1.9696e-02	1.4980e-10	8.5118e-03	8.1549e-11	7.2504e-03	CONV
113	0	1.8260e+03	5	3.2286e-11	5.3966e-05	5.9882e-11	6.5047e-05	2.2947e-11	1.2951e-05	CONV
113	1	1.8276e+03	0	2.8208e-05	1.2664e-01	1.1830e-05	7.7499e-02	1.4066e-05	4.3010e-02	CONV

Figure 2.36: INFOITER Report Example

Here, the *Iteration* column contains the number of times the linear equations were setup, same as the Lins column in the INFOSTEO report. Note, subtracting one from the values in the *Iteration* column, gives the number of Newton iterations. Columns containing *MB* indicate a phase's normalized material balance error, and columns containing *CNV* show a phase's normalized residual error. The last column, *WellStatus*, indicates if the iteration converged or failed. Thus, report step 82 converged in seven Newton iterations with all solutions converging; whereas, report step 101 took ten iterations with two failures.

Like the INFOSTEP report, the INFOITER report by default is not generated in order to maintain compatibility with commercial simulator. To generate the report one can use the --output-extra-convergence-info command line parameter, for instance:

```
flow --output-extra-convergence-info="iterations" CASENAME
```

will write the report to the CASENAME.INFOITER file in the same directory as the other output files. Finally to generate both reports use the following command line option.

```
flow --output-extra-convergence-info="stepsiterations" CASENAME
```

The following sections now attempt to give some guidance on how use the numerical controls to resolve the simulator's performance, assuming that all the issues with the input data have been successfully resolved.

2.4.3 TIME STEP CONTROL GUIDANCE

Figure 2.37 illustrates how the simulator advances from the start of the simulation to the end, and shows two user defined report steps, each containing two simulator selected time steps. Each time step within a report time step, contains one to three Newton iterations, and each Newton iteration contains a various number of linear iterations.

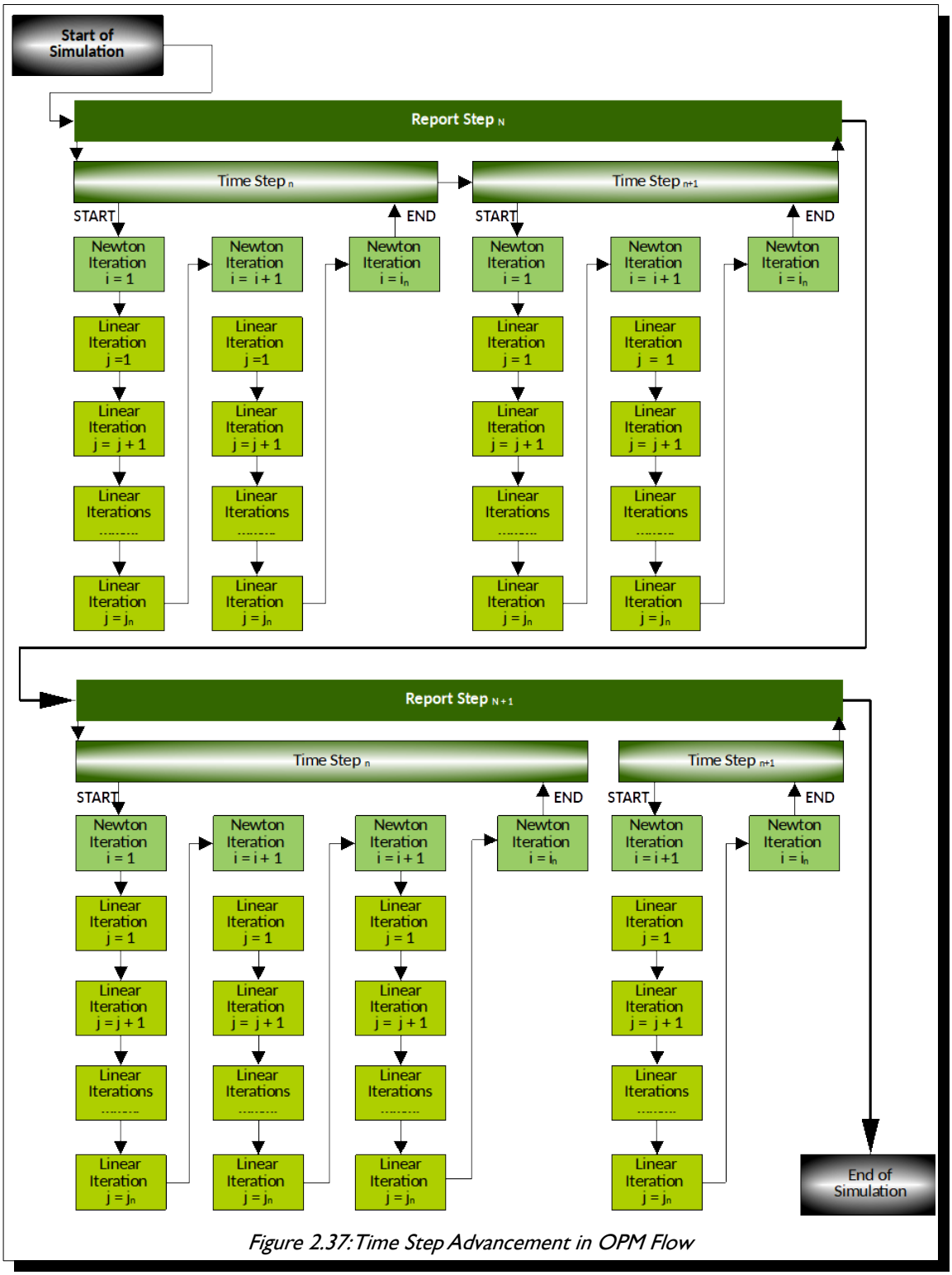


Figure 2.37: Time Step Advancement in OPM Flow

In an actual model the number of time steps are determined by the simulator's internal adaptive time stepping algorithm, combined with any user settings overriding the internal stepping algorithm and parameters. The default initial time step of one day can overwritten via either the command line parameter:

```
--initial-time-step-in-days
```

or via the *TUNING - Numerical Tuning Control* keyword in the SCHEDULE section, that offers more flexibility, as the initial time step size can be reset at various report time steps, by repeating the keyword and setting TUNING(TSINIT) to a given value. Alternatively, TUNING(TSMINZ) may be used to define the minimum length of all time steps. Note, however, the simulator only reads the first record of the TUNING keyword if, and only if, the following command line parameter has been activated:

```
--enable-tuning
```

One can also set the maximum time step size via both the command line parameter:

```
--solver-max-time-step-in-days
```

or via the TUNING(TSMAXZ) parameter, that defines the maximum length of the next time step following TUNING(TSINIT).

There are many other time stepping command line parameters, including those that are equivalent to all the first record parameters on the TUNING keyword; however, provided there are no data inconsistencies within the input deck, resetting the maximum time step size from the default value of 365 days will improve numerical performance. Typically the default value of 365 days is not binding, as the report time steps are more restrictive. So for example, in a history matching model, the production data is normally entered on a monthly basis, so the maximum report time step size would be 31 days. Even so, time step chops may still occur even in this scenario, and therefore reducing the maximum time step to half a month, that is 16 days may avoid time step chops. Sixteen days is preferred over 15 days as the maximum time step, because it enables only two successful time steps to complete a 31 day month (a 16 day plus a 15 day time step). If 15 days was used for the maximum time step size, then one would need three successful time steps to complete a 31 day month. In some cases using 11, or even a five day maximum time step may increase computational efficiency by avoiding expensive computational time step chops.

Basically, one has to balance the maximum length of time steps with the required number of time steps to reach the end of the run, versus the total elapsed time[55]. Time steps chops are computationally expensive and thus should therefore be minimized in order to reduce run times.

Figure 2.38 shows a typical report step and time step report from the simulator that contains two time step chops shown in the red outline boxes. Here the simulator is at:

1) Report step 15 (Current Step) out of the 33 steps (End Step) that are are needed to complete the run. Report step 15 is at October 9th, 1967 and the simulator's internal adaptive time stepping algorithm has determined that the next time step is 50 days.

2) As requested, the simulator writes out a restart record at report step 15, that is October 9th, 1967.

3) The simulator then attempts to advance 50 days, but fails due to a *Numerical problem encountered*, that is either the non-linear or linear solver failed to convergence within the maximum number of iterations. Thus, the time step was chopped from 50 days to 16.5 days, and simulator restarts again from report step 15, that is October 9th, 1967 and attempts to advance to October 25th, 1967. The 16.5 day time step converges in three Newton iterations using a total of 342 linear iterations, and the simulator has advanced to October 25th, 1967. Thus, for the next time step the simulator's internal adaptive time stepping algorithm has determined that the next time step should be 33.5 days.

4) The simulator then attempts to advance 33.5 days, but again fails due to a *Numerical problem encountered*, that is either the non-linear or the linear solver again failed to convergence within the maximum number of iterations. Thus, the time step was chopped form 33.5 days to 11.055 days, and

[55] *Elapsed real time, real time, wall-clock time, or wall time is the actual time taken from the start of a simulation to the end. In other words, it is the difference between the time at which the simulation finishes and the time at which the simulation started.*

simulator restarts again for report step 15, but from the last successful time step, that is October 25th, 1967 and attempts to advance to November 5th, 1967.

5) The 11.055 day time step converges in three Newton iterations using a total of 317 linear iteration, and the simulator has advanced to November 5th, 1967. For the next time step, the simulator's internal adaptive time stepping algorithm has determined that the next time step should be 22.445 days to coincide with the next report time step (report step 16) at November 28th, 1967.

6) Again, the 22.445 day time step is successful and converges in three Newton iteration using a total of 318 linear iterations.

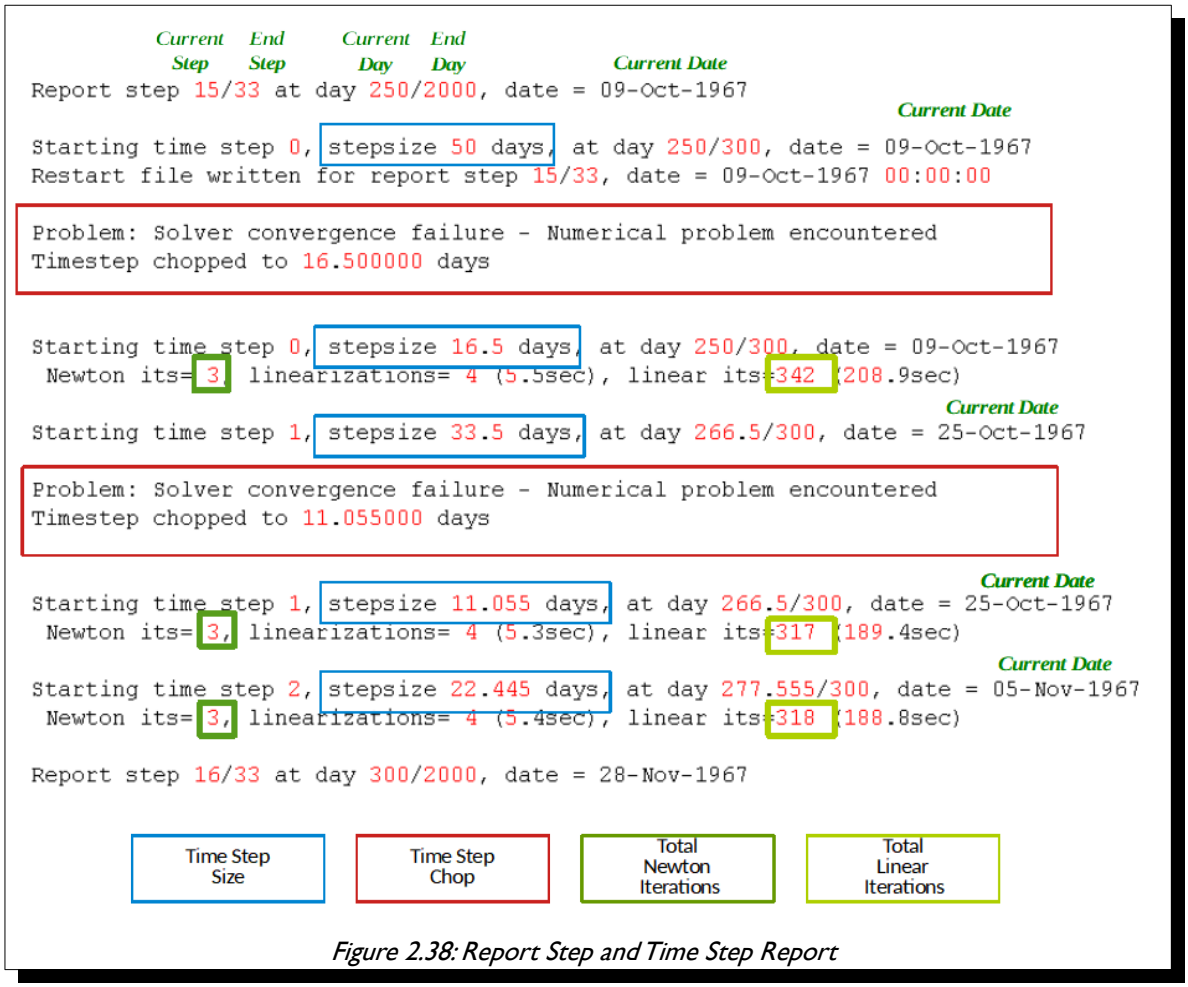


Figure 2.38: Report Step and Time Step Report

If the maximum time step was set to 16 days then advancing 50 days to November 28th, 1967 would take four time steps with a good chance of avoiding time step chops and thus reducing run times.

Various command line parameters can be used to control the behavior of the time stepping algorithm, as summarized in the Table 2.4, that lists the important time stepping control parameters for the simulator, including the TUNING equivalent parameters where applicable.

Major Solver Time Stepping Controls			
No.	**Command Line Parameter**	**TUNING Keyword Equivalent**	**Description**
1	--initial-time-step-in-days	TSINIT	A real double precision value that sets the size of initial time step in days. The default value is one day. Note that as the TUNING keyword can be entered multiple times, then TUNING(TSINIT) can be reset multiple times, whereas as *--initial-time-step-in-days*, is only set once at the beginning of the run.
2	--solver-growth-factor	TSDIFF	A real positive value that specifies the growth factor a time step can be increased by when recovering from one or more time step chops, subject to the maximum allowable time step size set by the *--solver-max-time-step-in-days* parameter. For example, if the current time step has converged at 5 days after a time step chop, and --solver-growth-factor is set to the default value of 2.0, then the next time step will be 2.0 * 5 days, that is at 10 days.
3	--solver-max-growth	TSFMAX	A real positive value that specifies the maximum growth factor a time step can be increased by after a report time step. subject to the maximum allowable time step size set by the --solver-max-time-step-in-days parameter. Thus, if the current time step has converged at 5 days after at a report step, and --solver-max-growth is set to the default value of 3.0, then the next time step will be 3.0 * 5 days, that is at 15 days.
4	--solver-max-restarts		A positive integer that defines the number of allowed consecutive restarts (or time step chops) before the simulation is terminated. Here the default value is 10. TUNING(TSMCHP) sets the minimum length of all chopped time steps.
5	--solver-max-time-step-in-days	TSMAXZ	A real positive double precision value that specifies the maximum allowed time step size in days. The default value of 365 days is really used in practise due to the report time steps being more restrictive, although in some specialized models (CO2 long term prediction cases) this may indeed be the upper limit. As mentioned previously, if the time steps chops are mostly occuring when the time step size is greater than Δt, then setting this parameter to a value less than Δt may reduce the number of time step chops. Provided of cause the value of Δt is reasonable.

6	--solver-min-time-step	TSMINZ	A real positive double precision value that specifie the minimum size of a time step in days for field and metric units, and hours for laboratory units If a time step cannot converge without getting cut below this time step size the simulator will stop, unless overwritten by the --solver-continue-on-convergence-failure command line parameter (see Table 2.1 for further details). Note if the current time step is smaller than this minimum and has not been subject to a time step chop, then in some circumstances it may still be accepted due to other numeric controls.
7	--solver-restart-factor	TSFCNV	A real positive double precision value that specifies the time step chop factor of the time step after a convergence failure. For example, if the current non-convergent time step is 30 days and this parameter is set to the default value of 0.33, then the time step will be repeated using 0.33 * 30 days as the time step, that is 9.9 days.
Notes: 1) Cells colored orange under the TUNING Equivalent Keyword column with no TUNING keyword parameter do not have an equivalent OPM Flow command line parameter.			

Table 2.4: Major Solver Time Stepping Controls

Note

OPM Flow also has a series of command line parameters of the form *--time-step** that are subordinate to the *--solver** series of command line parameters. The former, *--time-step**, are used to select the time step control algorithm and associated parameters, and in general should only be used by users with appropriate experience.

As mentioned above, the *--time-step** series of command line parameters should in general not be used; however, the following parameter may be useful if group and well events are resulting in time step chops:

```
--time-step-after-event-in-days
```

If the simulation is running smoothly with taking monthly historical report time steps, except for when a new well is bought on stream in a given month, or a group control is reset resulting in a significant change in production or injection, and the event causes time step chops, then using *--time-step-after-event-in-days* may improve the situation. The parameter can be used to automatically enforce a small time step when an event occurs, thus reducing the probability, or even eliminates a time step chop. Typically, a one or five day time step is used, as per:

```
--time-step-after-event-in-days=5.0
```

The drawback of cause is that control is applied to the complete run, which again needs to be taken into consideration.

Alternatively, one could use the NEXT or NEXTSTEP keywords in the SCHEDULE section, to incorporate similar behavior at specific times in the SCHEDULE section. See *NEXT – Maximum Next Time Step Size (Alias for NEXTSTEP)* or *NEXTSTEP – Maximum Next Time Step Size* for further information.

2.4.4 NEWTON-RAPHSON SOLVER ITERATION CONTROLS (NON-LINEAR ITERATIONS)

Figure 2.39 shows typical time step output for two report time steps, March 1st and April 1st, 2022; with both solving the equations with two Newton iterations per time step, and between four to six linear iterations per time step. The numerical count for the number of linearizations, is the number of times the Jacobian and the residuals have been constructed, and is generally one more than the number of Newton iterations. Figure 2.39, thus indicates very good performance, especially as the first iteration is taken from the previous successful time step. Note however, that the model is only using five day time steps in this case, which significantly contributes to the convergence stability of both the non-linear and linear solvers.

```
Report step  3/61 at day 59/1825, date = 01-Mar-2022

Starting time step 0, stepsize 5 days, at day 59/90, date = 01-Mar-2022
 Newton its= 2, linearizations= 3 (0.0sec), linear its=  5 (0.0sec)

Starting time step 1, stepsize 5 days, at day 64/90, date = 06-Mar-2022
 Newton its= 2, linearizations= 3 (0.0sec), linear its=  4 (0.0sec)

Starting time step 2, stepsize 5 days, at day 69/90, date = 11-Mar-2022
 Newton its= 2, linearizations= 3 (0.0sec), linear its=  4 (0.0sec)

Starting time step 3, stepsize 5 days, at day 74/90, date = 16-Mar-2022
 Newton its= 2, linearizations= 3 (0.0sec), linear its=  4 (0.0sec)

Starting time step 4, stepsize 5 days, at day 79/90, date = 21-Mar-2022
 Newton its= 2, linearizations= 3 (0.0sec), linear its=  4 (0.0sec)

Starting time step 5, stepsize 3 days, at day 84/90, date = 26-Mar-2022
 Newton its= 2, linearizations= 3 (0.0sec), linear its=  5 (0.0sec)

Starting time step 6, stepsize 3 days, at day 87/90, date = 29-Mar-2022

Report step  4/61 at day 90/1825, date = 01-Apr-2022

Starting time step 0, stepsize 5 days, at day 90/120, date = 01-Apr-2022
 Newton its= 2, linearizations= 3 (0.0sec), linear its=  5 (0.0sec)

Starting time step 1, stepsize 5 days, at day 95/120, date = 06-Apr-2022
 Newton its= 2, linearizations= 3 (0.0sec), linear its=  5 (0.0sec)

Starting time step 2, stepsize 5 days, at day 100/120, date = 11-Apr-2022
 Newton its= 2, linearizations= 3 (0.0sec), linear its=  4 (0.0sec)

Starting time step 3, stepsize 5 days, at day 105/120, date = 16-Apr-2022
 Newton its= 2, linearizations= 3 (0.0sec), linear its=  5 (0.0sec)

Starting time step 4, stepsize 5 days, at day 110/120, date = 21-Apr-2022
 Newton its= 2, linearizations= 3 (0.0sec), linear its=  6 (0.0sec)

Starting time step 5, stepsize 5 days, at day 115/120, date = 26-Apr-2022
```

Figure 2.39: Report Step and Time Step Summary Reporting Example

The number of Newton iterations taken to solve a time step is indicative of the degree of ease the simulator has in converging to a solution, with higher numbers of iterations indicating an increasingly difficult problem.

Note, unlike Figure 2.39, the number of non-linear iterations will vary as the simulation progresses, with periods of relatively efficient convergence and periods where the simulator is struggling to reach non-linear convergence. The latter is normally associated with operational events, for example new wells being brought on stream, major changes to group controls etc.

Note

As mentioned previously, in general the number of Newton iterations is indicative of the ease of convergence, with one to three iterations indicating that converge was relatively easy and efficient, four to nine iterations indicating an increasingly difficult problem, and greater than ten would tend to suggest that there may be an issue with the model.

It is important to stress that the convergence of the non-linear solver, is very dependent on the accuracy of the linear solver. If the accuracy of the linear solver is too relaxed or is not fully converged. For example, if the linear solver solution is not sufficiently converged, then the non-linear solution may no longer be well-posed[56]. That is the solution starts digressing from the true solution, and additional iterations only exasperate the problem. Normally, under these circumstances the OPM Flow will chop the time step, which should correct the divergence by reverting to previous converged time step. However, if the time step is too small to chop and the run continues with the minimum time step, then the divergence from the true solution will continue. This may manifest itself with various error messages and NaNs ("Not a Number"), or eventually with the simulator aborting. Figure 2.40 shows an example of this behavior.

[56] *A problem in differential equations is said to be well-posed if: (1) a solution exists; (2) that the solution is unique; (3) and the solution changes continuously with the changes in the data.*

```
Starting time step 0, stepsize 0.1089 days, at day 0/31, date = 01-Jan-2020
Problem: Solver convergence failure - Numerical problem encountered
Timestep chopped to 0.035937 days

Starting time step 0, stepsize 0.035937 days, at day 0/31, date = 01-Jan-2020
Problem: Solver convergence failure - Iteration limit reached
 Timestep chopped to 0.011859 days

Starting time step 0, stepsize 0.0118592 days, at day 0/31, date = 01-Jan-2020
Well OP01 will be shut because it cannot get converged.
Problem:
Problematic well(s) were shut: OP01 (retrying timestep)

Starting time step 0, stepsize 0.0118592 days, at day 0/31, date = 01-Jan-2020
 Newton its= 2, linearizations= 3 (0.2sec), linear its=  0 (0.0sec)

 Starting time step 1, stepsize 0.0237184 days, at day 0.0118592/31, date = 01-Jan-2020
flow: /home/david/OPM/GitHub/opm-simulators/opm/simulators/timestepping/TimeStepControl.cpp:140: virtual double
[david-VirtualBox:252949] *** Process received signal ***
[david-VirtualBox:252949] Signal: Aborted (6)
[david-VirtualBox:252949] Signal code:  (-6)
[david-VirtualBox:252949] [ 0] /lib/x86_64-linux-gnu/libc.so.6(+0x42520)[0x7f4c57af1520]
[david-VirtualBox:252949] [ 1] /lib/x86_64-linux-gnu/libc.so.6(pthread_kill+0x12c)[0x7f4c57b45a7c]
[david-VirtualBox:252949] [ 2] /lib/x86_64-linux-gnu/libc.so.6(raise+0x16)[0x7f4c57af1476]
[david-VirtualBox:252949] [ 3] /lib/x86_64-linux-gnu/libc.so.6(abort+0xd3)[0x7f4c57ad77f3]
[david-VirtualBox:252949] [ 4] /lib/x86_64-linux-gnu/libc.so.6(+0x2871b)[0x7f4c57ad771b]
[david-VirtualBox:252949] [ 5] /lib/x86_64-linux-gnu/libc.so.6(+0x39e96)[0x7f4c57ae8e96]
[david-VirtualBox:252949] [ 6] /home/david/OPM/GitHub/opm-simulators/build/bin/flow(+0x194c2f4)[0x556648c6f2f4]
[david-VirtualBox:252949] [ 7] /home/david/OPM/GitHub/opm-simulators/build/bin/flow(+0x194c33a)[0x556648c6f33a]
[david-VirtualBox:252949] [ 8] /home/david/OPM/GitHub/opm-simulators/build/bin/flow(+0x91c686)[0x556647c3f686]
[david-VirtualBox:252949] [ 9] /home/david/OPM/GitHub/opm-simulators/build/bin/flow(+0x91e9cd)[0x556647c419cd]
[david-VirtualBox:252949] [10] /home/david/OPM/GitHub/opm-simulators/build/bin/flow(+0x91fa3b)[0x556647c42a3b]
[david-VirtualBox:252949] [11] /home/david/OPM/GitHub/opm-simulators/build/bin/flow(+0x90dc2e)[0x556647c30c2e]
[david-VirtualBox:252949] [12] /home/david/OPM/GitHub/opm-simulators/build/bin/flow(+0x87c7f1)[0x556647b9f7f1]
[david-VirtualBox:252949] [13] /home/david/OPM/GitHub/opm-simulators/build/bin/flow(+0x26c6a5)[0x55664758f6a5]
[david-VirtualBox:252949] [14] /lib/x86_64-linux-gnu/libc.so.6(+0x29d90)[0x7f4c57ad8d90]
[david-VirtualBox:252949] [15] /lib/x86_64-linux-gnu/libc.so.6(__libc_start_main+0x80)[0x7f4c57ad8e40]
[david-VirtualBox:252949] [16] /home/david/OPM/GitHub/opm-simulators/build/bin/flow(+0x3902a5)[0x5566476b32a5]
[david-VirtualBox:252949] *** End of error message ***
Aborted (core dumped)
Process Complete (0)
```

Figure 2.40: Time Step Summary Showing Non-Convergence Example.

Fortunately, OPM Flow has various controls that can resolve these type of issues similar to the commercial simulator, and in addition, and unlike the commercial simulator, OPM Flow has several linear solvers and preconditioners that increase the likelihood a solution may be found for various non-convergence issues.

As mentioned previously in section 1.6 Numerical Solution of Equations, convergence is achieved when $R(y_n)$, the residual, in equation (2.5), satisfies a given tolerance.

$$J(y_n)(y_{n+1} - y_n) = -R(y_n) \tag{2.5}$$

Where:

$J(y_n)$ = the Jacobian matrix,

y_n = the solution of the non-linear equations at the nth non-linear iteration, and

y_{n+1} = the solution of the non-linear equations iteration at the n+1 iteration.

However, as the iterations proceed during a time step the residual convergence criteria is reduced in an effort to avoid a time step chop. Thus, by relaxing the convergence criteria as the non-linear iterations progress, the simulator is attempting to avoid throwing away all the previous work performed in trying to reach a solution for the time step.

From the user prospective, there are several command-line options that can be used to control the Newton-Raphson solver, in particular:

1) To modify the maximum number of non-linear iterations per time step, the following command line parameter can be used:

```
--newton-max-iterations
```

Although, the default value of 20 is sufficiently large, that it should not be necessary to increase this value.

2) There is also a command that sets the minimum number of Newton iterations per time step used by the simulator.

```
--newton-min-iterations
```

The default value of one ensures that at least one Newton iteration is performed after the previous time step. Increasing this parameter, to say two for example, may help convergence in difficult time steps, by ensuring slightly more accurate results in preceding "easy" time steps.

3) Thirdly, the mass balance error as a reservoir average saturation error, and can be use to slacked or tighten the material balance tolerance.

```
--tolerance-mb
```

In general, if the non-linear solver is having difficulty in converging, then tightening the linear mass balance error as a reservoir average saturation error, may help the non-linear solver converge more rapidly.

4) Fourthly, maximal local residual error can be slacked or tighten using:

```
--tolerance-cnv
```

command line parameter. And may be used to set the maximal local residual error.

For completeness, Table 2.5 lists the major non-linear command line parameters used to control the behavior of the non-linear solver, including those that relate to relaxed convergence criteria. However, in general it is recommend not modify these parameters, unless one has the sufficient expertise in this area.

Major Newton-Raphson Solver Controls			
No.	**Command Line Parameter**	**TUNING Keyword Equivalent**	**Description**
1	--newton-max-iterations	NEWTMX	A positive integer that defines the maximum number of Newton iterations per time step used by the simulator. The default value is 20 which compares to the commercial simulator's value of 12, so there should not really be a requirement to increase this parameter.
2	--newton-min-iterations	NEWTMN	A real positive integer that sets the minimum number of Newton iterations per time step used by the simulator. The default value of one ensures that at least one Newton iteration is performed after the previous time step. Increasing this parameter, to say two for example, may help convergence in difficult time steps, by ensuring slightly more accurate results in preceding "easy" time steps.

Major Newton-Raphson Solver Controls			
No.	**Command Line Parameter**	**TUNING Keyword Equivalent**	**Description**
3	--newton-max-relax		A real positive value that sets the maximum relaxation factor of a Newton iteration used by the simulator. The default is 0.5. In general, it is not recommended to modify this parameter.
4	--newton-relaxation-type		A character string that defines the type of relaxation used in Newton's method, the default is dampen. In general, it is not recommended to modify this parameter.
5	--max-newton-iterations-with-inner-welliterations	MXWSIT	A positive integer that specifies the maximum Newton iterations with inner well iterations. The default value is eight, the same as the commercial simulator. If there are issues with well convergence, then sometimes increasing this value, to say 12, may improve the situation.
6	--min-strict-cnv-ite		A positive integer that sets the minimum number of Newton iterations before relaxed tolerances are used for the CNV convergence criterion. The default value is zero, means that relaxed convergence criterion is applied after the initial iteration., that is iteration zero in Figure 2.42 in section 2.4.5 Linear Solver Controls and Options. Increasing this parameter may improve convergence by applying stricter convergence controls over additional Newton iterations, but may also increase the number of Newton iterations over a time step before the time step is accepted.
7	--tolerance-cnv		A real positive double precision value that specifies the maximum non-linear tolerance error. This is the local convergence tolerance (maximum of local saturation errors). The default value is 0.01
8	--tolerance-mb		A real positive double precision value that sets the maximum mass balance error, that is the tolerated mass balance error relative to total mass present. the default value is 1.0 x 10-6.

Notes:

1) Cells colored gray under the No. column, are not recommend to be modified.
2) Cells colored orange under the TUNING Equivalent Keyword column with no TUNING keyword parameter do not have an equivalent OPM Flow command line parameter.
3) Cells colored orange under the TUNING Equivalent Keyword column with a TUNING keyword parameter, means OPM Flow does not support reading this parameter from the TUNING keyword and therefore the OPM Flow command line option should be used.

Table 2.5: Newton-Raphson Solver Controls

Detailed descriptions of all of the command line parameters are given in Table 2.1 in section *2.2 Running OPM Flow 2023-10 From The Command Line*.

2.4.5 LINEAR SOLVER CONTROLS AND OPTIONS

There are numerous options for the linear solver, which can be overwhelming even for the most experienced users, given the rich feature set enabled by the various command line parameters. Fortunately, the simulator summarizes the options in use in the *Linear Solver Property Tree* printed in both the *.PRT and *.DBG files, right after processing the input data and just before the simulation starts.

```
Property tree for linear solver:
{
    "tol": "0.01",
    "maxiter": "200",
    "verbosity": "1",
    "solver": "bicgstab",
    "preconditioner": {
        "type": "ParOverILU0",
        "relaxation": "0.90000000000000002",
        "ilulevel": "0"
    }
}
```

Figure 2.41: Linear Solver Property Tree

Figure 2.41 above shows an example of the Linear Property Tree, based on using mostly the linear solver default parameters. Here the tol value of 0.01 is equivalent to the --tolerance-cnv command line parameter, and maxiter is equivalent to --linear-solver-max-iter, as summarized in Table 2.6 for all the other parameters.

No.	Property Tree Parameter	Equivalent Command Line Parameter
1	tol	--tolerance-cnv
2	maxiter	--linear-solver-max-iter
3	verbosity	--linear-solver-verbosity
4	solver	--use-gmres=false
5	preconditioner	
6	type	--linear-solver
7	relaxation	--ilu-relaxation
8	ilulevel	--ilu-fillin-level

Notes:

1) BiCGSTAB stands for Biconjugate Gradient Stabilized linear solver.
2) GMRES is for the Generalized Minimal Residual linear solver.

Table 2.6: Linear Solver Property Tree versus Command Line Parameters

Table 2.7 list the major linear solver command line parameters discussed in this section.

Figure 2.42 shows typical output for the linear solver for the March 1st report step advancing to the next report step. Naturally, the Newton and linear iteration counts are same as for Figure 2.39 for the common periods; however, oil, gas and water material balance values are printed for each non-linear iteration, together with the respective convergence values.

```
Report step  3/61 at day 59/1825, date = 01-Mar-2022

Starting time step 0, stepsize 5 days, at day 59/90, date = 01-Mar-2022
Iter     MB(O)      MB(W)      MB(G)      CNV(O)     CNV(W)     CNV(G)
   0  2.796e-04  2.542e-04  4.337e-05  2.483e-02  1.566e-02  3.852e-03
   1  3.213e-06  2.965e-06  4.986e-07  2.074e-02  2.037e-03  3.217e-03
   2  1.953e-08  5.454e-08  3.030e-09  1.417e-04  6.553e-05  2.199e-05
Overall linear iterations used: 5
 Newton its= 2, linearizations= 3 (0.0sec), linear its=  5 (0.0sec)

Starting time step 1, stepsize 5 days, at day 64/90, date = 06-Mar-2022
Iter     MB(O)      MB(W)      MB(G)      CNV(O)     CNV(W)     CNV(G)
   0  2.788e-04  2.535e-04  4.325e-05  3.719e-02  1.539e-02  5.771e-03
   1  3.141e-06  2.818e-06  4.874e-07  3.073e-02  2.001e-03  4.768e-03
   2  5.835e-09  7.564e-08  9.053e-10  1.708e-04  1.291e-04  2.650e-05
Overall linear iterations used: 4
 Newton its= 2, linearizations= 3 (0.0sec), linear its=  4 (0.0sec)

Starting time step 2, stepsize 5 days, at day 69/90, date = 11-Mar-2022
Iter     MB(O)      MB(W)      MB(G)      CNV(O)     CNV(W)     CNV(G)
   0  2.780e-04  2.528e-04  4.314e-05  3.708e-02  1.545e-02  5.752e-03
   1  2.945e-06  2.718e-06  4.569e-07  3.066e-02  1.519e-03  4.757e-03
   2  8.059e-09  7.314e-08  1.250e-09  1.301e-04  1.049e-04  2.018e-05
Overall linear iterations used: 4
 Newton its= 2, linearizations= 3 (0.0sec), linear its=  4 (0.0sec)

Starting time step 3, stepsize 5 days, at day 74/90, date = 16-Mar-2022
Iter     MB(O)      MB(W)      MB(G)      CNV(O)     CNV(W)     CNV(G)
   0  2.773e-04  2.522e-04  4.303e-05  3.604e-02  1.518e-02  5.592e-03
   1  2.922e-06  2.665e-06  4.533e-07  2.982e-02  1.458e-03  4.626e-03
   2  5.543e-09  7.134e-08  8.601e-10  1.146e-04  9.402e-05  1.778e-05
Overall linear iterations used: 4
 Newton its= 2, linearizations= 3 (0.0sec), linear its=  4 (0.0sec)

Starting time step 4, stepsize 5 days, at day 79/90, date = 21-Mar-2022
Iter     MB(O)      MB(W)      MB(G)      CNV(O)     CNV(W)     CNV(G)
   0  2.766e-04  2.515e-04  4.292e-05  3.530e-02  1.494e-02  5.477e-03
   1  2.804e-06  2.584e-06  4.350e-07  2.920e-02  1.435e-03  4.531e-03
   2  9.103e-09  7.195e-08  1.413e-09  1.204e-04  1.098e-04  1.868e-05
Overall linear iterations used: 4
 Newton its= 2, linearizations= 3 (0.0sec), linear its=  4 (0.0sec)
```

Time Step | Newton Iteration | Total Linear Iterations | Material Balance | Convergence Tolerance

Figure 2.42: Time Step Summary Reporting Linear Solver Example

The total number of linear iterations displayed in Figure 2.42 is the total number of iterations for all the non-linear iterations, not for a particular non-linear iteration. To get the number of linear iterations per non-linear iteration use the following command line parameter:

```
--linear-solver-verbosity=1
```

Which will produce a report similar to that shown in Figure 2.43, with the IT parameter indicating the number of linear iterations for the given Newton iteration.

```
Starting time step 0, stepsize 0.0118592 days, at day 0/10, date = 01-Jan-1967
=== Dune::BiCGSTABSolver
=== rate=3.08338e-09, T=0.000472239, TIT=0.000472239, IT=1
=== Dune::BiCGSTABSolver
=== rate=0.35949, T=0.00317358, TIT=0.000634715, IT=5
=== Dune::BiCGSTABSolver
=== rate=0.51016, T=0.00358736, TIT=0.00051248, IT=7
=== Dune::BiCGSTABSolver
=== rate=0.0874993, T=0.0032023, TIT=0.00160115, IT=2
=== Dune::BiCGSTABSolver
=== rate=0.0976473, T=0.00200675, TIT=0.00100338, IT=2
=== Dune::BiCGSTABSolver
=== rate=0.260942, T=0.000812008, TIT=0.000270669, IT=3
=== Dune::BiCGSTABSolver
=== rate=0.00482599, T=0.00161764, TIT=0.00161764, IT=1
=== Dune::BiCGSTABSolver
=== rate=0.0938774, T=0.00202222, TIT=0.00101111, IT=2
=== Dune::BiCGSTABSolver
=== rate=0.20818, T=0.000666642, TIT=0.000222214, IT=3
=== Dune::BiCGSTABSolver
=== rate=0.155462, T=0.000522632, TIT=0.000261316, IT=2
 Newton its=10, linearizations=11 (0.0sec), linear its= 28 (0.0sec)
```

Figure 2.43: Number of Linear Iterations Per Newton Iteration Report Example

As mentioned previously, the convergence of the non-linear solver is dependent on the accuracy of the linear solvers. Solving the linear system to a loose accuracy may lead to poor convergence of the non-linear solver. The tuning of the time stepping procedure, and the tolerance used in the linear and non-linear solvers are highly related, and optimal choices are case and problem dependent.

From the user prospective, there are several command-line options that can be used to configure and control the linear solver, in particular:

1) Using an alternative linear solver by changing

   `--use-gmres`

   The default value of false sets the linear solver to the Biconjugate Gradient Stabilized ("BiCGSTAB") linear solver, whereas a value of true will set the linear solver to use the Generalized Minimal Residual linear solver ("GMRES"). If the default linear solver is not converging then changing to the GMRES solver may help. Note, that the GMRES linear solver has a slightly higher memory consumption, as it stores some of the previous search iterations as part of the optimization process. The default number of search iterations stored is 15, which means 14 more vectors than BiCGSTAB solver.

2) Changing the linear solver preconditioner can dramatically improve the performance of the linear solver, which is accomplished by the following command line parameter:

   `--linear-solver`

   In many cases using a Constraint Pressure Residual ("CPR") preconditioner instead of the default Incomplete Lower-triangular Upper-triangular factorization with level of fill in 0 ("ILU0"), will result in difficult problems being solved at the expense of additional work. For example, model two in the Tenth SPE Comparative Solution Project[57], fails to run using the default preconditioner, but runs satisfactory with the CPR preconditioner. Note that the commercial simulator also has a CPR option

[57] *Christie, M.A., and M.J. Blunt. "Tenth SPE Comparative Solution Project:A Comparison of Upscaling Techniques." SPE Res Eval & Eng 4 (2001): 308–317. doi: https://doi.org/10.2118/72469-PA.*

that is implemented via the CPR keyword in the RUNSPEC, but the implementation is different to that of OPM Flow.

3) And finally increasing the number linear iterations may be appropriate, via:

```
--linear-solver-max-iter
```

that defines the maximum number of linear iterations. Sometimes increasing this value may improve performance.

Major Linear Solver Controls			
No.	**Command Line Parameter**	**TUNING Keyword Equivalent**	**Description**
1	--linear-solver		"ilu0"A defined quoted character string that sets the configuration of the solver, valid values are: 1) ilu0 (default), 2) cpr (an alias for cpr_trueimpes)[58] [59] and [60] 3) cpr_quasiimpes, 4) cpr_trueimpes, 5) cprw, 6) amg,[61] or 7) a file name that has the extension ".json", that contains the linear solver configuration parameters. Option (5) extends the existing Constrained Pressure Residual ("CPR") preconditioner to include wells. This option can also be invoked via the CPR keyword in the RUNSPEC section; however, the command line parameter takes precedence. For option (6) one enters a character string enclosed in quotes that defines the filename of a JSON configuration file for the flexible linear solver system. In this case the file extension should be '.json'. Note that the *.PRT file contains the "Property tree for the linear solver" listing, which is the JSON specification of the current case, and can be used as a guide to configure a user specific linear solver JSON file. The default is "ilu0". **The option "cpr" now is an alias for "cprw" instead of "cpr_trueimpes".**

[58] *Wallis, J. R., Little, T. E., and Nolen, J. S.: "Constrained Residual Acceleration of Conjugate Residual Methods," paper SPE 13536 presented at the SPE Reservoir Simulation Symposium, Dallas, Texas, USA (February 10-13, 1985).*

[59] *R. Scheichl, M. Roland, J. Wendebourg, Decoupling and block preconditioning for sedimentary basin simulations, Computational Geosciences 7 (2003) 295{318.*

[60] *Klemetsdal, Ø.S., Møyner, O. & Lie, KA. Accelerating multiscale simulation of complex geomodels by use of dynamically adapted basis functions. Comput Geosci 24, 459–476 (2020). https://doi.org/10.1007/s10596-019-9827-z.*

[61] *M. Blatt, A parallel algebraic multigrid method for elliptic problems with highly discontinuous coefficients, Ph.D. thesis, Ruprecht-Karls-Universität Heidelberg (2010).*

Major Linear Solver Controls			
No.	**Command Line Parameter**	**TUNING Keyword Equivalent**	**Description**
2	--linear-solver-ignore-convergence-failure		A Boolean value set to true or false that if set to true convergences failures in the linear solver are ignored. This option should be used with care, as the results may be unreliable.
3	--linear-solver-max-iter	LITMAX	A positive integer value that defines the maximum number of linear iterations.
4	--linear-solver-reduction		A real positive double precision value that sets the minimum reduction of the residual for the linear solver. The linear solver convergences when the residual is reduced sufficiently. In general, it is not recommended to modify this parameter.
5	--linear-solver-require-full-sparsity-pattern		A Boolean value set to true or false that if set to true the simulator will produce the full sparsity pattern for the linear solver. n general, it is not recommended to modify this parameter.
6	--linear-solver-restart		A positive integer value that sets the number of iterations after which GMRES is restarted. n general, it is not recommended to modify this parameter.
7	--use-gmres		A Boolean value that when set to true OPM Flow will use Generalized Minimal Residual ("GMRES") solver instead of Biconjugate Gradient Stabilized ("BiCGSTAB") as the linear solver within the Newton iterations.

Notes:

1) Cells colored gray under the No. column, are not recommend to be modified.
2) Cells colored orange under the TUNING Equivalent Keyword column with no TUNING keyword parameter do not have an equivalent OPM Flow command line parameter.
3) Cells colored orange under the TUNING Equivalent Keyword column with a TUNING keyword parameter, means OPM Flow does not support reading this parameter from the TUNING keyword and therefore the OPM Flow command line option should be used.

Table 2.7: Linear Solver Controls

Again Detailed descriptions of all of the command line parameters are given in Table 2.1 in section 2.2 Running OPM Flow 2023-10 From The Command Line.

CHAPTER 3: KEYWORD DOCUMENTATION STRUCTURE

3.1 INTRODUCTION

The OPM Flow manual is constructed in a manner to enable the reader to reference various parts of the document by using the table contents or by simply pressing on a link embedded in the text. This automatic cross referencing has been extensively employed to ensure effective documentation of the keywords used by the simulator.

There are several key features that can be used to navigate the manual in an efficient manner. The first is the "footer" which can be used to move to various sections in the manual. For example the "Table of Contents" footer shown below:

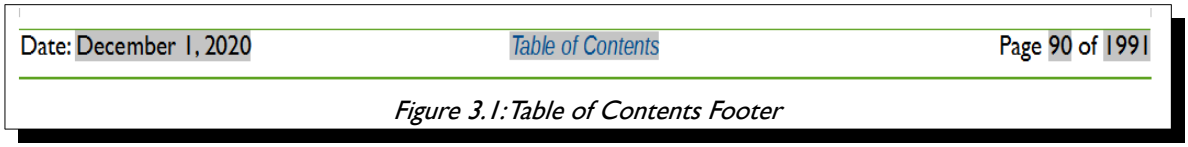


Figure 3.1: Table of Contents Footer

Allows the reader to move to the Table of Contents by clicking on the link (highlighted in italic blue text on a gray background in Figure 3.1). Note also that the entries in the table of contents are also "clickable" enabling the reader to move to the desired entry.

The second type of footer is the "Keyword Index" footer that also contains the Table of Contents link mentioned above. This footer is illustrated in Figure 3.2 and is displayed on a keyword definition page.

Figure 3.2: Keyword Index Footer

Clicking on a letter (highlighted in italic blue text on a gray background in) takes the reader to an alphabetic listing of all the keywords beginning with the selected letter (Figure 3.3).

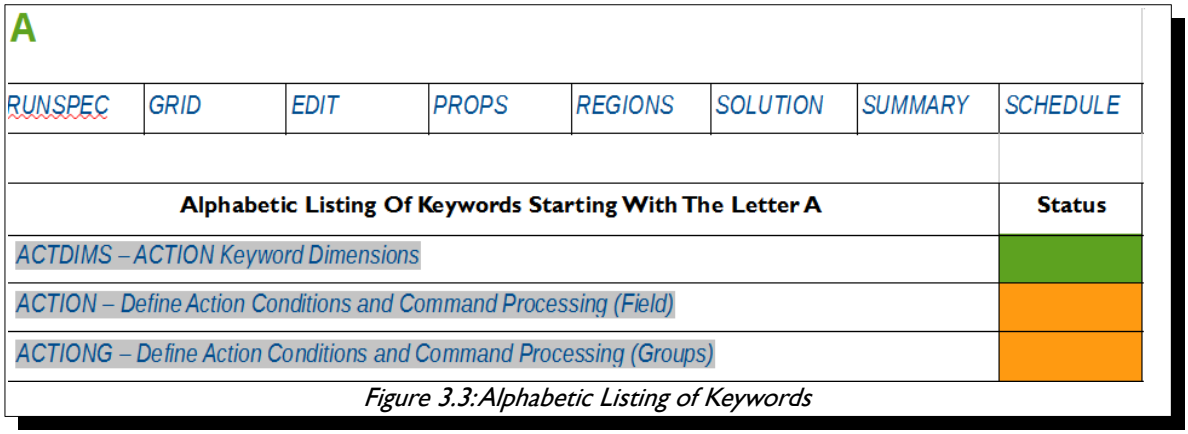


Figure 3.3: Alphabetic Listing of Keywords

The list is color coded, so one instantly knows what keywords are implemented, with green colored cells indicating the keyword is available and is mostly or fully implemented. Cells colored orange show that keyword is recognized but not implemented in OPM Flow. Finally, cells colored red mean that keyword is available in the commercial simulator but has not been implemented in OPM Flow, and may cause an error if used in the input deck. Clicking on a keyword in the list it will move the reader to the keyword definition. Note also that clicking one of the section names, RUNSPEC, GRID, etc., in Figure 3.3, will take the reader to the beginning of the selected section.

Finally, in the PDF version of the manual if one displays the "bookmarks" in the PDF reading software one can jump to a particular keyword or section without having to scroll up or down.

3.2 KEYWORD DEFINITIONS

Each keyword is defined in it's own section that contains a section header, that contains the keyword name in capital letters followed by a brief description of the keyword's function. This is then followed by Keyword Table Section which defines the status of the keyword and which sections of the input deck the keyword can be utilized. Table 3.1 illustrates a typical Keyword Table Section defining the keyword status with the various OPM Flow sections.

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Table 3.1: Example Keyword Table Section

The cells are colored in such a manner as to quickly indicate to the reader the keyword's section availability and function availability, with green colored cells indicating the keyword is available for this section and is mostly or fully implemented. Cells colored gray indicate that keyword cannot be used in that particular section, and cells colored orange show that cell is only partially implemented within OPM Flow, for example OPM Flow may simple just recognize the keyword and ignore the keyword's function. Finally, cells colored red mean that keyword is available in the commercial simulator but has not been implemented in OPM Flow. In this scenario the keyword should not be used in OPM Flow as it will result in unpredictable results, including causing the simulator to abort or throw an exception.

3.3 MULTI-SECTION KEYWORDS

As there are numerous keywords that can be used within multiple OPM Flow sections of the input file, for example the *ADD* and *EQUALS* keywords, there is a need to avoid duplication of the keyword definitions but at the same time attempt to define only those keywords for a given section. Thus for multi-section keywords, the keyword is defined in the first available section that the keyword can be found. The Keyword Table Section as shown below for the *ADD* keyword below, indicates which sections the keyword can be utilized.

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Table 3.2:ADD Keyword Table Section

Here the keyword can be used in the GRID, EDIT, PROPS, REGIONS and SOLUTION sections as indicated by those cells colored green and not for the cells colored in light gray. In subsequent sections that the keyword can be used, there is a brief description description with a link to the full description of the keyword, as shown below for the ADD keyword.

> The ADD keyword adds a constant to a specified array or part of an array. The constant can be real or integer depending on the array type; however, the arrays that can be operated on is dependent on which section the ADD keyword is being applied. See *ADD – Add a Constant to a Specified Array* in the GRID section for a full description.

A complete list of keywords in alphabetic order is given in section *APPENDIX A: KEYWORD INDEX - ALPHABETIC LISTING* , and clicking on a specific keyword will take the reader to the keyword definition in a particular section.

3.4 KEYWORD FORMATS

All keywords in OPM Flow should be entered in capital case and start in column one, lowercase entry of keywords will be produce errors and keywords not starting in column one will not be recognized. There are five types of keyword format types used by OPM Flow for data input. The description of the five types is given in the next three section together with some examples.

3.4.1 KEYWORD FORMAT TYPE – COMMENT

Comments in the input deck can occur anywhere in the file are preceded by "--" in columns one and two, for example for the EQUL keyword:

```
--
--       MAX     MAX     RSVD    TVDP    TVDP
--       EQLNUM  DEPTH   NODES   TABLE   NODES
EQLDIMS
         9       1*      20      1*      1*                                  /
```

In addition, comments can be place after "/" that terminates a record entry as shown below;

```
--
-- -- ARRAY    CONSTANT --  ---------- BOX ---------
--                         I1  I2   J1  J2   K1  K2
MULTIPLY
   'PERMZ'     0.50000     1*  1*   1*  1*   1*  1* / PERMZ * 0.5
/
```

3.4.2 KEYWORD FORMAT TYPE – ACTIVATION

This type of keyword format only consists of the keyword itself and is usually used to invoke a feature or to switch on or off a processing feature. The keyword is document by describing the functionality or action the keyword performs, followed by an example. Examples of this type of keyword include API (to switch on API tracking), GAS (to activate the gas phase in the model), ECHO (to switching echoing of the input file to the output file), and SKIP (for skipping parts of the input deck). For example the GAS keyword in the RUNSPEC section would be described as:

Description

This keyword indicate that the gas phase is present in the model and must be used for oil-gas, gas-water, oil-water-gas input decks that contain the gas phase. The keyword will also invoke data input file checking to ensure that all the required gas phase input parameters are defined in the input deck.

There is no data required for this keyword.

Example

```
--
--       GAS PHASE IS PRESENT IN THE RUN
--
GAS
```

The above example declares that the gas phase is active in the model.

3.4.3 KEYWORD FORMAT TYPE - VECTOR (ROW VECTOR)

Vector based keywords consist of the keyword followed by a vector of parameters on a separate line and may consists of multiple lines of vectors with each line representing a data set (see the second example for this type of vector keyword). The vector may contain integer, real and character parameters depending on the keywords requirements. This type of keyword is documented by describing the functionality or action the keyword performs, a table describing the parameters associated with the keyword, followed by one or two examples on how to use the keyword. For example the DIMENS keyword in the RUNSPEC section would be described as:

Description

DIMENS defines the dimensions of the model entered as integer vector. The keyword can be used with all grid types.

No.	Name	Description	Default
1	NX	The number of grid blocks in the x direction for Cartesian grids or the number of grid blocks in the r direction for radial grids	None
2	NY	The number of grid blocks in the y direction for Cartesian grids or the number of grid blocks in the theta direction for radial grids.	None
3	NZ	The number of grid blocks in the z direction for both Cartesian and radial grids.	None
Notes: 1) The keyword is terminated by a "/".			

Table 3.3: DIMENS Keyword Description

Note that NX, NY and NZ are not maximum values but the actual size of the grid. OPM Flow applies these parameters when reading in particular data sets. For example if NX, NY, and NZ are set to 10, 10 and 10 respectively, then for the grid property data like PORO; OPM Flow expects to read in 10 x 10 x 10 or 1,000 porosity values for the PORO array. If the number of porosity values is not equal to 1,000 then OPM Flow will produce an error.

Example

```
--
--        MAX      MAX      MAX
--        NDIVIX   NDIVIY   NDIVIZ
DIMENS
          46       112      22                                                  /
```

The above example defines the dimensions for the Norne model of 36 cells in the x direction, 122 cells in the y direction and 22 cells in the z direction.

For vector keywords that have parameters associated with units, then there is a slightly different table format to that used above to take into account documenting the default values for the three sets of units supported by OPM Flow, for example for the ROCK keyword is describe as follows:

Description

ROCK defines the rock compressibility for various regions in the model. The number of ROCK vector data sets is defined by the NTPVT parameter on the TABDIMS keyword in the RUNSPEC section and the allocation of the ROCK tables to different grid blocks in the model is done via the PVTNUM keyword in the REGION section. One data set consists of one record or line which is terminated by a "/".
This keyword must be defined in the OPM Flow input deck.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	Pref	Pref is a real number defining the reference pressure for the other parameters for this data set.			
		psia 1.032	barsa 1.032	atma 1.032	Default
2	Cf	Cf is a real number defining the rock compressibility at the reference pressure, Cf(Pref) and is defined as: $$C_f = -\frac{1}{V}\left(\frac{dV}{dP}\right)$$			
		1/psia 0.0	1/barsa 0.0	1/atma 0.0	Defined
Notes: 1) The each data set terminated by terminated by a "/" at the end of the line, there is no terminator for the keyword.					

Table 3.4: ROCK Keyword Description

Example

The following shows the PVTW keyword for when NTPVT on the TABDIMS keyword in the RUNSPEC section is set to one.

```
--
-- ROCK COMPRESSIBILITY
--
-- (1) REFERENCE PRESSURE IS TAKEN FROM THE HCPV WEIGHTED RESERVOIR PRESSURE
--     AS THE PORV IS ALREADY AT RESERVOIR CONDITIONS (FLOW USES THE REFERENCE
--     PRESSURE) TO CONVERT THE GIVEN PORV TO RESERVOIR CONDITIONS USING THE DATA
--     ON THE ROCK KEYWORD)
--
ROCK
    3566.9    5.0E-06                          / ROCK COMPRESSIBILITY REGION 1
    3966.9    5.5E-06                          / ROCK COMPRESSIBILITY REGION 2
    4566.9    6.0E-06                          / ROCK COMPRESSIBILITY REGION 3
```

There is no terminating "/" for this keyword.

In this case the example shows a multiple data set entry of the vector format keyword, with three ROCK data sets being defined by the keyword.

3.4.4 KEYWORD FORMAT TYPE – VECTOR (COLUMNAR VECTOR)

Columnar vector based keywords consist of the keyword followed by a columnar vector of parameters in a separate column for each parameter. The vector may contain integer, real and character parameters depending on the keywords requirements. This type of keyword is documented by describing the functionality or action the keyword performs, a table describing the parameters associated with the keyword, followed by one or two examples on how to use the keyword. For example the SWFN keyword in the PROPS section would be described as:

Description

The SWFN keyword defines the water relative permeability and water-oil capillary pressure data versus water saturation tables for when water is present in the input deck. This keyword should only be used if water is present in the run.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SWAT	A columnar vector of real monotonically increasing down the column values starting from zero and terminating at one, that defines the water saturation.			
		dimensionless	dimensionless	dimensionless	None
2	KRW	A columnar vector of real values that are either equal or increasing down the column and that are greater than or equal to zero and less than or equal to one that defines the water relative permeability with respect to gas saturation. The first value in the column should be zero.			
		dimensionless	dimensionless	dimensionless	None
3	PCWO	A columnar vector of real values that are either equal or increasing down the column that defines the water-oil relative capillary pressure. If the SWATINIT keyword has been used to initialize the model then columnar vector has to be strictly monotonically increasing.			
		psia	bars	atm	None

Notes:

1) The keyword is followed by NTSFUN tables as declared on the TABDIMS keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NSSFUN rows as declared on the TABDIMS keyword in the RUNSPEC section.
3) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 3.3: SWFN Keyword Description

Example

```
--
--       WATER RELATIVE PERMEABILITY TABLES (SWFN)
--
SWFN
--       SWAT       KRW        PCOW
--       FRAC       FRAC       PSIA
--       --------   --------   -------
            0.15     0.00000     1*
            0.30     0.00050     1*
            0.40     0.00390     1*
            0.50     0.01500     1*
            0.60     0.04100     1*
            0.65     0.06250     1*
            0.70     0.09150     1*
            0.80     0.17850     1*
```

```
            0.90     0.31640     1*
            0.95     0.40960     1*
            1.00     0.52200     1*                          / TABLE NO. 1
```

The example defines two SWFN tables for use when water is present in the run. In the tables the water-oil capillary pressure data has been defaulted with "1*" and will be set to zero as there are no other values for the water-oil capillary pressure columns.

3.4.5 KEYWORD FORMAT TYPE – ARRAY

This type of keyword defines a property for the grid or an area of the grid using a previously entered BOX keyword to define the area where the property will be defined. For array data a full set of values for each element in the array is required. For example, the documentation for the PORO array would be:

Description

PORO defines the porosity for all the cells in the model via an array. The keyword can be used with all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	PORO	PORO is an array of real numbers assigning the porosity values to each cell in the model. The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword. Repeat counts may be used, for example 30*100.0.			None
		dimensionless	dimensionless	dimensionless	
Notes: 1) The keyword is terminated by a "/".					

Table 3.3: PORO Keyword Description

See also the DX, DY and TOPS keywords to fully define a Cartesian Regular Grid.

Example

```
--
--  DEFINE GRID BLOCK POROSITY DATA FOR ALL CELLS (BASED ON NX x NY x NZ = 300)
--
PORO
  300*0.300
/
```

3.5 INPUT FILE STRUCTURE

OPM Flow input files are similar to commercial simulators that are used in the oil and gas industry, that is the input file is separate into sections in an effort to avoid an engineer's input data errors and a computer programmer's code to interpret the data. OPM Flow has been designed, from an engineer's prospective and input structure to be similar to Schlumberger's industry wide ECLIPSE 100[62] simulator. Table 3.4 lists the various section together with a brief description of purpose of the section. As well as declaring if section is mandatory or not for a run to form a valid input deck.

Section Name	Description	Required Optional
RUNSPEC	This is the first section in the OPM Flow input file and defines the key parameters for the simulator including the dimensions of the model, phases present in the model (oil, gas and water for example), number of tables for a given property and the maximum number of rows for each table, the maximum number of groups, wells and well completions, as well as various options to be invoked by OPM Flow.	Required
GRID	Defines the basic grid properties, including structure, faults and various static rock properties (porosity, permeability etc.). Upon completion of reading this section, the software calculates the pore volume (PORV) for each and the transmissibilities (TRANX, TRANY and TRANX,) between all the cells, as well as calculating the transmissibilities of the Non-Neighbor Connections ("NNC")	Required
EDIT	The properties calculated by OPM Flow in the GRID section are available for editing in this section (PORV, TRANX etc.).	Optional
PROPS	This section defines the fluid properties for all the phases present in the run, for example oil viscosity, oil formation volume factor etc. The section also defines the rock flow properties as per the relative permeabilities and the distribution of the fluids based on the capillary pressure functions.	Required
REGIONS	The REGIONS section allows the engineer to define various regions in the model for reporting purposes and to define how the fluid and rock property defined in the PROPS section are allocated throughout the model.	Required
SOLUTION	Defines the parameters to initialize the model, fluid contacts, reservoir pressures etc., together with the data from the previous sections. This section, if requested, reports the initial in-place volumes for phases present in the model, as well as the average pressure for the various defined regions.	Required
SUMMARY[1]	Defines the time vector property data to be written out to various report output files for reviewing the results of the simulation. This data includes field, group, well and well completion production and injection data, for example field oil rate versus time. Grid block data can also be reported versus time as well, for example grid block pressure versus time[2].	Optional
SCHEDULE	The final section is the SCHEDULE that the defines the field, group and well parameters, targets and constraints that should be applied to the field, group or wells, numerical controls, the operating schedule and reporting requirements.	Required

[62] *ECLIPSE Industry-Reference Reservoir Simulator – Reference Manual 2019.1, Schlumberger.*

Section Name	Description	Required Optional

Notes:

1) Although the SUMMARY section is optional, it is nearly always included in order to obtain results from the simulation run.
2) The OPM ResInsight three-dimensional visualization software has a feature to display a grid block property change versus time, so it should not be necessary to export the grid block data to the SUMMARY report output files.

Table 3.4: OPM Flow Input Deck Sections

CHAPTER 4: GLOBAL SECTION KEYWORDS

4.1 INTRODUCTION

Keywords used in this section can be used in all input file sections, that is the RUNSPEC, GRID, EDIT PROPS, REGIONS, SOLUTION, SUMMARY and SCHEDULE sections.

4.2 DATA REQUIREMENTS

There are no data requirements for the GLOBAL section keywords, as all the keywords are optional and considered to utility keywords that can be used throughout the input deck. A complete list of these global keywords is given in Table 4.1.

No.	GLOBAL Keyword	Description	Section Applied
1	COLUMNS	*COLUMNS – Define Input File Column Margins*. The COLUMNS keyword defines the input file column margins; characters outside the margins are ignored by the input parser.	ALL
2	DEBUG	*DEBUG – Define the Debug Data to be Printed to File*. This keyword defines the debug data to be written to the debug file (*.DBG), it is ignored by OPM Flow.	ALL
3	ECHO	*ECHO – Activate Echoing of User Input Files to the Print File*. Turns on echoing of all the input files to the print file; note that this keyword is activated by default and can subsequently be switched off by the NOECHO activation keyword.	ALL
4	END	*END – Define the End of the Input File*. This keyword marks the end of the input file and can occur in any section. Any keywords and data after the END keyword are ignored.	ALL
5	ENDINC	*ENDINC – Define the End of an Include File*. This keyword marks the end of an include file specified on the INCLUDE keyword. When the ENDINC keyword is encountered in the INCLUDE file, input data is read from the next keyword in the current file.	ALL
6	ENDSKIP	*ENDSKIP – DeActivate Skipping of Keywords and Input Data*. Turns off skipping of keywords that was activated by the SKIP, SKIP100, or SKIP300 keywords.	ALL
7	EXTRAPMS	*EXTRAPMS – Activate Extrapolation Warning Messages*. The EXTRAPMS keyword activates extrapolation warning messages for when simulator extrapolates the PVT or VFP tables.	ALL
8	FORMFEED	*FORMFEED – Defined the Print File Form-Feed Character*. The keyword defines the form-feed character, or carriage control character, for the output print (*.PRT) run summary (*.RSM) files.	ALL
9	INCLUDE	*INCLUDE – Load Another Data File at the Current Position*. The INCLUDE keyword informs OPM Flow to continue reading input data from the specified INCLUDE file.	ALL
10	MESSAGE	*MESSAGE – Output User Message*. The MESSAGE keyword outputs a user message to the terminal, as well as to the print (*.PRT) and debug (*.DBG) files.	ALL

No.	GLOBAL Keyword	Description	Section Applied
11	MESSAGES	*MESSAGES – Define Message Print Limits and Stop Limits*. The MESSAGES keyword defines the print and stops levels for various messages.	ALL
12	NOECHO	*NOECHO – Deactivate Echoing of User Input Files to the Print File*. Turns off echoing of all the input files to the print file.	ALL
13	NOWARN	*NOWARN – Deactivate Warning Messages*. Turns off warning messages to be printed to the print file.	ALL
14	SKIP	*SKIP – Activate Skipping of All Keywords and Input Data*. The keyword turns on skipping of keywords until the ENDSKIP activation keyword is encountered.	ALL
15	SKIP100	*SKIP100 – Activate Skipping of Black-Oil Keywords and Input Data*. This keyword turns on skipping of black-oil keywords until the ENDSKIP activation keyword is encountered.	ALL
16	SKIP300	*SKIP300 – Activate Skipping of "Compositional" Keywords and Input Data*. Turns on skipping of "compositional" keywords until the ENDSKIP activation keyword is encountered.	ALL
17	WARN	*WARN – Activate Warning Messages*. Turns on warning messages to be printed to the print file (*.PRT); note that this keyword is activated by default.	ALL

Notes:

1) Note that not all keywords and features listed above are implemented in OPM Flow. Cells not colored in the No. column indicate the keyword is supported or partially supported by OPM Flow, cells colored gray indicate that the keyword is not applicable, and finally, cells colored in orange indicate keywords that are not supported by OPM Flow and will be ignored by the simulator.

Table 4.1: Alphabetic List of GLOBAL Section Keywords

A more detailed description of the keywords is presented in the following sections.

4.3 KEYWORD DEFINITIONS

4.3.1 COLUMNS – DEFINE INPUT FILE COLUMN MARGINS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The COLUMNS keyword defines the input file column margins; characters outside the margins are ignored by the input parser.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

4.3.2 DEBUG – DEFINE THE DEBUG DATA TO BE PRINTED TO FILE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword defines the debug data to be written to the debug file (*.DBG). This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

4.3.3 ECHO – ACTIVATE ECHOING OF USER INPUT FILES TO THE PRINT FILE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

Turns on echoing of all the input files to the print file; note that this keyword is activated by default and can subsequently be switched off by the NOECHO activation keyword.

There is no data required for this keyword.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

Example

```
--
--         SWITCH OFF ECHOING OF INPUT FILES
--
NOECHO
--
--         INCLUDE SIMULATION GRID WITH SLOPING FAULTS
--
INCLUDE
           './INCLUDE/GRID/IRAP_1005.GRDECL' /
--
--         SWITCH ON ECHOING OF INPUT FILES
--
ECHO
```

The examples deactivates the echoing of the input files, reads in the grid geometry data using the INCLUDE keyword, and then activates the echoing of the input files again.

Note

Especially for the large voluminous data sets in the GRID section, it is good practice to deactivate the echoing of the input files when loading this data to avoid the print output file becoming too large to view in a text editor.

4.3.4 END – DEFINE THE END OF THE INPUT FILE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword marks the end of the input file and can occur in any section. Any keywords and data after the END keyword are ignored.

There is no data required for this keyword.

Example

```
-- ------------------------------------------------------------------------
-- SCHEDULE SECTION - 2006-01-01
-- ------------------------------------------------------------------------
RPTSCHED
'WELLS=2'    'WELSPECS'    'CPU=2'     'FIP=2'                              /

DATES
 1  JAN   2006  /
/

RPTSCHED
'NOTHING'                                                                 /

DATES
 1  APR   2006  /
 1  JLY   2006  /
 1  OCT   2006  /
/
ECHO
--
-- ************************************************************************
-- END OF FILE
-- ************************************************************************
END
-- ------------------------------------------------------------------------
-- SCHEDULE SECTION - 2007-01-01
-- ------------------------------------------------------------------------
RPTSCHED
'WELLS=2'    'WELSPECS'    'CPU=2'     'FIP=2'                              /

DATES
 1  JAN   2007  /
/
```

In the above example OPM Flow will process the data up to October 1, 2006 only, and then start to run the simulation. All keywords after the END file keyword will not be read or processed.

4.3.5 ENDINC – DEFINE THE END OF AN INCLUDE FILE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword marks the end of an include file specified on the INCLUDE keyword. When the ENDINC keyword is encountered in the INCLUDE file, input data is read from the next keyword in the current file. Any keywords and data after the ENDINC keyword in the INCLUDE file are ignored.

There is no data required for this keyword.

Example

```
-- ------------------------------------------------------------------------
-- SCHEDULE SECTION - 2006-01-01
-- ------------------------------------------------------------------------
RPTSCHED
'WELLS=2'    'WELSPECS'    'CPU=2'     'FIP=2'                              /

DATES
 1  JAN   2006  /
/

RPTSCHED
'NOTHING'                                                                 /

DATES
 1  APR   2006  /
 1  JLY   2006  /
 1  OCT   2006  /
/
ECHO
--
-- ************************************************************************
-- END OF INCLUDE FILE PROCESSING
-- ************************************************************************
ENDINC
-- ------------------------------------------------------------------------
-- SCHEDULE SECTION - 2007-01-01
-- ------------------------------------------------------------------------
RPTSCHED
'WELLS=2'    'WELSPECS'    'CPU=2'     'FIP=2'                              /

DATES
 1  JAN   2007  /
/
```

In the above example OPM Flow will process the data up to October 1, 2006 only, and return control to the file that called the INCLUDE keyword, and then continue processing the input files. All keywords after the ENDINC keyword in the INCLUDE FILE will not be read or processed.

4.3.6 ENDSKIP – DEACTIVATE SKIPPING OF KEYWORDS AND INPUT DATA

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

Turns off skipping of keywords that was activated by the SKIP, SKIP100, or SKIP300 keywords. Note that for each SKIP keyword activated there must be a corresponding ENDSKIP keyword.

There is no data required for this keyword.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

Example

```
--
--        SWITCH ON SKIPPING OF KEYWORDS AND DATA
--
SKIP
--
--        INCLUDE SIMULATION GRID WITH SLOPING FAULTS
--
INCLUDE
          './INCLUDE/GRID/IRAP_1005.GRDECL' /
--
--        SWITCH ON READING OF KEYWORDS AND DATA
--
ENDSKIP
```

The example skips reading of the of the grid geometry data using the INCLUDE keyword, and then reverts back to reading the input files again.

4.3.7 EXTRAPMS – ACTIVATE EXTRAPOLATION WARNING MESSAGES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The EXTRAPMS keyword activates extrapolation warning messages for when OPM Flow extrapolates the PVT or VFP tables. Frequent extrapolation warning messages should be investigated and resolved as this would indicate possible incorrect data and may result in the simulator extrapolating to unrealistic values.

No.	Name	Description	Default
1	EXTRAP	Defines a single integer that activates the extrapolation warning message options for PVT and VFP tables. EXTRAP can have the following values: 1) 0 – No warning messages are give (the default). 2) 1 – PVT table extrapolation warnings are printed. 3) 2 – VFP table extrapolation warnings are printed. 4) 3 – PVT and VFP table extrapolation warnings are printed. 5) 4 - PVT and VFP table extrapolation warnings are printed with additional information.	0
Notes: 1) In addition extrapolation warnings will also be given for Rs and Rv if options (1), (3), and (4) are requested. 2) The keyword is terminated by a "/".			

Table 4.2: EXTRAPMS Keyword Description

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

Example

```
--
--       ACTIVATE EXTRAPOLATION MESSAGES
--
EXTRAPMS
         2                                                                  /
```

The above example activates the default the VFP table extrapolation warnings option.

4.3.8 FORMFEED – DEFINED THE PRINT FILE FORM-FEED CHARACTER

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The FORMFEED keyword defines the form-feed character, or carriage control character, for the output print (*.PRT) run summary (*.RSM) files. The keyword should be place at the very top of the input file.

No.	Name	Description	Default
1	FORMFEED	Defines a single integer that defines the carriage control character activates, and should be set to: 1) 1 – Standard FORTRAN (the default). 2) 2 – Form-feed character ASCII(12) 3) 3 – None.	0
Notes: 1) The keyword is terminated by a "/".			

Table 4.3: FORMFEED Keyword Description

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

Example

```
--
--       ACTIVATE EXTRAPOLATION MESSAGES
--
FORMFEED
         3                                                                  /
```

The above example sets the carriage return character to no form-feed character.

4.3.9 INCLUDE – LOAD ANOTHER DATA FILE AT THE CURRENT POSITION

Description

The INCLUDE keyword informs OPM Flow to continue reading input data from the specified INCLUDE file. When the end of the INCLUDE file is reached, or the ENDINC keyword is encountered in the included file, input data is read from the next keyword in the current file. Although INCLUDE files can be nested, that is INCLUDE files within INCLUDE files etc., in practice this should be avoided due to the complexity of tracking the files.

No.	Name	Description	Default
1	FILENAME	A character string enclosed in quotes that defines a file to read in and be processed by OPM Flow.	None
Notes: 1) The keyword is terminated by a "/".			

Table 4.4: INCLUDE Keyword Description

Examples

The first example shown below loads the grid file from the same directory as the data file.

```
--
--       LOAD INCLUDE FILE
--
INCLUDE
         'NOR-OPM-A00-GRID.inc'  /
```

The next example loads the same file one directory above from where the data file is located.

```
--
--       LOAD INCLUDE FILE
--
INCLUDE
         '../NOR-OPM-A00-FAULTS.inc'  /
```

The final example loads the same file from a separate include directory found in the parent directory relative to where the data file is located.

```
--
--       LOAD INCLUDE FILE
--
INCLUDE
         '../INCLUDE/NOR-OPM-A00-FAULTS.inc'  /
```

4.3.10 MESSAGE – OUTPUT USER MESSAGE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The MESSAGE keyword outputs a user message to the terminal, as well as to the print (*.PRT) and debug (*.DBG) files. Note this is different to the MESSAGES keyword, that defines OPM Flows message print limits and stop limits generated by the simulator.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

4.3.11 MESSAGES – DEFINE MESSAGE PRINT LIMITS AND STOP LIMITS

Description

The MESSAGES keyword defines the print and stops levels for various messages. The "print limits" set the maximum number of messages that will be printed, after which no more messages will be printed and the "stop limits" terminate the run when these limits are exceeded. There are six levels of message that increase in severity from informative all the way to programming errors, as outlined in Table 4.5.

No.	Name	Description	Default
1	PRTSMESG	An integer defining the maximum number of MESSAGE type messages to be printed after which this type of message stops printing.	1,000,000
2	PRTSCOMT	An integer defining the maximum number of COMMENT type messages be printed after which this type of message stops printing.	1,000,000
3	PRTSWARN	An integer defining the maximum number of WARNING type messages be printed after which this type of message stops printing.	10,000
4	PRTSPROB	An integer defining the maximum number of PROBLEM type messages to be printed after which this type of message stops printing.	100
5	PRTSERRS	An integer defining the maximum number of ERROR type messages to be printed after which this type of message stops printing.	100
6	PRTSBUGS	An integer defining the maximum number of BUG type messages to be printed after which this type of message stops printing.	100
7	STOPMESG	An integer defining the maximum number of MESSAGE type messages to be printed after which OPM Flow terminates the run. Not used by OPM Flow.	1,000,000
8	STOPCOMT	An integer defining the maximum number of COMMENT type messages to be printed after which OPM Flow terminates the run. Not used by OPM Flow.	1,000,000
9	STOPWARN	An integer defining the maximum number of WARNING type messages to be printed after which OPM Flow terminates the run. Not used by OPM Flow.	10,000
10	STOPPROB	An integer defining the maximum number of PROBLEM type messages to be printed after which OPM Flow terminates the run. Not used by OPM Flow.	100
11	STOPERRS	An integer defining the maximum number of ERROR type messages to be printed after which OPM Flow terminates the run. Not used by OPM Flow.	10
12	STOPBUGS	An integer defining the maximum number of BUG type messages to be printed after which OPM Flow terminates the run. Not used by OPM Flow.	1

No.	Name	Description	Default
13	PRTGRPMS	An integer defining the maximum number of GROUP MESSAGE type messages to be printed after which this type of message stops printing. Not used by OPM Flow.	10

Notes:

1) MESSAGE type messages are informative messages.
2) COMMENT type messages are probably not data errors.
3) WARNING type message are possible data errors and should be investigated.
4) PROBLEM type errors messages are associated with numerical problems.
5) ERROR type messages are errors that need to be fixed before the simulator can run the input deck.
6) BUG type of messages are potential programming errors.
7) The keyword is terminated by a "/".

Table 4.5: MESSAGES Keyword Description

Example

```
--
--       MESS  COMMT WARN  PROBL ERROR BUG   MESS COMMT WARN  PROBL ERROR BUG
--       LIMIT LIMIT LIMIT LIMIT LIMIT LIMIT STOP STOP  STOP  STOP  STOP  STOP
MESSAGES
         1*    1*    1*    1500  1*    1*    1*   1*    1*    1000  1*    1*   /
```

The above example sets the PROBLEM type message print limit to 1,500 and the stop limit to 1,000.

4.3.12 NOECHO – DEACTIVATE ECHOING OF USER INPUT FILES TO THE PRINT FILE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

Turns off echoing of all the input files to the print file. Note by default echoing of the inputs files is active. but can subsequently be switched off by the NOECHO activation keyword.

There is no data required for this keyword.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

Example

```
--
--        SWITCH OFF ECHOING OF INPUT FILES
--
NOECHO
--
--        INCLUDE SIMULATION GRID WITH SLOPING FAULTS
--
INCLUDE
          './INCLUDE/GRID/IRAP_1005.GRDECL' /
--
--        SWITCH ON ECHOING OF INPUT FILES
--
ECHO
```

The example deactivates the echoing of the input files, reads in the grid geometry data using the INCLUDE keyword, and then activates the echoing of the input files again.

Note

Especially for the large voluminous data sets in the GRID section, it is good practice to deactivate the echoing of the input files when loading this data to avoid the print output file becoming too large to view in a text editor.

4.3.13 NOWARN – DEACTIVATE WARNING MESSAGES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

Turns off warning messages to be printed to the print file; note that this keyword is deactivated by default and can subsequently be switched off by the WARN activation keyword. The warning messages may be turned on and off using keywords WARN and NOWARN.

It is recommended that WARN should always be used and action taken if necessary. For subsequent runs, the warning messages can be turned off.

There is no data required for this keyword.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

Example

```
--
--      SWITCH OFF WARNING MESSAGES
NOWARN
--
--      INCLUDE SIMULATION GRID WITH SLOPING FAULTS
--
INCLUDE
     './INCLUDE/GRID/IRAP_1005.GRDECL' /
--
--      SWITCH ON WARNING MESSAGES
--
WARN
```

The example deactivates the warning messages before reading the grid geometry data using the INCLUDE keyword, and then activates the warning messages after reading the INCLUDE file.

4.3.14 SKIP – ACTIVATE SKIPPING OF ALL KEYWORDS AND INPUT DATA

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The SKIP keyword turns on skipping of keywords until the ENDSKIP activation keyword is encountered. All keywords and between the SKIP and ENDSKIP keywords are ignored by OPM Flow. See also the SKIP100 and SKIP300 keywords that skip only the black-oil and compositional keywords, respectively.

There is no data required for this keyword.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

Example

```
--
--        SWITCH ON SKIPPING OF ALL KEYWORDS AND DATA
--
SKIP
--
--        INCLUDE SIMULATION GRID WITH SLOPING FAULTS
--
INCLUDE
          './INCLUDE/GRID/IRAP_1005.GRDECL' /
--
--        SWITCH ON READING OF ALL KEYWORDS AND DATA
--
ENDSKIP
```

The example skips reading of the of the grid geometry data using the INCLUDE keyword, and then reverts back to reading the input files again.

4.3.15 SKIP100 – ACTIVATE SKIPPING OF BLACK-OIL KEYWORDS AND INPUT DATA

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword turns on skipping of black-oil keywords until the ENDSKIP activation keyword is encountered. All black-oil keywords between the SKIP100 and ENDSKIP keywords are ignored by OPM Flow. See also the SKIP and SKIP300 keywords that skip all and only the compositional keywords, respectively. There is no data required for this keyword.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

Example

```
--
--       SWITCH ON SKIPPING OF BLACK-OIL KEYWORDS AND DATA
--
SKIP100
--
--       INCLUDE BLACK-OIL PVT DATA
--
INCLUDE
         './INCLUDE/'BLACK-OIL-PVT'         /
--
--       INCLUDE COMPOSITIONAL PVT DATA
--
INCLUDE
         './INCLUDE/'COMPOSITION-PVT-EOS'   /
--
--       SWITCH ON READING OF ALL KEYWORDS AND DATA
--
ENDSKIP
--
--       WATER PVT TABLE
--
PVTW
--       REF PRES  BW        CW        VISC     VISC
--       PSIA      RB/STB    1/PSIA    CPOISE   GRAD
--       --------  ------    -------   ------   ------
          4840.0    1.019     2.7E-6    0.370    1*         / WATER DATA REGION 1
--
--       OIL      WAT        GAS
--       DENSITY  DENSITY    DENSITY
--       -------  -------    -------
DENSITY
          39.0     62.37     0.04520                       / PVT DATA REGION 1
--       ROCK COMPRESSIBILITY
--
--       REF PRES  CF
--       PSIA      1/PSIA
--       --------  -------
ROCK
          3966.9   5.0E-06                                 / ROCK COMPRESSIBILITY
```

The example skips reading of the of the 'BLACK-OIL-PVT' data set and reads the 'COMPOSITION-PVT-EOS' file using the INCLUDE keyword, before proceeding to revert back to reading the input files again.

4.3.16 SKIP300 – ACTIVATE SKIPPING OF "COMPOSITIONAL" KEYWORDS AND INPUT DATA

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

Turns on skipping of "compositional" keywords until the ENDSKIP activation keyword is encountered. All "compositional" keywords between the SKIP300 and ENDSKIP keywords are ignored by OPM Flow. See also the SKIP and SKIP100 keywords that skip all and only the black-oil keywords, respectively. There is no data required for this keyword.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

Example

```
--
--       SWITCH ON SKIPPING OF E300 KEYWORDS AND DATA
--
SKIP300
--
--       INCLUDE BLACK-OIL PVT DATA
INCLUDE
         './INCLUDE/'BLACK-OIL-PVT'         /
--
--       INCLUDE COMPOSITIONAL PVT DATA
--
INCLUDE
         './INCLUDE/'COMPOSITION-PVT-EOS'   /
--
--       SWITCH ON READING OF KEYWORDS AND DATA
--
ENDSKIP
--
--       WATER PVT TABLE
--
PVTW
--       REF PRES  BW        CW        VISC     VISC
--       PSIA      RB/STB    1/PSIA    CPOISE   GRAD
--       --------  ------    -------   ------   ------
          4840.0    1.019     2.7E-6    0.370    1*          / WATER DATA REGION 1

--
--       OIL      WAT        GAS
--       DENSITY  DENSITY    DENSITY
--       -------  -------    -------
DENSITY
          39.0     62.37     0.04520                        / PVT DATA REGION 1

--       ROCK COMPRESSIBILITY
--
--       REF PRES  CF
--       PSIA      1/PSIA
--       --------  -------
ROCK
          3966.9   5.0E-06                                  / ROCK COMPRESSIBILITY
```

The example reads the 'BLACK-OIL-PVT' file using the INCLUDE keyword, then skips reading of the 'COMPOSITION-PVT-EOS" data set, before proceeding to revert back to reading the input files again.

4.3.17 WARN – ACTIVATE WARNING MESSAGES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

Turns on warning messages to be printed to the print file (*.PRT); note that this keyword is activated by default and can subsequently be switched off by the NOWARN activation keyword. The warning messages may be turned on and off using keywords WARN and NOWARN. OPM Flow always prints error messages.

It is recommended that WARN should always be used and action taken if necessary for the initial runs, once the run has been "cleaned up" the warning messages can be turned off.

There is no data required for this keyword.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

Example

```
--
--        SWITCH OFF WARNING MESSAGES
--
NOWARN

--
--        INCLUDE SIMULATION GRID WITH SLOPING FAULTS
--
INCLUDE
          './INCLUDE/GRID/IRAP_1005.GRDECL' /

--
--        SWITCH ON WARNING MESSAGES
--
WARN
```

The example deactivates the warning messages before reading the grid geometry data using the INCLUDE keyword, and then activates the warning messages after reading the INCLUDE file.

CHAPTER 5: RUNSPEC SECTION

5.1 INTRODUCTION

This is the first section in the OPM Flow input file and defines the key parameters for the simulator including the dimensions of the model, phases present in the model (oil, gas and water for example), number of tables for a given property and the maximum number of rows for each table, the maximum number of groups, wells and well completions, as well as various options to be invoked by the simulator.

The section is defined by the RUNSPEC keyword and is terminated by the GRID keyword.

Keywords associated with the RUNSPEC section are either Activation type keywords to switch on a feature or declare a phase, or a Vector (Row Vector) type of keyword that declares various options or array dimensions. For example, the SATOPTS keyword that activates relative permeability assignment options, or the WELLDIMS keyword that defines the maximum number of wells, completions per well, and maximum number of groups etc. for the run. The keywords can be in any order, but the values entered impact the data the simulator expects in subsequent sections. So for example if the VAPOIL keyword has been invoked to declare that the vaporized oil phase (condensate) in wet gas[63] is present in the run, then the simulator expects the PVTG keyword in the PROPS section to define the wet gas PVT properties. If the PVDG keyword (dry gas[64] PVT data set), has been entered instead in the PROPS section, then the simulator will report this as an error when it reads the PVDG keyword or after processing the PROPS section.

Note that in some instances the numerical values entered need to be the actual values, as per the DIMENS keyword, or maximum values, for example the previously mentioned WELLDIMS keyword. If the requirement for the keyword is for the actual value and an incorrect value has been entered then this may or may not be directly trapped by the simulator, and may result in numerous error message associated with the data instead. Thus, if the error reporting messages in a given section are associated with data that is required to be declared via a RUNSPEC keyword, one should also verify that the RUNSPEC keyword has been correctly defined with the associated problematic data error messages. This is particularly true for the TABDIMS keyword in the RUNSPEC section where some of the variables are defined as the actual number of tables, whereas the length of the tables are a maximum value. To be clear, if NTSFUN on the TABDIMS keyword has been set to five, then there must be exactly five sets of relative permeability data entered in the PROPS section, otherwise data input errors will occur.

5.2 DATA REQUIREMENTS

Table 5.1 outlines the minimum set of keywords required by OPM Flow in order for the simulator to successfully parse the RUNSPEC section.

No.	RUNSPEC Keyword	Description	Section Applied
1	TITLE	*TITLE – Define the Title for the Input Deck* - this is printed on most reports routed to the print file.	RUNSPEC
2	DIMENS	*DIMENS – Define the Dimensions of the Model* - the values entered must be the exact dimensions of the grid, otherwise the simulator will report errors.	GRID

[63] *Natural gas that contains significant heavy hydrocarbons such as propane, butane and other liquid hydrocarbons is known as wet gas or rich gas.*

[64] *Natural gas that occurs in the absence of condensate or liquid hydrocarbons, or gas that had condensable hydrocarbons removed, is called dry gas. It is primarily methane with some intermediates*

No.	RUNSPEC Keyword	Description	Section Applied
3	OIL, DISGAS, GAS, VAPOIL, and/or WATER	*OIL – Activate the Oil Phase in the Model* *DISGAS – Activate the Dissolved Gas Phase in the Model* *GAS – Activate the Gas Phase in the Model* *VAPOIL – Activate the Vaporized Oil in Wet Gas Phase in the Model* *WATER – Activate the Water Phase in the Model* Only the phases present in the run should be declared, so for a dry gas run then only GAS and WATER keywords should be declared, since water is always present in hydrocarbon accumulations.	PROPS
4	FIELD, LAB, or METRIC	*FIELD – Activate the Oil Field System of Units for the Model* *LAB - Activate the Laboratory System of Units for the Model* *METRIC – Activate the Metric System of Units for the Model* States the system of units to be used for the input deck.	ALL
5	WELLDIMS	*WELLDIMS – Define the Wells and Group Dimensions* Sets the well and group dimensions etc.	SCHEDULE
Notes: 1) This limited input for the RUNSPEC section is never used in practice, as real full field models are more complicated.			

Table 5.1: Minimum Set of RUNSPEC Keywords Required by OPM Flow

If the minimum set of RUNSPEC keywords is only entered, then the default values of the missing keywords will apply. This means for example only one set of PVT, relative permeability tables, and one equilibrium region will be assumed by the simulator.

A complete list of RUNSPEC keywords in alphabetic order is shown in Table 5.2 together with a generalized Topic column that classifies the functionality of the keyword. Note that not all keywords and features listed in Table 5.2 are implemented in OPM Flow. Cells not colored in the No. column indicate the keyword is supported, cells colored gray indicate that the keyword is not applicable, and finally, cells colored in orange indicate keywords that are not supported by OPM Flow.

No.	RUNSPEC Keyword	Description	Topic
1	ACTDIMS	*ACTDIMS – ACTION Keyword Dimensions*.	Action
2	ACTPARAM	*ACTPARAM – Define Action Facility Target and Tolerance Parameters*.	Action
3	AITS	*AITS – Activate Intelligent Time Stepping*.	Numeric
4	AITSOFF	*AITSOFF – Deactivate Intelligent Time Stepping*.	Numeric
5	ALKALINE	*ALKALINE – Activate the Alkaline Phase and Model*.	Alkaline
6	API	*API – Activate API Tracking*.	API
7	AQUDIMS	*AQUDIMS – Define Aquifer Dimensions*.	Aquifer

No.	RUNSPEC Keyword	Description	Topic
8	AUTOREF	*AUTOREF - Define Auto Refinement Options*.	LGR
9	BIGMODEL	*BIGMODEL – Activate Big Model Option (Retired)*.	N/A
10	BLACKOIL	*BLACKOIL – Activate Black-Oil Phases*.	Phases
11	BPARA	*BPARA – Activate Block Parallel License*.	N/A
12	BPIDIMS	*BPIDIMS – Define the Dimensions of the Interpolated Block Quantities*.	Output
13	BRINE	*BRINE – Activate Brine Tracking Option*.	Brine
14	CART	*CART – Activate Cartesian Geometry*. This the default grid type that is normally not required to be entered.	Input
15	CBMOPTS	*CBMOPTS – Define Coal Bed Methane Options*.	CBM
16	CO2STORE	*CO2STORE – Activate the CO2 Storage Model*. This is an OPM Flow implementation that is different to the commercial simulator, the keyword is a compositional keyword in the commercial simulator but has been implemented in OPM Flow's black-oil model. The keyword actives the carbon dioxide storage model for the run to account for both carbon dioxide and water phase solubility the simulators CO2-Brine PVT model.	Phases
17	COAL	*COAL – Activate the Coal Phase (CBM Model)*.	CBM
18	COLUMNS	*COLUMNS – Define Input File Column Margins*.	Input
19	CPR	*CPR – Activate Constrained Pressure Residual ("CPR") Linear Solver*. See section *2.2 Running OPM Flow 2023-10 From The Command Line* on how to invoke various numerical schemes via the OPM Flow command line interface.	Numeric
20	DEBUG	*DEBUG – Define the Debug Data to be Printed to File*.	Output
21	DIFFUSE	*DIFFUSE – Activate Molecular Diffusion Option*.	Diffusion
22	DIMENS	*DIMENS – Define the Dimensions of the Model*. The values entered must be the exact dimensions of the grid, otherwise the simulator will report errors.	Input
23	DISGAS	*DISGAS – Activate the Dissolved Gas Phase in the Model*.	Phases
24	DISGASW	*DISGASW – Activate Dissolved Gas in the Water Phase in the Model* This is an OPM Flow specific keyword that activates dissolved gas in the water phase in the model	Phases
25	DISPDIMS	*DISPDIMS – Define the Maximum Number of Dispersion Tables*.	Dispersion
26	DUALPERM	*DUALPERM – Activate Dual Permeability Model*.	Dual-Porosity

No.	RUNSPEC Keyword	Description	Topic
27	DUALPORO	*DUALPORO – Activate Dual Porosity Model*.	Dual-Porosity
28	DYNRDIMS	*DYNRDIMS – Define Dynamic Region Dimensions*.	Input
29	ECHO	*ECHO – Activate Echoing of User Input Files to the Print File*.	Output
30	ECLMC	*ECLMC – Activate Multi-Component Brine Model*.	Brine
31	END	*END – Define the End of the Input File*.	Input
32	ENDINC	*ENDINC – Define the End of an Include File*.	Input
33	ENDSCALE	*ENDSCALE – Activate Relative Permeability End-Point Scaling Option*.	End-Point
34	ENDSKIP	*ENDSKIP – DeActivate Skipping of Keywords and Input Data*.	Input
35	EQLDIMS	*EQLDIMS – Define the Equilibration Data Dimensions*.	Equilibration
36	EQLOPTS	*EQLOPTS – Activate the Equilibration Options*.	Equilibration
37	EXTRAPMS	*EXTRAPMS – Activate Extrapolation Warning Messages*.	Output
38	FAULTDIM	*FAULTDIM – Define the Number of Fault Segments*.	Input
39	FIELD	*FIELD – Activate the Oil Field System of Units for the Model*.	Input
40	FMTHMD	*FMTHMD – Activate The Format History Match Gradient File Option*.	Output
41	FMTIN	*FMTIN – Activate The Format Input File Option*.	Input
42	FMTOUT	*FMTOUT – Activate The Format Output File Option*.	Output
43	FOAM	*FOAM – Activate the Foam Phase and Model*.	Foam
44	FORMFEED	*FORMFEED – Defined the Print File Form-Feed Character*.	Output
45	FRICTION	*FRICTION – Activate Wellbore Friction Option*.	Well
46	FULLIMP	*FULLIMP – Activate Fully Implicit Solution Option*. See section *2.2 Running OPM Flow 2023-10 From The Command Line* on how to invoke various numerical schemes via the OPM Flow command line interface.	Numeric
47	GAS	*GAS – Activate the Gas Phase in the Model*.	Phases
48	GASFIELD	*GASFIELD – Define Gas Field Operations Options*.	Gas Field
49	GDIMS	*GDIMS – Activate Instantaneous Gradient Option and Define Dimensions* .	Gradient
50	GIMODEL	*GI - Define the Initial Equilibration Gi Values for All Grid Blocks*.	Phases

No.	RUNSPEC Keyword	Description	Topic
51	GRAVDR	*GRAVDR – Activate Gravity Drainage and Imbibition for Dual Porosity Model*.	Dual-Porosity
52	GRAVDRB	*GRAVDRB - Activate Vertical Discretized Gravity Drainage and Imbibition for Dual Porosity Model* .	Dual-Porosity
53	GRAVDRM	*GRAVDRM - Activate Alternative Gravity Drainage and Imbibition for Dual Porosity Model*.	Dual-Porosity
54	GRIDOPTS	*GRIDOPTS - Grid Processing Options*.	Input
55	H2STORE	*H2STORE – Activate the H2 Storage Model*. The keyword actives the hydrogen storage model for the run to account for both hydrogen and water phase solubility using the simulator's H2-Brine PVT model.	Phases
56	HMDIMS	*HMDIMS – Define History Match Gradient Parameter Dimensions*.	Gradient
57	HYST	*HYST – Activate the Hysteresis Option (Retired)*. The HYST keyword activates the hysteresis option, the keyword should be avoided and the hysteresis option should be enabled by the HYSTER parameter on the SATOTPS keyword see - *SATOPTS – Activate Relative Permeability Assignment Options* .	Hysteresis
58	IMPES	*IMPES – Activate Implicit Pressure Explicit Saturation Solution Option*. See section *2.2 Running OPM Flow 2023-10 From The Command Line* on how to invoke various numerical schemes via the OPM Flow command line interface.	Numeric
59	IMPLICIT	*IMPLICIT – Activate Fully Implicit Solution Option*. See section *2.2 Running OPM Flow 2023-10 From The Command Line* on how to invoke various numerical schemes via the OPM Flow command line interface.	Numeric
60	INCLUDE	*INCLUDE – Load Another Data File at the Current Position*.	Input
61	INSPEC	*INSPEC – Activate the INSPEC File Option*.	Output
62	LAB	*LAB - Activate the Laboratory System of Units for the Model*.	Input
63	LGR	*LGR – Define Local Grid Refinement Dimensions and Parameters*.	LGR
64	LGRCOPY	*LGRCOPY – Activate Local Grid Refinement Inheritance*.	LGR
65	LICENSES	*LICENSES – Define Required Licenses for Run*.	N/A
66	LIVEOIL	*LIVEOIL – Activate the Live Oil Phase (Oil with Free and Dissolved Gas)*.	Phases
67	LOAD	*LOAD – Load a SAVE File for a Fast Restart*.	Input
68	LOWSALT	*LOWSALT – Activate the Low Salt Brine Phase in the Brine Model*.	Brine

No.	RUNSPEC Keyword	Description	Topic
69	MEMORY	*MEMORY – Define Allocated Memory (Retired)*.	N/A
70	MESSAGE	*MESSAGE – Output User Message*. The MESSAGE keyword outputs a user message to the terminal, as well as to the print (*.PRT) and debug (*.DBG) files.	Output
71	MESSAGES	*MESSAGES – Define Message Print Limits and Stop Limits*. The MESSAGES keyword defines the print and stops levels for various messages.	Output
72	MESSSRVC	*MESSSRVC - Activate or Deactivate Database Message File Output*.	Output
73	METRIC	*METRIC – Activate the Metric System of Units for the Model*.	Input
74	MICP	*MICP – Activate the Microbially Induced Calcite Precipitation Model*. This is an OPM Flow specific keyword that activates the Microbial Induced Calcite Precipitation ("MICP") Model used to investigate CO2 leakage remediation. The module requires that both the MICP and WATER keywords in the RUNSPEC to be active.	MICP
75	MINNPCOL	*MINNPCOL - Define the Minimum Number of Newton Iterations Used to Update Well Targets*. This is an OPM Flow specific keyword that sets the minimum number of Newton iterations, as oppose to the commercial simulator's NUPCOL keyword that defines the maximum number of Newton iterations within a time step, after which well targets are frozen.	Numeric
76	MISCIBLE	*MISCIBLE – Define Miscibility Todd-Longstaff Parameters*.	Miscible
77	MONITOR	*MONITOR – Activate Output of the Monitoring Data and File*.	Output
78	MSGFILE	*MSGFILE – Active or Deactivate Message File Output*.	Output
79	MULTIN	*MULTIN – Activate the Non-Unified Multiple Input File Option*.	Input
80	MULTOUT	*MULTOUT – Activate the Non-Unified Multiple Output File Option*.	Output
81	MULTOUTS	*MULTOUTS – Activate Non-Unified Multiple Summary Output File Option*.	Output
82	MULTREAL	*MULTREAL – Activate Commercial Simulator's Multi-Realization License*.	N/A
83	NETWORK	*NETWORK – Activate the Extended Network Option and Define Parameters*.	Network
84	NINEPOIN	*NINEPOIN – Activate the Nine-Point Discretization Option*.	Numeric
85	NMATRIX	*NMATRIX – Activate the Discretized Matrix Dual Porosity Option*.	Dual-Porosity
86	NNEWTF	*NNEWTF – Activate the Non-Newtonian Fluid Model*.	Numeric

No.	RUNSPEC Keyword	Description	Topic
87	NOCASC	*NOCASC – Activate Linear Solver Tracer Algorithm*. See section *2.2 Running OPM Flow 2023-10 From The Command Line* on how to invoke various numerical schemes via the OPM Flow command line interface.	Tracers
88	NODPPM	*NODPPM – Deactivate Fracture Porosity-Permeability Calculation*	Dual-Porosity
89	NOECHO	*NOECHO – Deactivate Echoing of User Input Files to the Print File*.	
90	NOHYST	*NOHYST - Deactivate the Hysteresis Option*.	Hysteresis
91	NOINSPEC	*NOINSPEC – Deactivate Output of the INIT Index File*.	Output
92	NOMONITO	*NOMONITO – Deactivate Output of the Monitoring Data and File*.	Output
93	NONNC	*NONNC – Deactivate Non-Neighbor Connections*.	Input
94	NORSSPEC	*NORSSPEC – Deactivate Output of the RESTART Index File*.	Output
95	NOSIM	*NOSIM – Activate the No Simulation Mode for Data File Checking*.	Numeric
96	NOWARN	*NOWARN – Deactivate Warning Messages*.	Output
97	NRSOUT	*NRSOUT – Defined Maximum Number of RESTART Elements* .	Output
98	NSTACK	*NSTACK – Define the Stack Length for the Iterative Linear Solver*.	Numeric
99	NUMRES	*NUMRES – Define the Number of Reservoir Grids*. Currently, this should only be set to one in OPM Flow.	Input
100	NUPCOL	*NUPCOL – Define the Maximum Number of Newton Iterations Used to Update Well Targets*.	Numeric
101	OIL	*OIL – Activate the Oil Phase in the Model*.	Phases
102	OPTIONS	*OPTIONS – Activate Various Program Options*.	Options
103	PARALLEL	*PARALLEL – Define Parallel Run Configuration*. The keyword defines the run to use parallel processing and sets the domain decomposition options. See section *2.2 Running OPM Flow 2023-10 From The Command Line* on how to run OPM Flow in parallel mode.	Numeric
104	PARTTRAC	*PARTTRAC – Activate and Define Partitioned Tracer Option*.	Tracers
105	PATHS	*PATHS – Define Filename Directory Path Aliases*.	Input
106	PEDIMS	*PEDIMS – Define Petro-Elastic Model Regions and Table Dimensions*.	Petro-Elastic
107	PETOPTS	*PETOPTS – Define Petrel and Generic Simulation File Options*.	Input
108	PIMTDIMS	*PIMTDIMS – Define Well Productivity Scaling Table Dimensions*.	Wells

No.	RUNSPEC Keyword	Description	Topic
109	PINTDIMS	*PINTDIMS – Define Polymer Molecular Weight Model Table Dimensions*. This is an OPM Flow specific keyword for the simulator's Polymer Molecular Weight Transport option that uses the polymer molecular weight in calculating the polymer viscosity, as well as accounting for formation damage due to the water and polymer injection, by adjusting the wellbore skin pressure.	Polymer
110	POLYMER	*POLYMER – Activate the Polymer Phase in the Model*.	Polymer
111	POLYMW	*POLYMW – Activate the Polymer Molecular Weight Transport Option* This is an OPM Flow specific keyword for the simulator's Polymer Molecular Weight Transport option that uses the polymer molecular weight in calculating the polymer viscosity, as well as accounting for formation damage due to the water and polymer injection, by adjusting the wellbore skin pressure.	Polymer
112	PRECSALT	*PRECSALT – Activate the OPM Flow Salt Precipitation Model*. This is an OPM Flow specific keyword for the simulator's Salt Precipitation Model, note that this is an extension to the commercial simulator's Brine Model.	Water Vaporization
113	PSTEADY	*PSTEADY – Activate Pseudo Steady State Flow Calculation Option*.	Numeric
114	RADIAL	*RADIAL – Radial Grid Activation Option*.	Radial
115	REGDIMS	*REGDIMS – Define the Maximum Number of Regions for a Region Array*.	Input
116	RIVRDIMS	*RIVRDIMS – Define the River Dimensions and Associated Data*.	Rivers
117	ROCKCOMP	*ROCKCOMP – Activate Rock Compaction* .	Compaction
118	RPTCPL	*RPTCPL – Activate Couple Simulation Reporting*.	Output
119	RPTHMD	*RPTHMD - Define Well History Match Gradient Reporting Options*.	Gradient
120	RPTRUNSP	*RPTRUNSP – Activate RUNSPEC Reporting*.	Output
121	RSSPEC	*RSSPEC – Activate Output of the RESTART Index File*.	Output
122	RUNSPEC	*RUNSPEC - Define the Start of the RUNSPEC Section of Keywords*	
123	SAMG	*SAMG – Activate Algebraic Multi-Grid Linear Solver*. See section *2.2 Running OPM Flow 2023-10 From The Command Line* on how to invoke various numerical schemes via the OPM Flow command line interface.	Numeric
124	SATOPTS	*SATOPTS – Activate Relative Permeability Assignment Options* .	Input
125	SAVE	*SAVE – Activate Output of a SAVE File for Fast Restarts*.	Output

No.	RUNSPEC Keyword	Description	Topic
126	SCDPDIMS	*SCDPDIMS – Define Scale Deposition and Damage Table Dimensions*.	Scale Deposition
127	SKIP	*SKIP – Activate Skipping of All Keywords and Input Data*.	Input
128	SKIP100	*SKIP100 – Activate Skipping of Black-Oil Keywords and Input Data*.	Input
129	SKIP300	*Error: Reference source not found*.	Input
130	SMRYDIMS	*SMRYDIMS – Define Maximum Number of Summary Vectors to be Written* .	Output
131	SOLVDIMS	*SOLVDIMS – Define PEBI Grid Nested Factorization Solver Dimensions* . See section *2.2 Running OPM Flow 2023-10 From The Command Line* on how to invoke various numerical schemes via the OPM Flow command line interface.	Numeric
132	SOLVENT	*SOLVFRAC – Define the Initial Gas Solvent Fraction for All Grid Blocks*	Solvent
133	SPIDER	*SPIDER – Spider Grid Activation Option*. This is an OPM Flow specific keyword for the simulator's Spider Grid option, that emulates a Radial Grid via corner-point geometry. The option employs the standard GRID section radial grid keywords to construct the grid - see the example in the section on Spider Grids (*Spider Grids*).	Radial
134	START	*START – Simulation Start Date*.	Numeric
135	SURFACT	*SURFACT – Activate the Surfactant Phase in the Model*.	Surfactant
136	SURFACTW	*SURFACTW – Activate the Surfactant Phase with Wettability Changes in the Model*.	Surfactant
137	TABDIMS	*TABDIMS – Define the Number of Tables and the Table Dimensions*.	Input
138	TEMP	*TEMP – Activate the Temperature Modeling Option*. The temperature option (TEMP keyword) and the thermal option (THERMAL keyword) are two separate modeling facilities in the commercial simulator. See *THERMAL– Activate the Thermal Modeling Option* for additional information.	Thermal
139	THERMAL	*THERMAL– Activate the Thermal Modeling Option*. This keyword activates OPM Flow's thermal modeling option. The energy black-oil implementation in OPM Flow is a mixture of the commercial simulators black-oil and the commercial simulator's "compositional thermal" keywords, as well as some OPM Flow specific keywords. See the keyword for further information.	Thermal
140	TITLE	*TITLE – Define the Title for the Input Deck*.	Input
141	TRACERS	*TRACERS – Activate Tracer Options and Set Tracer Array Dimensions*.	Tracers

No.	RUNSPEC Keyword	Description	Topic
142	TRPLPORO	*TRPLPORO – Activate the Triple Porosity Model Option*.	Triple-Porosity
143	UDADIMS	*UDADIMS – Define the Dimensions of the User Defined Arguments*.	Action
144	UDQDIMS	*UDQDIMS – Define the Dimensions of the User Defined UDQ Feature*.	Action
145	UDQPARAM	*UDQPARAM – Define Parameters for the User Defined Quantity Feature*.	Action
146	UDTDIMS	*UDTDIMS – Define the Dimensions of the User Defined Tables*.	Action
147	UNCODHMD	*UNCODHMD – Activate History Match Gradient Unencoded Output*.	Gradient
148	UNIFIN	*UNIFIN – Activate The Unified Input File Option*.	Input
149	UNIFOUT	*UNIFOUT – Activate The Unified Output File Option*.	Output
150	UNIFOUTS	*UNIFOUTS – Activate The Unified Output Summary File Option*.	Output
151	UNIFSAVE	*UNIFSAVE – Activate The Unified Output Save File Option*.	Output
152	VAPOIL	*VAPOIL – Activate the Vaporized Oil in Wet Gas Phase in the Model*	Phases
153	VAPWAT	*VAPWAT – Activate Vaporized Water in the Dry and Wet Gas Phases*. This is an OPM Flow specific keyword for the simulator's Water Vaporization Model that is activated by declaring that vaporized water is present in the run.	Water Vaporization
154	VE	*VE – Activate Vertical Equilibrium Model (Global)*.	Input
155	VFPIDIMS	*VFPIDIMS – Injection Vertical Flow Performance Table Dimensions*.	Well
156	VFPPDIMS	*VFPPDIMS – Production Vertical Flow Performance Table Dimensions*.	Well
157	VISAGE	*VISAGE - Activate External Reservoir Geo-Mechanics VISAGE Option*.	N/A
158	VISCD	*VISCD – Activate Dual Porosity Viscous Displacement Option* .	Dual-Porosity
159	WARN	*WARN – Activate Warning Messages*.	Output
160	WATER	*WATER – Activate the Water Phase in the Model*.	Phases
161	WELLDIMS	*WELLDIMS – Define the Wells and Group Dimensions*.	Well
162	WPOTCALC	*WPOTCALC – Well Potential Calculation Options*.	Well
163	WSEGDIMS	*WSEGDIMS – Define Multi-Segment Well Dimensions*.	Multi-Segment Well

No.	RUNSPEC Keyword	Description	Topic

Notes:

1) Note that not all keywords and features listed above are implemented in OPM Flow. Cells not colored in the No. column indicate the keyword is supported or partially supported by OPM Flow, cells colored gray indicate that the keyword is not applicable, and finally, cells colored in orange indicate keywords that are not supported by OPM Flow and will be ignored by the simulator.
2) For the Topic column the definition of the entries is defined as follows:
   1) **Action:** Keywords associated with the ACTIONX or OPM Flow's Python facility.
   2) **API:** Keywords associated with the API option.
   3) **Brine:** Keywords associated with the BRINE option.
   4) **CBM:** COAL BED METHANE activation and options (not implemented in OPM Flow).
   5) **Diffusion:** DIFFUSION model activation and options.
   6) **Dispersion:** DISPERSION model activation and options.
   7) **Dual-Porosity:** DUAL POROSITY and DUAL PERMEABILITY model activation and parameters.
   8) **End-Point:** END-POINT SCALING activation and options.
   9) **Equilibration** EQUIL section control and options.
   10) **Gas Field:** GAS FIELD model activation, control and options.
   11) **Gradient:** Keywords associated with the GRADIENT option, used in history matching.
   12) **Foam:** FOAM model activation.
   13) **Hysteresis:** HYSTERESIS model.
   14) **Input:** Input control and options.
   15) **LGR:** LOCAL GRID REFINEMENT dimensions and options.
   16) **MICP:** MICROBIAL INDUCED CALCITE PRECIPITATION model activation.
   17) **Multi-Segment**: Multi-Segment well dimensions and options.
   18) **Network:** Activation and options for the NETWORK model.
   19) **Numeric**: Keywords related to the various numeric control and options for the run.
   20) **Output:** Output control and options.
   21) **Phases:** Declarations related to the active phases.
   22) **Scale Deposition:** Scale and BRINE options.
   23) **Solvent:** SOLVENT phase options
   24) **Surfactant**: SURFACTANT phase activation and options.
   25) **Thermal:** THERMAL model activation and options.
   26) **Tracers:** TRACER model options.
   27) **Triple-Porosity:** TRIPLE POROSITY model option activation.
   28) **Water Vaporization:** WATER VAPORIZATION activation.

Table 5.2:Alphabetic List of RUNSPEC Keywords

Note that a number of keywords can be classified under several topics, so the Topic column should only be used as a general reference guide.

Example

A typical RUNSPEC section using various keywords is shown below and on the next few pages for reference.

```
-- ================================================================================
--
-- RUNSPEC SECTION
--
-- ================================================================================
RUNSPEC
--
--       DEFINE THE TITLE FOR THE RUN
--
TITLE
CASE CASENAME - SIMULATION RUN TITLE
--
--       DEFINE THE START DATE FOR THE RUN
--
START
         01 'JAN' 2021                                                        /
--
--       SWITCH NO SIMULATION MODE FOR DATA CHECKING COMMENT OUT TO RUN THE MODEL
--
NOSIM
-- --------------------------------------------------------------------------------
-- FLUID TYPES AND TRACER OPTIONS
-- --------------------------------------------------------------------------------
--
--       OIL PHASE IS PRESENT IN THE RUN
--
OIL
--
--       WATER PHASE IS PRESENT IN THE RUN
--
WATER
--
--       GAS PHASE IS PRESENT IN THE RUN
--
GAS
--
--       DISSOLVED GAS IN LIVE OIL IS PRESENT IN THE RUN
--
DISGAS
--
--       VAPORIZED OIL IN WET GAS IS PRESENT IN THE RUN
--
VAPOIL
--
--       NUMBER AND TYPE OF TRACERS
--       NO OIL  NO WAT  NO GAS  NO ENV  DIFF    MAX    MIN    TRACER
--       TRACERS TRACERS TRACERS TRACERS CONTL   NONLIN NONLIN NONLIN
TRACERS
         0       0       1       0      'NODIFF' 1*     1*     1*             /
```

```
-- ------------------------------------------------------------------------
-- GRID AND EQUILBRATION DIMENSIONS AND OPTIONS
-- ------------------------------------------------------------------------
--
--       MAX     MAX     MAX
--       NDIVIX  NDIVIY  NDIVIZ
DIMENS
         46      112     22                                                /
--
--       FAULT
--       SEGMS
FAULTDIM
         10000                                                             /
--
--       MAX     MAX     RSVD    TVDP    TVDP
--       EQLNUM  DEPTH   NODES   TABLE   NODES
EQLDIMS
         9       1*      20      1       1                                 /
--
--       MAX     TOTAL   INDEP   FLUX    TRACK  CBM    OPERN  WORK  WORK  POLY
--       FIPNUM  REGNS   REGNS   REGNS   REGNS  REGNS  REGNS  REAL  INTG  REGNS
REGDIMS
         9       9       1*      1*      1*     1*     1*     1*    1*    1*   /
--
--       NEG      MAX     MAX
--       MULTS    MULTNUM PINCHNUM
GRIDOPTS
         YES      9       1*                                                /
--
--       ACTIVATE EQUILIBRATION OPTIONS
--       MOBILE ENDPOINT(MOBILE) STEADY STATE(QUIESC) THRESHOLD(THPRES)
--       IRREVERSIBLE THRESHOLD(IRREVERS)
EQLOPTS
         'MOBILE'   'QUIESC'   'THPRES'   'IRREVERS'                       /
-- ------------------------------------------------------------------------
-- ROCK AND SATURATION TABLES DIMENSIONS AND OPTIONS
-- ------------------------------------------------------------------------
--
--       MAX     MAX     MAX     MAX     MAX     MAX    E300
--       NTSFUN  NTPVT   NSSFUN  NPPVT   NTFIP   NRPVT  BLANK  NTEND
TABDIMS
         15      9       40      30      9       25     1*     1              /
--
--       DIRC   REVERSE  MAX     MAX
--       SCALE  SCALE    TABLES  NODES
ENDSCALE
         NODIR  REVERS   1*      1*                                        /
--
--       ACTIVATE RELATIVE PERMEABILITY ASSIGNMENT HYSTERESIS OPTIONS
--       DIRECTTIONAL(DIRECT) IRREVERSIBLE(IRREVERS) HYSTERESIS(HYSTER)
SATOPTS
         'DIRECT'   'IRREVERS'   'HYSTER'                                  /
```

```
-- ------------------------------------------------------------------------
-- GROUP, WELL AND VFP TABLE DIMENSIONS
-- ------------------------------------------------------------------------
--
--       WELL    WELL    GRUPS   GRUPS
--       MXWELS  MXCONS  MXGRPS  MXGRPW
WELLDIMS
         60      110     18      40                                        /
--
--       DEFINE WELL PRODUCTIVITY SCALING TABLE DIMENSIONS
--       MAX     MAX
--       TABLES  ENTRIES
PIMTDIMS
         1       51                                                        /
--
--       INJECTING VFP TABLES
--       VFP     VFP     VFP
--       MXMFLO  MXMTHP  NMMVFT
VFPIDIMS
         10      10      12                                                /
--
--       PRODUCING VFP TABLES
--       VFP     VFP     VFP     VFP     VFP     VFP
--       MXMFLO  MXMTHP  MXMWFR  MXMGFR  MXMALQ  NMMVFT
VFPPDIMS
         20      10      10      10      6       9                         /
-- ------------------------------------------------------------------------
-- MISCELLANEOUS OPTIONS
-- ------------------------------------------------------------------------
--
--       USER DEFINED ARGUMENT DIMENSIONS FACILITY
--       MAX     MAX     MAX    MAX    MAX    MAX   MAX   MAX   MAX   MAX    RAND
--       FUNCS   ITEMS   CONNS  FIELD  GROUP  REGS  SEGTM WELL  AQUF  BLCKS  OPT
UDQDIMS
         16      16      0      0      0      0     0     0     0     0      N  /
--
--       USER DEFINED ARGUMENT DIMENSIONS
--       NO.     NOT     TOTAL
--       ARGS    USED    UDQ
UDADIMS
         2       1*      1*                                                /
--
--       USER DEFINED DEFAULT VALUES
--       SEED    RANGE    UNDEFINED   COMPARISON
--       INTG    -AND+    VALUE       TOLERANCE
UDQPARAM
         1       1*       0.0         1.0E-4                               /
-- ------------------------------------------------------------------------
-- NUMERICAL AND RUN CONTROL OPTIONS
-- ------------------------------------------------------------------------
--
--       SET STACK SIZE FOR LINEAR SOLVER
--
NSTACK
         30                                                                /
--
--       DEFINE THE NUMBER OF ITERATIONS TO UPDATE WELL FLOW TARGETS
--
NUPCOL
         3                                                                 /
```

```
--
--       SKIP      ACTIVATE
--       OPTIONS   OPTION
OPTIONS
         77*0      1                                                        /
-- ------------------------------------------------------------------------
-- INPUT AND OUTPUT OPTIONS
-- ------------------------------------------------------------------------
--
--       METRIC SYSTEM OF UNITS FOR BOTH INPUT AND OUTPUT
--
METRIC
--
--       SWITCH ON THE UNIFIED INPUT FILES OPTION
--
UNIFIN
--
--       SWITCH ON THE UNIFIED OUTPUT FILES OPTION
--
UNIFOUT
--
--       MESS  COMMT WARN  PROBL ERROR BUG   MESS COMMT WARN  PROBL ERROR BUG
--       LIMIT LIMIT LIMIT LIMIT LIMIT LIMIT STOP STOP  STOP  STOP  STOP  STOP
MESSAGES
         3000  1*    1000  1000  1*    1*    1*   1*    9000  1*    9000  1*   /
--
--       DEBUG PRINTING OPTIONS
--
DEBUG
         8*0   1     11*0  1     30*0                                       /
-- ========================================================================
```

The above example is rather extensive but is not unrealistic in terms of the features being activated and the array size declarations.

Previously on 32-bit systems, array sizes were important to ensure that there was sufficient addressable memory to load the input deck and to be able to run given the memory limitations, this is no longer a concern on modern 64-bit systems like Linux.

5.3 KEYWORD DEFINITIONS

5.3.1 ACTDIMS – ACTION KEYWORD DIMENSIONS

Description

The ACTDIMS keyword defines the maximum number of properties associated with the ACTION keyword. The ACTION keyword allows the user to enter computational logic and calculations to the simulation run based on the how the simulation run is proceeding.

No.	Name	Description	Default
1	MXACTNS	A positive integer value that defines the maximum number of ACTION keywords defined in the input deck.	2
2	MXLINES	A positive integer value that defines the maximum number of lines in an ACTION statement.	50
3	MXCHARS	A positive integer value that defines the maximum characters in an ACTION statement.	80
4	MXSTATMS	A positive integer value that defines the maximum number of conditional statements in the ACTION statement.	3
Notes: 1) The keyword is terminated by a "/".			

Table 5.3:ACTDIMS Keyword Description

Only the ACTIONX keyword and computational logic have been implemented in OPM Flow.

Example

```
--       ACTION    ACTION   ACTION   ACTION
--       MXACTNS   MXLINES  MXCHARS  MXSTATMS
ACTDIMS
         2         50       80       3                                   /
```

The above example defines the default values for the ACTDIMS keyword.

5.3.2 ACTPARAM – DEFINE ACTION FACILITY TARGET AND TOLERANCE PARAMETERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The ACTPARAM keyword defines the maximum target percent value for the ACTION series of keywords and the fractional equality tolerance for determining if two numbers are numerically equal when comparing values using the ACTION series of keywords. The ACTION keyword allows the user to enter computational logic and calculations to the simulation run based on the how the simulation run is proceeding.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	MXTOLS	A positive real value that defines the maximum target percent number for the ACTION series of keywords. The default value of 100 means the target is not applied.			
		percent 100.0	percent 100.0	percent 100.0	Defined
2	MXEQLS	MXEQLS a real positive number greater than zero and less than one that defines the tolerance used to determine if two real values are equal for comparing values in the ACTION series of keywords. Floating-point numbers (as implemented in computers) are never exact, one cannot compare floating point numbers for exact equality. Thus, MXEQLS defines a tolerance. For example, the default value of 1 x 10-4 means that if the difference between two real values is less than 1 x 10-4 then the values are considered equal.			
		fraction 1 x 10-4	fraction 1 x 10-4	fraction 1 x 10-4	Defined
Notes: 1) The keyword is terminated by a "/".					

Table 5.4:ACTPARAM Keyword Description

Although this keyword is read by OPM Flow, the ACTION and UDQ computational logic and calculations have not been fully implemented and therefore this keyword should not be used as it may result in OPM Flow terminating.

Example

```
--
--        ACTION    ACTION
--        MXTOLS    MXEQLS
ACTPARAM
          5.0       1.0E-4                                          /
```

The above example defines the maximum tolerance to be 5% and the equality tolerance to be the default value of 1.0 x 10-4.

5.3.3 AITS – ACTIVATE INTELLIGENT TIME STEPPING

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

Turns on the commercial simulator's intelligent time stepping.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

5.3.4 AITSOFF – DEACTIVATE INTELLIGENT TIME STEPPING

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

Turns off the commercial simulator's intelligent time stepping.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

5.3.5 ALKALINE – ACTIVATE THE ALKALINE PHASE AND MODEL

Description

This keyword indicates that an alkaline phase is present in the model and to activate the Alkaline Model in the run. The keyword will also invoke data input file checking to ensure that all the required Alkaline Model input parameters are defined in the input deck.

There is no data required for this keyword and there is no terminating "/" for this keyword.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

Example

```
--
--       ALKALINE PHASE IS PRESENT IN THE RUN
--
ALKALINE
```

The above example declares that the alkaline phase is active in the model to activate Alkaline Model.

5.3.6 API – ACTIVATE API TRACKING

Description

This keyword switches on API tracking so that the various "oil types" are tracked in the model.

In many reservoirs the initial API gravity of oil varies with depth due to the heavy viscous fractions occupying the deepest part of the reservoir whilst the lighter more mobile fractions will occupy the upper part of the reservoir. As a reservoir is depleted the API gravity of oil in a cell will gradually change as the different fluids mix.

In OPM Flow it is possible to define different PVT regions in a reservoir, as in all finite difference formulated simulators, oil moving from one region to another will suddenly assume the properties of that region it has moved to. The fluid type tracking option allows the smooth change of PVT properties in a cell to be simulated by correlating PVT properties against the API gravity of oil.

There is no data required for this keyword and there is no terminating "/" for this keyword.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

Example

```
--
--       ACTIVATE THE API TRACKING OPTION
--
API
```

The above example switches on the API tracking facility.

5.3.7 AQUDIMS – DEFINE AQUIFER DIMENSIONS

Description

The AQUDIMS keyword defines the dimensions of the various aquifer property data. The data is normally entered on a single line and is terminated by a "/".

No.	Name	Description	Default
1	MXNAQN	A positive integer value that defines the AQUNUM keyword maximum number of lines associated with this keyword, that is the maximum number of numerical aquifers	1
2	MXNAQC	A positive integer value that defines the AQUCON keyword maximum number of lines of connection data associated with this keyword, that is the maximum number of lines of connection data for numerical aquifers.	1
3	NIFTBL	A positive integer value that defines the AQUTAB keyword maximum number of Carter-Tracy aquifer tables associated with this keyword.	1
4	NRIFTB	A positive integer value that defines the AQUTAB keyword maximum number of rows in the Carter-Tracy aquifer tables associated with this keyword. NRIFTB must not be less than 36 in order to accommodate the default infinite acting Carter-Tracy aquifer influence function.	36
5	NANAQ	A positive integer value that defines the AQUFETP, AQUFLUX and AQUCT maximum number of analytical aquifers defined by these three keywords.	1
6	NCAMAX	A positive integer value that defines the maximum number of cells connected to an analytical aquifer.	1
7	MXNALI	A positive integer value that defines the maximum number of aquifer lists.	0
8	MXAAQL	A positive integer value that defines the maximum number of analytical aquifers in any single aquifer list as defined by (7).	0

Notes:

1) The keyword is terminated by a "/".

Table 5.5:AQUDIMS Keyword Description

Example

```
--
--        AQF      AQF      AQF      AQF      AQF      AQF      AQF      AQF
--        MXAQN    MXNAQC   NIFTBL   NRIFTB   NANAQU   NCAMAX   MXNALI   MXAAQL
AQUDIMS
          1*       1*       1*       1*       1*       1*       1*       1*              /
```

The above example defines the default values for the AQUDIMS keyword.

5.3.8 AUTOREF - DEFINE AUTO REFINEMENT OPTIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The AUTOREF keyword activates the Auto Refinement option and defines the parameters for this feature.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

5.3.9 BIGMODEL – ACTIVATE BIG MODEL OPTION (RETIRED)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The original intention in the commercial simulator was to define an optimized memory allocation method to handle large models; this has since become redundant and has been retired in the commercial simulator.

This keyword is ignored by both OPM Flow and the commercial simulator and has no effect on the simulation but is documented here for completeness.

5.3.10 BLACKOIL – ACTIVATE BLACK-OIL PHASES

Description

This keyword switches on the black-oil formulation, and is equivalent to setting the phases present in the model to be oil, vaporized oil, gas, and dissolved gas. Note if water is present in the model this needs to be explicitly stated via the WATER keyword in the RUNSPEC section (see also the LIVEOIL keywords in the RUNSPEC section). The keyword is used by the commercial simulator's compositional THERMAL option to set the phases present in the model.

There is no data required for this keyword and there is no terminating "/" for this keyword.

Example

The following example activates the black-oil phases in the model.

```
--
--       ACTIVATE BLACK-OIL PHASES
--
BLACKOIL
```

Alternatively one could explicitly declare the phases using the following keywords in the RUNSPEC section.

```
--
--       OIL PHASE IS PRESENT IN THE RUN
--
OIL
--
--       VAPORIZED OIL IN WET GAS IS PRESENT IN THE RUN
--
VAPOIL
--
--       GAS PHASE IS PRESENT IN THE RUN
--
GAS
--
--       DISSOLVED GAS IN LIVE OIL IS PRESENT IN THE RUN
--
DISGAS
```

The above example switches on the black-oil phases in the model.

5.3.11 BPARA – ACTIVATE BLOCK PARALLEL LICENSE

Description

The BPARA keyword activates the block parallel license in the commercial simulator. There is no data required for this keyword; however the keyword should be followed by the PARALLEL keyword in the RUNSPEC section, as illustrated in the example below.

There is no data required for this keyword and there is no terminating "/" for this keyword.

OPM Flow is an open source project and therefore there is no license management of the various implemented options or the number of cores/threads that can be utilized; hence this keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

See section *2.2 Running OPM Flow 2023-10 From The Command Line* on how to run OPM Flow in parallel mode.

Example

```
--
--       ACTIVATE BLOCK PARALLEL LICENSE
--
BPARA
--
--       PARALLEL MULTI-CORE OPTIONS
--       NDMAIN     MACHINE TYPE
PARALLEL
           8      DISTRIBUTED                                                /
```

The above example sets the number of domains (or processors) to eight and for the simulation to run in block parallel mode. This has no effect in OPM Flow input decks.

5.3.12 BPIDIMS – DEFINE THE DIMENSIONS OF THE INTERPOLATED BLOCK QUANTITIES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The BPDIMS keyword defines the dimensions of the interpolated grid block quantities for the BPR_X, BHD_X,BHDF_X, BSCN_X and BCTRA_X, etc. variables declared in the SUMMARY section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

5.3.13 BRINE – ACTIVATE BRINE TRACKING OPTION

Description

The BRINE keyword activates the standard Brine Tracking model and optionally defines the water phase to have various salinities if the ECLMC keyword in the RUNSPEC section has been used to activate the Multi-Component Brine model, that allows for the water phase to have multiple water salinities. Note that the Multi-Component Brine model is not supported by OPM Flow.

No.	Name	Description	Default
1	SALTS	An optional character vector string that defines the salts to be tracked for when the Multi-Component Brine model has been activated by the ECLMC keyword in the RUNSPEC section. SALTS should be set to one or more of the following salt chemical formulae: Salt Name — Salt Chemical Formulae Sodium Chloride — NaCl Potassium Chloride — KCl Calcium Chloride — $CaCl_2$ Magnesium Chloride — $MgCl_2$ Sodium Carbonate — Na_2CO_3 Potassium Carbonate — K_2CO_3 Calcium Carbonate — $CaCO_3$ Magnesium Carbonate — $MgCO_3$ Sodium Sulfate — Na_2SO_4 Potassium Sulfate — K_2SO_4 Calcium Sulfate — $CaSO_4$ Magnesium Sulfate — $MgSO_4$ Note that the ECLMC option is currently not available in OPM Flow, so only the BRINE keyword without the optional SALT variables should be declared in the input deck.	None

Notes:

1) There is no data required for this keyword if the standard Brine Tracking option is being activated and there should be no terminating "/" in this case. However, if the Multi-Component Brine Tracking option has been invoked by the ECLMC keyword, a list of SALTS must be supplied and in this case.
2) The keyword is terminated by a "/".

Table 5.6: BRINE Keyword Description

See also the PRECSALT and VAPWAT keywords in the RUNSPEC section that activates OPM Flow's Salt Precipitation model, and the PVTWSALT keyword in the PROPS section to define the water properties with respect to salt concentration.

Example

The first example actives the standard Brine model and has no terminating "/".

```
--
--       ACTIVATE STANDARD BRINE MODEL IN THE RUN
--
BRINE
```

The second example illustrates how to activate OPM Flow's Salt Precipitation model.

```
--
--       ACTIVATE STANDARD BRINE MODEL IN THE RUN
--
BRINE
--
--       ACTIVATE THE OPM FLOW SALT PRECIPITATION MODEL (OPM FLOW KEYWORD)
--
PRECSALT
--
--       VAPORIZED WATER IN DRY/WET GAS IS PRESENT IN THE RUN (OPM FLOW KEYWORD)
--
VAPWAT
```

The third and final example activates the Multi-Component brine model with four different salts.

```
--
--       ACTIVATE MULTI-COMPONENT BRINE MODEL
--
ECLMC
--
--       DEFINE WATER PHASE MULTI-COMPONENT BRINE COMPONENTS
--
--       SALT1   SALT2   SALT3   SALT4   SALT5
BRINE
         NACL    CACL2   MGC03   K2CO3                                    /
```

This option is currently not available in OPM Flow.

5.3.14 CART – ACTIVATE CARTESIAN GEOMETRY

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

CART activates the Cartesian grid geometry for the main model, as oppose to a radial geometry. This is the default geometry and therefore the keyword does have to be used to activate this type of geometry.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

5.3.15 CBMOPTS – DEFINE COAL BED METHANE OPTIONS

Description

This keyword sets the options for the Coal Bed Methane model which is activated via the COAL keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

5.3.16 CO2STORE – ACTIVATE THE CO2 STORAGE MODEL

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The CO2STORE keyword activates the carbon dioxide (CO_2) storage model for the run to account for both carbon dioxide and water phase solubility, via the simulator's CO_2-Brine PVT model. This keyword is a compositional keyword in the commercial simulator but has been implemented in OPM Flow's black-oil model.

The CO_2-Brine PVT model computes the PVT properties such as density, viscosity, and enthalpy internally as functions of pressure, temperature, and composition by using analytic correlations and models from the literature rather than by interpolation from tabulated values. These values are transformed to the standard black-oil equivalent PVT tables internally by the simulator. A full description of the underlying PVT models is described by Sandve et al.[65]. This means that the normal PVT keywords like DENSITY, PVTO, PVDG etc. are not required by OPM Flow when this model is activated, and if entered will be ignored by the simulator. Note that the CO_2-Brine PVT properties depend on the temperature and salinity and these must therefore be entered in the PROPS section. The reservoir temperature can be defined using, e.g., the RTEMP keyword. Region based salinity can be provided using the SALINITY keyword.

The CO2STORE keyword must be used together with either: (1) the GAS and WATER keywords (or alternatively the GASWAT keyword), or (2) the GAS and OIL keywords in the RUNSPEC section. It is recommended that the standard method option (1) is used.

The DISGASW keyword in the RUNSPEC section can be used with option (1) to model dissolution of CO_2 in the Brine.

Option (1) has the advantage that it can be used with the VAPWAT and PRECSALT keywords in the RUNSPEC section to model the impact of both vaporization of residual water and salt precipitation in the near wellbore region on injectivity of CO_2 injection wells.

In option (1), the GAS and WATER (or GASWAT) keywords declare that the gas and water phases are present in the model. When the CO2STORE option is used the water phase represents the brine and the gas phase represents CO_2. Note that the input and output keywords need to be consistent with this assumption, e.g., GSF (gas saturation function) and WSF (water saturation function) should be used for the CO_2-Brine relative permeability, etc.

Although, the DISGAS and VAPOIL keywords can be used with option (2) to model CO_2 dissolution and water vaporization, salt precipitation is not supported with option (2).

Option (2) currently has the advantage that it can be used with the DRSDTCON keyword in the SCHEDULE section to control convective dissolution of CO_2 into the in-situ brine. Support for the use of DRSDTCON with option (1) is planned to be added in the next release.

In option (2), the GAS and OIL keywords declare that the gas and oil phases are present in the model. Internally when CO2STORE is used the oil phase refers to the brine and the gas phase to CO_2. Again, the input and output keywords need to be consistent with this assumption, e.g., SGOF (gas-oil relative permeability) is used for the CO_2-Brine relative permeability, FOIP (Field Oil-In-Place) shows the total amount of brine in the reservoir, etc.

There is no data required for this keyword and there is no terminating "/" for this keyword.

[65] *Tor Harald Sandve1, Sarah E. Gasda, Atgeirr Rasmussen, and Alf Birger Rustad. Convective dissolution in field scale CO_2 storage simulation using the OPM Flow simulator. Submitted to TCCS 11 – Trondheim Conference on CO_2 Capture, Transport and Storage Trondheim, Norway – June 21-23, 2021.*

Examples

The first example shows the standard usage of CO2STORE with Option (1) the Gas-Water model (GASWAT). Here we also activate the dissolved gas in water (DISGASW) and vaporized water in gas (VAPWAT) options in the RUNSPEC section.

```
-- ================================================================================
--
-- RUNSPEC SECTION
--
-- ================================================================================
RUNSPEC
-- --------------------------------------------------------------------------------
-- FLUID TYPES AND TRACER OPTIONS
-- --------------------------------------------------------------------------------
--
--       ACTIVATE CO2 STORAGE IN THE MODEL (OPM FLOW CO2 STORAGE KEYWORD)
--
CO2STORE
--
--       ACTIVATE GAS-WATER THE MODEL (OPM FLOW KEYWORD)
--
GASWAT
--
--       DISSOLVED GAS IN WATER IS PRESENT IN THE RUN (OPM FLOW KEYWORD)
--
DISGASW
--
--       VAPORIZED WATER IN DRY/WET GAS IS PRESENT IN THE RUN (OPM FLOW KEYWORD)
--
VAPWAT
```

The second part of the example covers the data required for the PROPS section, in which the two-phase relative permeability functions are set using GSF and WSF keywords.

```
-- ================================================================================
--
-- PROPS SECTION
--
-- ================================================================================
PROPS
--
--       RESERVOIR
--       TEMPERATURE
--       -----------
RTEMP
         80.0                                              / RESERVOIR TEMP
--
--       ROCK COMPRESSIBILITY
--
--       REFERENCE PRESSURE IS TAKEN FROM THE HCPV WEIGHTED RESERVOIR PRESSURE
--
--       REF PRES  CF
--       BARSA     1/BARSA
--       --------  --------
ROCK
         200.0     5.0E-05                                 / ROCK COMPRESSIBILITY
--
--       GAS RELATIVE PERMEABILITY TABLES (OPM FLOW KEYWORD)
--
GSF
--       SGAS       KRG        PCGW
--       FRAC                  PSIA
```

```
--        --------  --------   -------
             0.000     0.000       0.0
             0.080     0.001       0.0
             0.170     0.010       0.0
             0.350     0.050       0.0
             0.530     0.200       0.0
             0.620     0.350       0.0
             0.650     0.390       0.0
             0.710     0.560       0.0
             0.800     1.000       0.0   / TABLE NO. 01
--
--        WATER RELATIVE PERMEABILITY TABLES (OPM FLOW KEYWORD)
--
WSF
--        SWAT       KRW
--        FRAC
--        --------   --------
             0.200   0.0000
             0.400   0.1000
             0.800   0.5000
             1.000   1.0000             / TABLE NO. 01
--
--        SET SALINITY FOR ALL CELLS (OPM FLOW KEYWORD)
--
SALINITY
0.7                                                    /

-- ======================================================================
```

No other data is required to define the fluid and rock properties in the PROPS section as the data is generated from internal analytic correlations and models by the simulator. Finally, note that units for salinity are to the 10^{-3}, thus for metric units we have 10^{-3} x kg-M/kg.

The third part of the example covers initializing the model in the SOLUTION section. Here we set the EQUIL(EQLOPT6) parameter equal to one, to use table number one of the RVWVD keyword, in order to set the vaporized water versus depth distribution for the model.

```
-- ======================================================================
--
-- SOLUTION SECTION
--
-- ======================================================================
SOLUTION
--
--        SYSTEM IS SATURATED WITH WATER
--
--        DATUM   DATUM   OWC     PCOW   GOC     PCGO   RS   RV   N    E300  RVW
--        DEPTH   PRESS   DEPTH   ----   DEPTH   ----   OPT  OPT  OPT  OPT   OPT
EQUIL
          2000.0  200.0   1800.0  0.00   1800.0  0.00   1*   1*   1*   2*    1   /
--
--        WATER VAPOR RATIO VS DEPTH (OPM FLOW KEYWORD)
--
--        DEPTH     RVW
--                  STB/MSCF
--        ------    --------
RVWVD
          1000.0    0.000
          3000.0    0.000                                 / RVW VS DEPTH EQUIL REGN 01
```

For the SUMMARY section, the simulator supports, several summary vectors specific to CO_2 storage, as shown below.

```
-- ================================================================================
--
-- SUMMARY SECTION
--
-- ================================================================================
SUMMARY
--
--       EXPORT STANDARD SUMMARY VARIABLE VECTORS TO FILE
--
ALL
--
--       FIELD C02 DISOLVED IN WATER PHASE
FWCD
--
--       FIELD C02 TRAPPED IN GAS PHASE
FGCDI
--
--       FIELD C02 MOBILE IN GAS PHASE
FGCDM
```

The final part of the example covers the SCHEDULE section. The standard WCONINJE keyword is used to set the gas injection rate, in this case 100,000 sm3/day of CO_2.

```
-- ================================================================================
--
-- SCHEDULE SECTION
--
-- ================================================================================
SCHEDULE
--
--       RESTART CONTROL BASIC = 4 (ALL=2, YEARLY=4, MONTHLY=5, TSTEP=6)
--
RPTRST
        'BASIC=1' 'ALLPROPS'                                                    /
--
--       WELL SPECIFICATION DATA
--
-- WELL  GROUP     LOCATION  BHP    PHASE  DRAIN  INFLOW  OPEN  CROSS  PVT
-- NAME  NAME        I    J  DEPTH  FLUID  AREA   EQUANS  SHUT  FLOW   TABLE
WELSPECS
INJ1     'G1'       25   25  1*     GAS                                     /
/
--
--       WELL CONNECTION DATA
--
-- WELL  --- LOCATION ---  OPEN   SAT   CONN   WELL   KH    SKIN   D     DIR
-- NAME   II  JJ  K1  K2   SHUT   TAB   FACT   DIA    FACT  FACT   FACT  PEN
COMPDAT
INJ1      25  25   1  10  OPEN    1*    1*    0.3                           /
/
--
--       WELL INJECTION CONTROLS
--
-- WELL  FLUID  OPEN/  CNTL  SURF   RESV   BHP   THP   VFP
-- NAME  TYPE   SHUT   MODE  RATE   RATE   PRES  PRES  TABLE
WCONINJE
INJ1     GAS    OPEN   RATE 100000  1*     300                             /
/
--
--       ADVANCE SIMULATION BY REPORTING TIME
--
TSTEP
         1   36*30
```

```
/
```

Note in order to get the liquid phase mole fractions of CO_2, that is, the mole factions of CO_2 in the water phase (XMFCO2), and the vapor phase mole factions (YMFWAT) to the restart file, one must use the command line parameter enable-opm-rst-file set equal to true.

The second example shows how to use CO2STORE with the alternative option (2). The example below declares that the carbon dioxide storage model is active for the run to account for both carbon dioxide and water phase solubility using OPM Flow's CO_2-Brine PVT model. Option (2) is used where the OIL phase refers to the brine and the GAS phase to CO_2.

```
-- ======================================================================
--
-- RUNSPEC SECTION
--
-- ======================================================================
RUNSPEC
-- ----------------------------------------------------------------------
-- FLUID TYPES AND TRACER OPTIONS
-- ----------------------------------------------------------------------
--
--       ACTIVATE CO2 STORAGE IN THE MODEL (OPM FLOW CO2 STORAGE KEYWORD)
--
CO2STORE
--
--       OIL PHASE IS PRESENT IN THE RUN BUT IS THE BRINE PHASE FOR CO2STORE
--
OIL
--
--       GAS PHASE IS PRESENT IN THE RUN BUT IS THE CO2 PHASE FOR CO2STORE
--
GAS
--
--       DISSOLVED GAS IN LIVE OIL IS PRESENT IN THE RUN (CO2 IN BRINE)
--
DISGAS
--
--       VAPORIZED OIL IN WET GAS IS PRESENT IN THE RUN (WATER IN CO2)
--
VAPOIL
```

The second part of the example covers the data required for the PROPS section, in which the input keywords need to be consistent with the OIL phase referring to the Brine and the GAS to CO_2; that is SGOF (gas-oil relative permeability) is used to define the CO_2-Brine relative permeability table.

```
-- ======================================================================
--
-- PROPS SECTION
--
-- ======================================================================
PROPS
--
--       RESERVOIR
--       TEMPERATURE
--       -----------
RTEMP
90.0                                          / RESERVOIR TEMPERATURE
--
--       ROCK COMPRESSIBILITY
--
--       REFERENCE PRESSURE IS TAKEN FROM THE HCPV WEIGHTED RESERVOIR PRESSURE
```

```
--
--       REF PRES  CF
--       BARSA     1/BARSA
--       --------  --------
ROCK
         1.0000    1.0E-06                                  / ROCK COMPRESSIBILITY
--
--       GAS-OIL RELATIVE PERMEABILITY TABLES (SGOF) - CO2STORE PHASES
SGOF
--       SG         KRG       KROG      PCOG
--       FRAC                           PSIA
--       -------   --------  -------   -------
         0.00000   0.000000  1.00000    0.0000
         1.00000   1.000000  0.00000    0.0000             / TABLE No. 01
--
--       SET SALINITY FOR ALL CELLS (OPM FLOW KEYWORD)
--
SALINITY
0.7                                                       /
-- ======================================================================================
```

The third part of the example covers initializing the model in the SOLUTION section. Here we set the EQUIL(EQLOPT1 and EQLOPT2) parameters equal to one, to use table number one of the RSVD and RVVD keywords, in order to set the initial dissolved CO_2 and vaporized water versus depth distribution for the model.

```
-- ======================================================================================
--
-- SOLUTION SECTION
--
-- ======================================================================================
SOLUTION
--
--       SYSTEM IS SATURATED WITH OIL (BRINE)
--
--       DATUM   DATUM   OWC     PCOW   GOC     PCGO   RS   RV   N    E300  RVW
--       DEPTH   PRESS   DEPTH   ----   DEPTH   ----   OPT  OPT  OPT  OPT   OPT
EQUIL
         2000.0  200.0   2200.0  0.00  1800.0  0.00   1    1    1*   2*    1*   /
--
--       DEPTH    RS
--                MSCF/STB
--       ------   --------
RSVD
         1000.0   0.000
         3000.0   0.000                                    /
--
--       DEPTH    RVW
--                STB/MSCF
--       ------   --------
RVVD
         1000.0   0.000
         3000.0   0.000                                    /
```

The forth and final part of the example sets the maximum dissolution rate for convective CO_2 mixing via the DRSDTCON keyword in the SCHEDULE section.

```
-- ======================================================================================
--
-- SCHEDULE SECTION
--
-- ======================================================================================
```

```
SCHEDULE
--
--       CO2 CONVECTIVE DISSOLUTION PARAMETER
--
DRSDTCON
--       CO2 CONV
--       DISSOLN
--       --------
         0.04                                              /
```

See the DRSDTCON keyword in the SCHEDULE section for further information on this keyword.

5.3.17 COAL – ACTIVATE THE COAL PHASE (CBM MODEL)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The COAL keyword actives the coal phase and the Coal Bed Methane ("CBM") model for the run.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

There is no data required for this keyword and there is no terminating "/" for this keyword.

Example

```
--
--       ACTIVATE THE COAL PHASE (CBM MODEL) IN THE MODEL
--
COAL
```

The above example declares that the Coal phase is active in the run and activates the CBM model option.

5.3.18 COLUMNS – DEFINE INPUT FILE COLUMN MARGINS

The COLUMNS keyword defines the input file column margins; characters outside the margins are ignored by the input parser.

See *COLUMNS – Define Input File Column Margins* in the GLOBAL section for a full description.

5.3.19 COMPS – ACTIVATE COMPOSITIONAL MODELING FORMULATION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

Turns on the Compositional Modeling Formulation, and declares the maximum number of components in the model. The keyword should only be used if the CO2STORE keyword and either the GASWAT or the GAS and WATER keywords in the RUNSPEC section, have also be activated for the gas-water two component model.

Note

This is an OPM Flow keyword used with OPM Flow's CO2STORE and GASWAT keywords in the RUNSPEC section, and should not be confused with the more general version of the COMPS keyword used in the commercial compositional simulator.

Secondly, although OPM Flow parses the keyword, the simulator currently ignores the data for this keyword.

No.	Name	Description	Default
1	COMPS	A positive integer defining the maximum number of compositional components in the model. Only the default value of two is currently supported by OPM Flow.	2

Notes:

1) The keyword is terminated by a "/".

Table 5.7: COMPS Keyword Description

Example

The following example defines how to define a two component formulation to be used with the CO2STORE and GASWAT options.

```
-- ======================================================================
--
-- RUNSPEC SECTION
--
-- ======================================================================
RUNSPEC

--
--       ACTIVATE CO2 STORAGE IN THE MODEL (OPM FLOW CO2 STORAGE KEYWORD)
--
CO2STORE
--
--       ACTIVATE COMPOSITIONAL MODELING FORMULATION (OPM FLOW KEYWORD)
--
COMPS
         2                                                              /
--
--       ACTIVATE GAS-WATER THE MODEL (OPM FLOW KEYWORD)
--
GASWAT
```

5.3.20 CPR – ACTIVATE CONSTRAINED PRESSURE RESIDUAL ("CPR") LINEAR SOLVER

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

Turns on the Constrained Pressure Residual ("CPR")[66], [67] and [68] preconditioner linear solver option, and declares how the solver should be applied. The keyword is equivalent to using the OPM Flow command line parameter --linear-solver= "cprw". Note that if the command line has been used, then this will take precedence over the CPR keyword.

No.	Name	Description	Default
1	CPROPTN	A defined character string that determines how the CPR linear solver should be applied, and should be set to one of the following: 1) ORIGINAL: Here the solver is applied for the whole of the simulation. 2) ADAPTIVE: This option applies the more computational demanding CPR linear solver, only in parts of the run that would benefit from its use, for example when linear convergence is challenging. Note that OPM Flow only supports the ORIGINAL option, which is the default value in OPM Flow, whereas the default value in the commercial simulator is ADAPTIVE. .	ORIGINAL
Notes: 1) The keyword is terminated by a "/".			

Table 5.8: CPR Keyword Description

See section *2.2 Running OPM Flow 2023-10 From The Command Line* on how to invoke various numerical schemes via the OPM Flow command line interface.

Example

```
--
--       ACTIVATE CONSTRAINED PRESSURE RESIDUAL LINEAR SOLVER FOR THE RUN
--
CPR
/
```

The above example activates the Constrained Pressure Residual preconditioner linear solver using the default option, that is the ORIGINAL option for OPM Flow.

To enable the CPR preconditioner linear solver option using the command line parameter, use:

```
flow --linear-solver="cpr" CASENAME.DATA
```

[66] *Wallis, J. R., Little, T. E., and Nolen, J. S.: "Constrained Residual Acceleration of Conjugate Residual Methods," paper SPE 13536 presented at the SPE Reservoir Simulation Symposium, Dallas, Texas, USA (February 10-13, 1985).*

[67] *R. Scheichl, M. Roland, J. Wendebourg, Decoupling and block preconditioning for sedimentary basin simulations, Computational Geosciences 7 (2003) 295{318.*

[68] *Klemetsdal, Ø.S., Møyner, O. & Lie, KA. Accelerating multiscale simulation of complex geomodels by use of dynamically adapted basis functions. Comput Geosci 24, 459–476 (2020). https://doi.org/10.1007/s10596-019-9827-z.*

which should usually be combined with the -matrix-add-well-contributions=true option, that is:

```
flow --linear-solver=cpr --matrix-add-well-contributions=true CASENAME.DATA
```

However, the new CPRW preconditioner, that includes wells does not need the latter option, so in this instance use:

```
flow --linear-solver=cprw CASENAME.DATA
```

As of this release the CPRW preconditioner, should be considered experimental, and the recommended configuration for running this is:

```
flow --linear-solver=cprw --linear-solver-reduction=0.005 --cpr-reuse-setup=4
     --cpr-reuse-interval=10 CASENAME.DATA
```

See new Rasmussen et al.[69] for further information on the available numerical algorithms available in OPM Flow.

[69] *Atgeirr Flø Rasmussen, Tor Harald Sandve, Kai Bao, Andreas Lauser, Joakim Hove, Bård Skaflestad, Robert Klöfkorn, Markus Blatt, Alf Birger Rustad, Ove Sævareid, Knut-Andreas Lie, Andreas Thune, The Open Porous Media Flow reservoir simulator, Computers & Mathematics with Applications, Volume 81, 2021, Pages 159-185, ISSN 0898-1221,* https://doi.org/10.1016/j.camwa.2020.05.014. *(https://www.sciencedirect.com/science/article/pii/S0898122120302182).*

5.3.21 DEBUG – DEFINE THE DEBUG DATA TO BE PRINTED TO FILE

This keyword defines the debug data to be written to the debug file (*.DBG). This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

See *DEBUG – Define the Debug Data to be Printed to File* in the GLOBAL section for a full description.

5.3.22 DIFFUSE – ACTIVATE MOLECULAR DIFFUSION OPTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The DIFFUSE keyword activates OPM Flow's Molecular Diffusion option based on fluid and grid data (Sandve et al.[70]), similar to the commercial simulator. For field-scale simulations diffusion is a sub-grid phenomenon and is typically not explicitly represented in the flow equations. However, for simulations on the laboratory scale diffusion plays a direct role and therefore needs to be explicitly represented in the flow equations. Diffusion coefficients that control the diffusion depend on temperature, pressure, and salinity. In OPM Flow, the diffusion coefficients are computed internally for pure water using McLachlan and Danckwerts[71] and modified to account for salinity using Ratcliff and Holdcroft[72]. The effect of the porous media on the diffusion is modeled using the relation suggested by Millington and Quirk[73]. The coefficients can also be given for each PVT region as an input parameter using the DIFFC keyword in the PROPS section.

There is no data required for this keyword and there is no terminating "/" for this keyword.

Note

The option has been tested in combination with the CO2STORE keyword, but not for the general case at this point.

Example

```
--
--       ACTIVATE MOLECULAR DIFFUSION OPTION
--
DIFFUSE
```

The above example switches on the molecular diffusion facility.

[70] *Tor Harald Sandve1, Sarah E. Gasda, Atgeirr Rasmussen, and Alf Birger Rustad. Convective dissolution in field scale CO_2 storage simulation using the OPM Flow simulator. Submitted to TCCS 11 – Trondheim Conference on CO_2 Capture, Transport and Storage Trondheim, Norway – June 21-23, 2021.*

[71] *McLachlan, C. N. S., & Danckwerts, P.V. (1972). Desorption of carbon dioxide from aqueous potash solutions with and without the addition of arsenite as a catalyst. Trans. Inst. Chem. Eng, 50, 300-309.*

[72] *Ratcliff, G.A., & Holdcroft, J. G. (1963). Diffusivities of gases in aqueous electrolyte solutions. Trans. Inst. Chem. Eng, 41(10), 315-319.*

[73] *Millington, R. J., & Quirk, J. P. (1961). Permeability of porous solids. Transactions of the Faraday Society, 57, 1200-1207*

5.3.23 DIMENS – DEFINE THE DIMENSIONS OF THE MODEL

Description

DIMENS defines the dimensions of the model entered as integer vector. The keyword can be used with all grid types.

No.	Name	Description	Default
1	NX	A positive integer value that defines the number of grid blocks in the x direction for Cartesian grids or the number of grid blocks in the r direction for radial grids	None
2	NY	A positive integer value that defines the number of grid blocks in the y direction for Cartesian grids or the number of grid blocks in the theta direction for radial grids.	None
3	NZ	A positive integer value that defines the number of grid blocks in the z direction for both Cartesian and radial grids.	None
Notes: 1) The keyword is terminated by a "/".			

Table 5.9: DIMENS Keyword Description

Note that NX, NY and NZ are not maximum values but the actual size of the grid. OPM Flow applies these parameters when reading in particular data sets. For example if NX, NY, and NZ are set to 10, 10 and 10 respectively, then for the grid property data like PORO. OPM Flow expects to read in 10 x 10 x 10 or 1,000 porosity values for the PORO array. If the number of porosity values is not equal to 1,000 then OPM Flow will produce an error.

Example

```
--
--        MAX      MAX      MAX
--        NDIVIX   NDIVIY   NDIVIZ
DIMENS
          46       112      22                                          /
```

The above example defines the dimensions for the Norne model of 46 cells in the x direction, 112 cells in the y direction and 22 cells in the z direction.

5.3.24 DISGAS – ACTIVATE THE DISSOLVED GAS PHASE IN THE MODEL

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword indicates that dissolved gas is present in live[74] oil in the model and the keyword should only be used if the there is both oil and gas phases in the model. The keyword may be used for oil-water and oil-water-gas input decks that contain the oil and gas phases. The keyword will also invoke data input file checking to ensure that all the required oil and gas phase input parameters are defined in the input deck.

If the oil has a constant and uniform dissolved gas concentration, Gas-Oil Ratio ("GOR"), and if the reservoir pressure never drops below the saturation pressure (bubble point pressure), then the model can be run more efficiently by omitting the GAS and DISGAS keywords from the RUNSPEC section, treating the oil as a dead oil[75], and defining a constant Rs (GOR) value with keyword RSCONST or RSCONSTT in the PROPS section. This results in the model being run with as a dead oil problem with no active gas phase. However, OPM Flow takes into account the constant Rs in the calculations and reporting.

There is no data required for this keyword and there is no terminating "/" for this keyword.

Example

```
--
--       DISSOLVED GAS IN LIVE OIL IS PRESENT IN THE RUN
--
DISGAS
```

The above example declares that the dissolved gas in the oil phase is active in the model.

[74] *"Live" oil is oil that contains gas in solution, which is normally the case for most conventional oil reservoirs. However, for oil reservoirs classified as heavy oil reservoirs, the in situ dissolved gas may be negligible and oil would then be classified as gas-free oil which is commonly referred to as "dead" oil.*

[75] *"Dead" oil is oil that it contains no dissolved gas or a relatively thick oil or residue that has lost its volatile components.*

5.3.25 DISGASW – ACTIVATE DISSOLVED GAS IN THE WATER PHASE IN THE MODEL

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword indicates that dissolved gas is present in the water phase in the model. The keyword may only be used for gas-water input decks that contain just the gas and the water phases, which must also be declared. The keyword will also invoke data input file checking to ensure that all the required gas and water phase input parameters are defined in the input deck.

Note

This is an OPM Flow specific keyword for the simulator's Dissolved Gas in Water Model that is activated by declaring that this phase is present in the run.

The activation of this phase may be used for modeling the production of gas from the water in gas fields, where gas has dissolved into the in situ water phase. Although this is always the case, the volumes are usually insignificant compared with the volumes in the hydrocarbon zones, and therefore in nearly all cases the dissolved gas volumes are usually ignored in terms of resource volumes. For gas fields, the interest is normally the amount of water in the gas phase, as this influences the liquid loading rate as the water is liberated from the gas as the fluid transverses up the wellbore.

The second, and primary, application of this black-oil formulation is for CO_2 storage, where injection of CO_2 injection is being conducted in either depleted gas fields or in water filled aquifer zones. Activating the dissolved gas in the water phase under these circumstances will allow the CO_2 to dissolve into the water phase based on the entered PVT properties in the PROPS section. In this case the CO2STORE keyword in the RUNSPEC section should also be present in the deck.

There is no data required for this keyword and there is no terminating "/" for this keyword.

Example

```
--
--        GAS PHASE IS PRESENT IN THE RUN
--
GAS
--
--        WATER PHASE IS PRESENT IN THE RUN
--
WATER
--
--        DISSOLVED GAS IN WATER IS PRESENT IN THE RUN (OPM FLOW KEYWORD)
--
DISGASW
--
--        ACTIVATE CO2 STORAGE IN THE MODEL (OPM FLOW CO2 STORAGE KEYWORD)
--
CO2STORE
```

The above example declares that the gas and water phases are present, and that the dissolved gas in the water phase is also active in the model, for modeling CO2 storage.

5.3.26 DISPDIMS – DEFINE THE MAXIMUM NUMBER OF DISPERSION TABLES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The DISPDIMS key defines the maximum number of dispersion tables, and the maximum number of velocity and concentration elements per table.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

5.3.27 DUALPERM – ACTIVATE DUAL PERMEABILITY MODEL

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The DUALPERM keyword activates the Dual Permeability option and the Dual Porosity option for the run. In a dual porosity system flow occurs between the matrix and the fracture only, whereas in a dual permeability system flow also occurs between the matrix grid blocks[76], [77], [78], [79] and [80].

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

There is no data required for this keyword and there is no terminating "/" for this keyword.

Example

```
--
--       ACTIVATE DUAL PERMEABILITY MODEL
--
DUALPERM
```

The above example declares that the Dual Permeability option is active for the run.

[76] *Warren, J.E. and Root, P.J. 1963. The Behavior of Naturally Fractured Reservoirs. SPE J. 3 (3): 245–255. SPE-426-PA. http://dx.doi.org/10.2118/426-PA.*

[77] *Gringarten, A.C. 1984. Interpretation of Tests in Fissured and Multilayered Reservoirs With Double-Porosity Behavior: Theory and Practice. J Pet Technol 36 (4): 549-564. SPE-10044-PA. http://dx.doi.org/10.2118/10044-PA.*

[78] *Serra, K., Reynolds, A.C., and Raghavan, R. 1983. New Pressure Transient Analysis Methods for Naturally Fractured Reservoirs(includes associated papers 12940 and 13014). J Pet Technol 35 (12): 2271-2283. SPE-10780-PA. http://dx.doi.org/10.2118/10780-PA*

[79] *Barenblatt, G.E., Zheltov, I.P., and Kochina, I.N. 1960. Basic Concepts in the Theory of Homogeneous Liquids in Fissured Rocks. J. Appl. Math. Mech. 24: 1286-1303.*

[80] *Kazemi, H., Merrill JR., L. S., Porterfield, K. L., and Zeman, P. R. "Numerical Simulation of Water-Oil Flow in Naturally Fractured Reservoirs," paper SPE 5719, Society of Petroleum Engineers Journal(1976) 16, No. 6, 317-326.*

5.3.28 DUALPORO – ACTIVATE DUAL POROSITY MODEL

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The DUALPORO keyword activates the Dual Porosity option for the run. In a dual porosity system flow occurs between the matrix and the fracture only, whereas in a dual permeability system flow also occurs between the matrix grid blocks[81], [82], [83], [84] and [85].

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

There is no data required for this keyword and there is no terminating "/" for this keyword.

Example

```
--
--       ACTIVATE DUAL POROSITY MODEL
--
DUALPORO
```

The above example declares that the Dual Porosity option is active for the run.

[81] *Warren, J.E. and Root, P.J. 1963. The Behavior of Naturally Fractured Reservoirs. SPE J. 3 (3): 245–255. SPE-426-PA. http://dx.doi.org/10.2118/426-PA.*

[82] *Gringarten, A.C. 1984. Interpretation of Tests in Fissured and Multilayered Reservoirs With Double-Porosity Behavior: Theory and Practice. J Pet Technol 36 (4): 549-564. SPE-10044-PA. http://dx.doi.org/10.2118/10044-PA.*

[83] *Serra, K., Reynolds, A.C., and Raghavan, R. 1983. New Pressure Transient Analysis Methods for Naturally Fractured Reservoirs(includes associated papers 12940 and 13014). J Pet Technol 35 (12): 2271-2283. SPE-10780-PA. http://dx.doi.org/10.2118/10780-PA*

[84] *Barenblatt, G.E., Zheltov, I.P., and Kochina, I.N. 1960. Basic Concepts in the Theory of Homogeneous Liquids in Fissured Rocks. J. Appl. Math. Mech. 24: 1286-1303.*

[85] *Kazemi, H., Merrill JR., L. S., Porterfield, K. L., and Zeman, P. R. "Numerical Simulation of Water-Oil Flow in Naturally Fractured Reservoirs," paper SPE 5719, Society of Petroleum Engineers Journal(1976) 16, No. 6, 317-326.*

5.3.29 DYNRDIMS – DEFINE DYNAMIC REGION DIMENSIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The DYNRDIMS keyword defines the dimensions for the parameters used by the Dynamic Regions facility, including the maximum number of dynamic regions. The Dynamic Regions facility allows for property and reporting regions to vary as the run progresses, based on the parameters and logic defined by the DYNAMICR keyword in the SOLUTION and PROPS section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

5.3.30 ECHO – ACTIVATE ECHOING OF USER INPUT FILES TO THE PRINT FILE

Turns on echoing of all the input files to the print file; note that this keyword is activated by default and can subsequently be switched off by the NOECHO activation keyword.

See *ECHO – Activate Echoing of User Input Files to the Print File* in the GLOBAL section for a full description.

5.3.31 ECLMC – ACTIVATE MULTI-COMPONENT BRINE MODEL

Description

The ECLM keyword activates the Multi-Component Brine model that allows for the water phase to have multiple water salinities. The keyword should be used in conjunction with the BRINE keyword in the RUNSPEC. Both keywords must be specified to activate the Multi-Component Brine model, whereas the BRINE keyword only is required to activate the standard brine tracking model.

There is no data required for this keyword and there is no terminating "/" for this keyword.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

Example

The first example activates the standard Brine Tracking model.

```
--
--       ACTIVATE STANDARD BRINE MODEL
--
BRINE
```

The next example shows the ECLMC and BRINE keywords for when the Multi-Component Brine model is required.

```
--
--       ACTIVATE MULTI-COMPONENT BRINE MODEL
--
ECLMC
--
--       DEFINE WATER PHASE MULTI-COMPONENT BRINE COMPONENTS
--
--       SALT1   SALT2   SALT3   SALT4   SALT5
BRINE
         NACL    CACL2   MGC03                                              /
```

The above example activates the Multi-Component Brine model with three different water salinities.

5.3.32 END – DEFINE THE END OF THE INPUT FILE

This keyword marks the end of the input file and can occur in any section. Any keywords and data after the END keyword are ignored.

See *END – Define the End of the Input File* in the GLOBAL section for a full description.

5.3.33 ENDINC – DEFINE THE END OF AN INCLUDE FILE

This keyword marks the end of an include file specified on the INCLUDE keyword. When the ENDINC keyword is encountered in the INCLUDE file, input data is read from the next keyword in the current file. Any keywords and data after the ENDINC keyword in the INCLUDE file are ignored.

See *ENDINC – Define the End of an Include File* in the GLOBAL section for a full description.

5.3.34 ENDSCALE – ACTIVATE RELATIVE PERMEABILITY END-POINT SCALING OPTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The ENDSCALE keyword activates OPM Flow's relative permeability end-point scaling option. The relative permeability functions are defined using the either the:

1) SWOF, SGOF, SLGOF series of saturation functions, or the
2) SWFN, SGFN, SGWFN, SOF2, SOF3, SOF32D series of functions.

And are allocated to the grid cells via the SATNUM keyword.

End-point scaling allows the entered relative permeability functions to be re-scaled based on the saturation end-points allocated to each cell by the SWL, SWCR, SWU, SGL, SGCR, SGU, SOWCR, and SOGCR saturation grid arrays for the saturation end-points, and the KRG, KROG, KROW and KRW relative permeability grid cell arrays for the relative permeability end-point data. In addition end-point scaling may be directional dependent in which case the directional dependent versions of the aforementioned arrays should be used, that is SWLX, SWLY and SWLZ instead of SWL etc. There is also the facility to make the directional end-point scaling reversible or non-reversible and if the non-reversible option is selected the non-reversible versions of the aforementioned arrays should be used, that is SWLX, SWLX-, SWLY, SWLY- SWLZ and SWLZ-, instead of SWL or the SWLX, SWLY and SWLZ set of keywords.

The keyword also defines the number of saturation end-point tables that allows for the re-scaling of the saturation functions to be a function of depth as opposed to being a grid property array. This is accomplished via the ENKRVD and ENPTVD keywords in the PROPS section.

No.	Name	Description	Default
1	DIRECT	A character string that activates or deactivates directional end-point scaling option. If DIRECT is set to NODIR then directional end-point scaling is switched off and the same saturation function is used in the x, y and z directions (unless activated otherwise by the SATOPTS keyword in the RUNSPEC section). In this case the SWL, SWCR, SWU, SGL, SGCR, SGU, SOWCR and SOGCR saturation grid arrays and the KRG, KRORG, KRORW and KRW relative permeability grid cell arrays should be used to enter the grid block end-point data. If DIRECT is set to DIRECT then directional end-point scaling is switched on and different saturation functions are used in the x, y and z directions (unless activated otherwise by the SATOPTS keyword in the RUNSPEC section). Here the directional form of the SWL, SWCR, SWU, SGL, SGCR, SGU, SOWCR and SOGCR saturation grid arrays and the KRG, KRORG, KRORW and KRW relative permeability grid cell arrays should be use to enter the grid block end-point data. For example SWLX, SWLY and SWLZ for SWL. Only the default option is supported by OPM Flow.	NODIR

No.	Name	Description	Default
2	IRREVERS	A character string that activates or deactivates non-reversible end-point scaling option. If IRREVERS is set to REVERS then the end-point scaling is set to reversible and results in the same set of end-point arrays being used for flow from the x_l to x_{l+1} direction as for the flow from the x_l to the x_{l-1} for all directions (x, y and z). Here the SWLX, SWLY and SWLZ series of keywords should be used instead of SWL type of keywords. Alternatively, if IRREVERS is set to IRREVERS then the end-point scaling is set to non-reversible and results in different sets of end-point arrays being applied for flow from the x_l to x_{l+1} direction and the x_l to the x_{l-1} direction, for all directions (x, y, z). in this case the SWLX+, SWLX-, SWLY+, SWLY- SWLZ+ and SWLZ- series of keywords should be utilized instead of SWL or the SWLX, SWLY and SWLZ set of keywords. Only the default option is supported by OPM Flow.	REVERS
3	NTENDP	A positive integer that defines the maximum number of saturation end-point depth tables. The end-point depth tables are used to re-scale the saturation tables as a function of depth as opposed to being a grid block property. NTENDP may also be specified on the TABDIMS keyword, and if specified on both here and on the TABDIMS keyword the maximum value of the two is used. Only the default option is supported by OPM Flow.	1
4	NNODES	A positive integer the defines the maximum number entries for saturation end-point depth tables.	20
5	MODE	A positive integer that activates the options for temperature dependent saturation end-point scaling in the commercial compositional simulator. MODE should be defaulted with either 1* or zero, which means that scaling can only be performed by grid block end-point scaling properties or via saturation end-point depth tables.	0

Notes:

1) Note that the IRREVERS option can only be set to IRREVERS if the DIRECT parameter is set equal to DIRECT.
2) The keyword is terminated by a "/".

Table 5.10: ENDSCALE Keyword Description

Example

```
--        DIRC    REVERSE  MAX      MAX
--        SCALE   SCALE    TABLES   NODES
ENDSCALE
          NODIR   REVERS   1*       1*                                    /
```

The above example invokes the end-point scaling option with end-point scaling being non-directional and reversible with the default number of saturation end-point depth tables (one) with 20 entries per table.

5.3.35 ENDSKIP – DEACTIVATE SKIPPING OF KEYWORDS AND INPUT DATA

Turns off skipping of keywords that was activated by the SKIP, SKIP100, or SKIP300 keywords. Note that for each SKIP keyword activated there must be a corresponding ENDSKIP keyword.

See *ENDSKIP – DeActivate Skipping of Keywords and Input Data* in the GLOBAL section for a full description.

5.3.36 EQLDIMS – DEFINE THE EQUILIBRATION DATA DIMENSIONS

Description

The EQLDIMS keyword defines the maximum number of properties associated with equilibrating the model, that is initializing the model. A reservoir grid can be separated into separate regions in order to apply different pressure regimes and/or fluid contacts. Care should be taken that the different regions are not in communication if the pressures or fluid contacts are different for the various regions, as this would lead to an unstable initialization and would also imply errors in the model description as implemented.

No.	Name	Description	Default
1	NTEQUL	A positive integer value that defines the number of equilibration regions entered using the EQLNUM keyword in the REGIONS section and the number of entries associated with the EQUIL keyword in the SOLUTION section.	1
2	NPRSVD	A positive integer value setting the number of pressure versus depth entries used by OPM Flow in determining equilibration parameters. Unless there is a requirement for a very fine equilibration this parameter should be defaulted.	100
3	NDRXVD	A positive integer value that defines the maximum number of depth entries in equilibration property versus depth tables (RSVD, RVVD, PBVD or PDVD etc.) as defined in the SOLUTION section.	20
4	NTTRVD	A positive integer that defines the maximum number of TVDP tables that describe the initial tracer concentration versus depth.	1
5	NSTRVD	A positive integer that defines the maximum number of depth entries in the TVDP tables as described in (4)	20

Notes:

1) NTEQUL is the exact number of entries must be entered on the EQUIL keyword, otherwise OPM Flow will report an error. It is not a maximum number as for the other region arrays.

2) The keyword is terminated by a "/".

Table 5.11: EQLDIMS Keyword Description

It is common that the EQLNUM and FIPNUM arrays are identical so that the fluid in-place reporting matches the equilibration regions. Thus, in order to avoid errors in this case, one should just use one array (say the FIPNUM property array) and use the COPY keyword to generate the EQLNUM array.

Example

```
--
--        MAX     MAX     RSVD    TVDP    TVDP
--        EQLNUM  DEPTH   NODES   TABLE   NODES
EQLDIMS
          9       1*      20      1*      1*                    /
```

The above example defines nine equilibration regions the default values for the remaining parameters on the EQLDIMS keyword.

5.3.37 EQLOPTS – ACTIVATE THE EQUILIBRATION OPTIONS

Description

The EQLOPTS keyword defines the equilibration options by stating the character command to activate an option to be used for initializing the model. Multiple commands may be utilized to activate several equilibration options following the keyword.

No.	Name	Description	Default
1	MOBILE	A character string that activates the mobile fluid critical saturation end point correction. If the MOBILE command is stated then this option is activated. This option is not supported by OPM Flow but would change the results if supported so the simulation will be stopped. Therefore this character string should be omitted.	None
2	QUIESC	A character string that activates the initial quiescence option that modifies the equilibrium calculated phase pressures to ensure that a steady state solution is obtained. This options ensures that there is no flow potential between the grid blocks in a given region, which is the normal case when block-centered equilibration is used by setting BOINIT on the EQUIL keyword to zero in the SOLUTION section. If the QUIESC command is stated then this option is activated. This option is not supported by OPM Flow but would change the results if supported so the simulation will be stopped. Therefore this character string should be omitted.	None
3	THPRES	A character string that activates the inter-region equilibration flow option. This option allows for a threshold pressure variable entered via the THPRES keyword to define a pressure which prevents flow between regions until the THPRES value between regions is exceeded. If the THPRES command is stated then this option is activated.	None
4	IRREVERS	A character string that activates the irreversible inter-region equilibration flow option. This option can only be invoked if the THPRES command has been stated. The option allows for different THPRES values for different directions. If the IRREVERS command is stated then this option is activated. This option is not supported by OPM Flow but would change the results if supported so the simulation will be stopped. Therefore this character string should be omitted.	None

Notes:

1) The keyword is terminated by a "/".

Table 5.12: EQLOPTS Keyword Description

Example

```
--
--       ACTIVATE EQUILIBRATION OPTIONS
--       MOBILE END-POINT(MOBILE) STEADY STATE(QUIESC) THRESHOLD(THPRES)
--       IRREVERSIBLE THRESHOLD(IRREVERS)
EQLOPTS
         'THPRES'   'IRREVERS'                                          /
```

The above example activates the threshold pressure option with different threshold pressure for different directions.

5.3.38 EXTRAPMS – ACTIVATE EXTRAPOLATION WARNING MESSAGES

The EXTRAPMS keyword activates extrapolation warning messages for when OPM Flow extrapolates the PVT or VFP tables. Frequent extrapolation warning messages should be investigated and resolved as this would indicate possible incorrect data and may result in the simulator extrapolating to unrealistic values.

See *EXTRAPMS – Activate Extrapolation Warning Messages* in the GLOBAL section for a full description.

5.3.39 FAULTDIM – DEFINE THE NUMBER OF FAULT SEGMENTS

Description

The FAULTDIM keyword defines the maximum number of records (or segments) that can be entered with the FAULTS keyword. The FAULTS keyword defines the faults in the grid than can be used for setting (or re-setting) transmissibility barriers across the fault planes.

No.	Name	Description	Default
1	MFSEGS	A positive integer value that defines the maximum number of records (segments) for the FAULTS keyword.	0
Notes: 1) The keyword is terminated by a "/".			

Table 5.13: FAULTDIM Keyword Description

Example

```
--
--        FAULT
--        SEGMS
--
FAULTDIM
          10000                                                    /
```

The above example defines the maximum number of records that can be entered using the FAULT keyword to be 10,000 segments.

5.3.40 FIELD – ACTIVATE THE OIL FIELD SYSTEM OF UNITS FOR THE MODEL

Description

This keyword switches on the oil FIELD system of units for the model.

OPM Flow has three sets of units, namely: METRIC, FIELD and LAB and one of these keyword should be invoked in the RUNSPEC section to avoid any ambiguity. Both the simulator input and output units are controlled by including one of the METRIC, FIELD or LAB keywords in the RUNSPEC section of the input file.

There is no data required for this keyword and there is no terminating "/" for this keyword.

Example

```
--
--       SWITCH ON THE FIELD SYSTEM OF UNITS FOR BOTH INPUT AND OUTPUT
--
FIELD
```

The above example switches on the FIELD system of units for the model.

5.3.41 FMTHMD – ACTIVATE THE FORMAT HISTORY MATCH GRADIENT FILE OPTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword switches on formatted output for the History Match Gradient output file, for when the History Match Gradient option has been activated by the HMDIMS keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

5.3.42 FMTIN – ACTIVATE THE FORMAT INPUT FILE OPTION

Description

This keyword switches on the Format Input Files option for all input files. Similar to the commercial simulator, OPM Flow has various options for reading various input files and writing the resulting OPM Flow output files as described in Table 5.14.

Process	Keyword	Description	Files
Input	FMTIN	A character string that defines the input files to be formatted as ASCII i.e. text files, as oppose to binary files. The input deck file is always of this type. The option relates to the OPM Flow derived files that used as input, for for example when restarting from another case. If the keyword is omitted then the default is for binary file input.	*.FEGRID *.FINSPEC *.FINIT *.FRSSPEC *.FUNRST *.FSMSPEC *.FUNSMRY
	MULTIN	A character string that defines the input files to be non-unified multiple files, as opposed to unified files. In this case, one file is read in per reporting time step, as opposed to all time steps reports being read from one file. If the keyword is omitted then the default is for one file per reporting time step.	*.RSSPEC *.X0001 *.SMSPEC *.S0001
	UNIFIN	A character string that defines the input files to be unified files, as opposed to non-unified multiple files. A unified file is a single file containing output for each reporting time step. For this option a single summary file and a single restart file will be read. If the keyword is omitted then the default is for one file per reporting time step.	*.RSSPEC *.UNRST *.SMSPEC *.UNSMRY
Output	FMTOUT	A character string that sets all output files to be formatted as ASCII i.e. text files, as opposed to binary files. The *.PRT, *.LOG and *.DBG files are always of this type. The option relates to the OPM Flow output files only. In this case the files will be portable across operating systems, but will also be very large in terms of hard disk space. For this reason it is recommend that the default option is used so that binary files are outputted. If the keyword is omitted then the default is for binary file input.	*.FEGRID *.FINSPEC *.FINIT *.FRSSPEC *.FUNRST *.FSMSPEC *.FUNSMRY
	MULTOUT	A character string that defines the output files to be non-unified multiple files, as opposed to unified files. In this case, one file is written for each reporting time step, as opposed to all time steps reports being written in one file. If the keyword is omitted then the default is for one file per reporting time step.	*.RSSPEC *.X0001 *.SMSPEC *.S0001

Process	Keyword	Description	Files
	UNIFOUT	A character string that defines the output files to be unified files, as opposed to non-unified multiple files. A unified file is a single file containing output for each reporting time step. Here a single summary file and a single restart file will be generated, as opposed to one file per report time step. If the keyword is omitted then the default is for one file per reporting time step.	*.RSSPEC *.UNRST *.SMSPEC *.UNSMRY

Notes:

1) A binary file is computer-readable but not human-readable. All executable programs are stored in binary files, as are most numeric data files. In contrast, text files are stored in a form (usually ASCII) that is human-readable.
2) For unified files if the run terminates unexpectedly, or there is sufficient disk space, then the last report output is not stored. Their main advantage is that if a number of simulations reside in one directory, their output is organized. There is no limit on the number of reporting steps that a unified file can store.

Table 5.14: FMTIN Keyword Description

There is no data required for this keyword.

See also *OPM FLOW OUTPUT FILE FORMATS* for a more detailed description of the various file types (ASCII or binary) and file structure formats (unified or non-unified formats).

Example

```
--
--       SWITCH ON THE FORMAT INPUT FILES OPTION
--
FMTIN
```

The above example switches on the format input file option.

5.3.43 FMTOUT – ACTIVATE THE FORMAT OUTPUT FILE OPTION

Description

This keyword switches on the Format Output Files option for all output files. Similar to the commercial simulator, OPM Flow has various options for reading various input files and writing the resulting OPM Flow output files as described in Table 5.15.

Process	Keyword	Description	Files
Input	FMTIN	A character string that defines the input files to be formatted as ASCII i.e. text files, as oppose to binary files. The input deck file is always of this type. The option relates to the OPM Flow derived files that used as input, for for example when restarting from another case. If the keyword is omitted then the default is for binary file input.	*.FEGRID *.FINSPEC *.FINIT *.FRSSPEC *.FUNRST *.FSMSPEC *.FUNSMRY
	MULTIN	A character string that defines the input files to be non-unified multiple files, as opposed to unified files. In this case, one file is read in per reporting time step, as opposed to all time steps reports being read from one file. If the keyword is omitted then the default is for one file per reporting time step.	*.RSSPEC *.X0001 *.SMSPEC *.S0001
	UNIFIN	A character string that defines the input files to be unified files, as opposed to non-unified multiple files. A unified file is a single file containing output for each reporting time step. For this option a single summary file and a single restart file will be read. If the keyword is omitted then the default is for one file per reporting time step.	*.RSSPEC *.UNRST *.SMSPEC *.UNSMRY
Output	FMTOUT	A character string that sets all output files to be formatted as ASCII i.e. text files, as opposed to binary files. The *.PRT, *.LOG and *.DBG files are always of this type. The option relates to the OPM Flow output files only. In this case the files will be portable across operating systems, but will also be very large in terms of hard disk space. For this reason it is recommend that the default option is used so that binary files are outputted. If the keyword is omitted then the default is for binary file input.	*.FEGRID *.FINSPEC *.FINIT *.FRSSPEC *.FUNRST *.FSMSPEC *.FUNSMRY
	MULTOUT	A character string that defines the output files to be non-unified multiple files, as opposed to unified files. In this case, one file is written for each reporting time step, as opposed to all time steps reports being written in one file. If the keyword is omitted then the default is for one file per reporting time step.	*.RSSPEC *.X0001 *.SMSPEC *.S0001

Process	Keyword	Description	Files
	UNIFOUT	A character string that defines the output files to be unified files, as opposed to non-unified multiple files. A unified file is a single file containing output for each reporting time step. Here a single summary file and a single restart file will be generated, as opposed to one file per report time step. If the keyword is omitted then the default is for one file per reporting time step.	*.RSSPEC *.UNRST *.SMSPEC *.UNSMRY

Notes:

1) A binary file is computer-readable but not human-readable. All executable programs are stored in binary files, as are most numeric data files. In contrast, text files are stored in a form (usually ASCII) that is human-readable.
2) For unified files if the run terminates unexpectedly, or there is sufficient disk space, then the last report output is not stored. Their main advantage is that if a number of simulations reside in one directory, their output is organized. There is no limit on the number of reporting steps that a unified file can store.

Table 5.15: FMOUT Keyword Description

There is no data required for this keyword.

See also *OPM FLOW OUTPUT FILE FORMATS* for a more detailed description of the various file types (ASCII or binary) and file structure formats (unified or non-unified formats).

Example

```
--
--       SWITCH ON THE FORMAT OUTPUT FILES OPTION
--
FMTOUT
```

The above example switches on the format output file option.

5.3.44 FOAM – ACTIVATE THE FOAM PHASE AND MODEL

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword activates the foam phase and modeling option. The keyword will also invoke data input file checking to ensure that all the required foam phase input parameters are defined in the input deck. Note in the commercial simulator the FOAM phase and model can be used in conjunction with the POLYMER and SURFACT phases; this is not the case for OPM Flow. OPM Flow's FOAM phase and model is a standalone implementation and cannot be used in conjunction with either the POLYMER or SURFACT phases.

Foam flooding is an enhanced oil recovery flood process that attempts to control injected gas breakthrough in an oil reservoir by changing the mobility of the injected fluid.

There is no data required for this keyword and there is no terminating "/" for this keyword.

Example

```
--
--       ACTIVATE THE FOAM PHASE IN THE MODEL
--
FOAM
```

The above example declares that the foam phase is active in the model.

5.3.45 FORMFEED – DEFINED THE PRINT FILE FORM-FEED CHARACTER

The FORMFEED keyword defines the form-feed character, or carriage control character, for the output print (*.PRT) run summary (*.RSM) files. The keyword should be place at the very top of the input file.

See *FORMFEED – Defined the Print File Form-Feed Character* in the GLOBAL section for a dull description.

5.3.46 FRICTION – ACTIVATE WELLBORE FRICTION OPTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The FRICTION keyword activates the Wellbore Friction option and defines the maximum number of wellbore friction wells together with the maximum number of well branches.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description	Default
1	MXWELS	A positive integer defining the maximum number of wellbore friction wells for this model.	0
3	MXBRAN	A positive integer defining the maximum number of branches per well. The default value of one implies a standard well with no branches.	1
Notes: 1) The keyword is terminated by a "/".			

Table 5.16: FRICTION Keyword Description

Example

```
--
--       WELL    BRANCH
--       MXWELS  MXBRAN
FRICTION
         5       1                                                    /
```

The above example defines the maximum number of wellbore friction wells to be five and the maximum number of branches set to one, for standard wells.

5.3.47 FULLIMP – ACTIVATE FULLY IMPLICIT SOLUTION OPTION

Description

The FULLIMP keyword activates the Fully Implicit Solution formulation and solution options. OPM Flow uses a different numerical scheme which makes this keyword redundant; hence, OPM Flow ignores this keyword. It is documented here for completeness. The keyword as the same function as the IMPLICIT keyword in the RUNSPEC section.

There is no data required for this keyword and there is no terminating "/" for this keyword.

See section *2.2 Running OPM Flow 2023-10 From The Command Line* on how to invoke various numerical schemes via the OPM Flow command line interface.

Example

```
--
--       ACTIVATES THE FULLY IMPLICIT SOLUTION OPTION
--
FULLIMP
```

The above example switches on the fully implicit solution option; however, this has no effect in OPM Flow input decks.

5.3.48 GAS – ACTIVATE THE GAS PHASE IN THE MODEL

Description

This keyword indicate that the gas phase is present in the model and must be used for oil-gas, gas-water, oil-water-gas input decks that contain the gas phase. The keyword will also invoke data input file checking to ensure that all the required gas phase input parameters are defined in the input deck.

There is no data required for this keyword and there is no terminating "/" for this keyword.

Example

```
--
--        GAS PHASE IS PRESENT IN THE RUN
--
GAS
```

The above example declares that the gas phase is active in the model.

5.3.49 GASFIELD – DEFINE GAS FIELD OPERATIONS OPTIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The GASFIELD keyword activates and specifies the Gas Field Operations options and determines if extended compressors are present in the run and if the expedited first pass DCQ calculation should be used.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

5.3.50 GASWAT – ACTIVATE THE GAS-WATER MODEL FORMULATION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword activates the two-phase Gas-Water model, as such it is equivalent to using both the GAS and WATER keywords in the RUNSPEC section..

Note

This is an OPM Flow keyword, and should not be confused with the more general version of the GASWAT keyword used in the commercial compositional simulator.

There is no data required for this keyword and there is no terminating "/" for this keyword.

Example

```
-- ================================================================================
--
-- RUNSPEC SECTION
--
-- ================================================================================
RUNSPEC
--
--       ACTIVATE CO2 STORAGE IN THE MODEL (OPM FLOW CO2 STORAGE KEYWORD)
--
CO2STORE
--
--       ACTIVATE COMPOSITIONAL MODELING FORMULATION (OPM FLOW KEYWORD)
--
COMPS
         2                                                                      /
--
--       ACTIVATE GAS-WATER THE MODEL (OPM FLOW KEYWORD)
--
GASWAT
```

The above example declares that the run should use the Gas-Water model, together with the CO2STORE option, and with two components.

5.3.51 GDIMS – ACTIVATE INSTANTANEOUS GRADIENT OPTION AND DEFINE DIMENSIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The GDIMS keyword activates the Instantaneous Gradient option and defines the maximum dimensions as used by the GWRTWCV keyword in the SCHEDULE section. The Instantaneous Gradient option calculates derivatives of solution quantities at the current time step with respect to variations in the variables at the current time step. This is different to Gradient option that calculates the derivatives of solution quantities at the current time step with respect to variations in the variables at the initial time step, that is a time equal to zero. Consequently, the Instantaneous Gradient option can be switched on and off by the GUPFREQ keyword in the SCHEDULE section, whereas the Gradient option cannot.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

5.3.52 GIMODEL – ACTIVATE GI PSEUDO COMPOSITIONAL OPTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, GIMODEL, activates the Gi Pseudo Compositional option for gas condensate and volatile oil fluids.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

The accuracy of gas condensate and volatile oil modeling using a black-oil reservoir simulator is now firmly accepted in the industry; both depletion and gas cycling above the saturation point can be modeled and yield an acceptable match compared with compositional simulators. The main deficiency with the existing black-oil formulation is the treatment of gas injection below the saturation point, where the compositional effect of the stripping of liquid components is inversely proportional to their molecular weights, is not considered. This is because the black-oil model assumes that the saturated hydrocarbon fluid properties are only functions of pressure. Thus, when dry gas is injected into a condensate reservoir below the saturation pressure the injected gas continues to re-vaporize liquid at a rate governed only by the cell pressures. Compositional modeling indicate that this not the case.

The GI option attempts to overcome the limitation of the standard black-oil approach by extending the black-oil model using the method of Cook et al.[86] to take into account the fluid property changes occurring during gas injection. This is done by extending the fluid property treatment so that the saturated fluid properties depend on pressure (as per the standard black-oil formulation) and also on an additional parameter which characterizes the compositional changes in the reservoir liquid and vapor phases at constant pressure.

There is no data required for this keyword.

Example

```
--
--       ACTIVATE THE GI PSEUDO COMPOSITIONAL OPTION
--
GIMODEL
```

The above example switches on the Gi Pseudo Compositional option.

[86] *Cook, R. E., Jacoby, R. H., and Ramesh, A. B. "A Beta-type Reservoir Simulator for Approximating Compositional Effects During Gas Injection" paper SPE 4272, Society of Petroleum Engineers Journal (1974) 14, No. 5, 471-481.*

5.3.53 GRAVDR – ACTIVATE GRAVITY DRAINAGE AND IMBIBITION FOR DUAL POROSITY MODEL

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword switches on gravity drainage and imbibition modeling between the matrix and the fracture grid blocks in dual porosity and dual permeability runs. Note that either DZMTRX or DZMTRXV keywords in the GRID section should be used to set the matrix vertical dimensions if this option is activated.

There is no data required for this keyword and there is no terminating "/" for this keyword.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

Example

```
--
--       ACTIVATE GRAVITY DRAINAGE AND IMBIBITION FOR DUAL POROSITY MODEL
--
GRAVDR
```

The above example switches on the gravity drainage and imbibition option for the run.

5.3.54 GRAVDRB - ACTIVATE VERTICAL DISCRETIZED GRAVITY DRAINAGE AND IMBIBITION FOR DUAL POROSITY MODEL

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword switches on vertical discretized gravity drainage and imbibition modeling between the matrix and the fracture grid blocks in dual porosity and dual permeability runs. Note that the geometry of the matrix sub-cells should be set to VERTICAL on the NMATOPS keyword in the GRID section if this option is activated.

There is no data required for this keyword and there is no terminating "/" for this keyword.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

Example

```
--
--       ACTIVATE VERTICAL DISCRETIZED GRAVITY DRAINAGE AND IMBIBITION
--
GRAVDRB
```

The above example switches on the vertical discretized gravity drainage and imbibition option for the run.

5.3.55 GRAVDRM - ACTIVATE ALTERNATIVE GRAVITY DRAINAGE AND IMBIBITION FOR DUAL POROSITY MODEL

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword switches on the alternative gravity drainage and imbibition modeling between the matrix and the fracture grid blocks in dual porosity and dual permeability runs. Either the GRAVDRM or GRAVDR keywords should be used to activate this standard or alternative type of formulation.

There is no data required for this keyword.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description	Default
1	OPTION1	A defined character string that sets the matrix flow in and out of the matrix block option, and should be set to one of the following: 1) YES: oil flow is bi-directional, that is oil can flow into and out of the matrix block. 2) NO: oil flow is uni-directional, that is oil can flow out of the matrix block.	YES
Notes: 1) The keyword is terminated by a "/".			

Table 5.17: GRAVDRM Keyword Description

Example

```
--
--        ACTIVATE ALTERNATIVE GRAVITY DRAINAGE AND IMBIBITION MODEL
--
--        MATRIX
--        OPTION
GRAVDRM
          YES                                                              /
```

The above example switches on the alternative gravity drainage and imbibition option for the run and sets oil flow to be bi-directional, that is oil can flow into and out of the matrix block.

5.3.56 GRIDOPTS - GRID PROCESSING OPTIONS

Description

GRIDOPTS activates the negative directional dependent transmissibility multipliers option, defines the maximum number of MULTNUM regions and the number of PINCHNUM regions for the model.

No.	Name	Description	Default
1	TRANMULT	A character string that activates the negative directional dependent transmissibility multipliers option by setting TRANMULT to YES. Setting the value to NO switches off this option. OPM Flow uses a positive directional dependent transmissibility formulation to describe the flow between two cells, that is for cell (I, J, K) OPM Flow calculates the x face transmissibility between (I, J, K) and (I +1, J, K) cell face. Modification to the transmissibilities in this case is accomplished by the MULTX, MULTY and MULTZ keywords. Setting TRANMULT to YES invokes the option to use a negative directional dependent multiplier scheme using the MULTX-, MULTY- and MULTZ- keywords. In this case OPM Flow applies the x face transmissibility between (I - 1, J, K) and (I, J, K) cell face when using the MULTX-, MULTY- and MULTZ- keywords. Note that if TRANMULT is defaulted, and there are negative directional dependent multiplier keywords in the input deck, then OPM Flow will continue to process the MULTX-, MULTY- and MULTZ keywords correctly. Whereas, the commercial simulator will terminate with an error.	NO
2	NRMULT	A positive integer value that defines the maximum number of MULTNUM regions for the MULTNUM array. The MULTNUM array is used in the GRID section to define various inter-region transmissibility regions in the model and NRMULT sets the maximum number of regions which is the maximum value of an element in the MULTNUM array. Inter-region MULTNUM transmissibility multipliers can be defined using the MULTREGT and regional pore volumes multipliers can be set using the MULTREGP keyword.	0
3	NRPINC	A positive integer value that defines the maximum number of PINCHNUM regions for the PINCHNUM array. The PINCHNUM array is used in the GRID section to define various regions in the model and NRPINC sets the maximum of regions which is the maximum value of an element in the PINCHNUM array. Each regions criteria for setting the pinch out criteria is set by the PINCHREG keyword.	0
Notes: 1) The keyword is terminated by a "/".			

Table 5.18: GRIDOPTS Keyword Description

See also the MULTNUM, MULTREGP, MULTREGT, PINCHNUM, and PINCHREG keywords.

Example

```
--
--       NEG       MAX      MAX
--       MULTS     MULTNUM PINCHNUM
GRIDOPTS
         NO        9        1*                                                  /
```

The above example switches off the negative directional dependent transmissibility multipliers option and defines the maximum of MULTNUM regions to be nine,. The NRPINC parameter is defaulted which means there the maximum number of PINCHREG regions is zero.

5.3.57 H2STORE – ACTIVATE THE H2 STORAGE MODEL

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The H2STORE keyword activates the hydrogen (H_2) storage model for the run to account for both hydrogen and water phase solubility. H2STORE is similar to CO2STORE, which activates the carbon dioxide (CO_2) storage model.

The H_2-Brine PVT model computes the PVT properties such as density, viscosity, and enthalpy internally as functions of pressure, temperature, and composition by using analytic correlations and models from the literature rather than by interpolation from tabulated values. These values are transformed to the standard black-oil equivalent PVT tables internally by the simulator. Dissolved hydrogen in brine is modeled using Li et al.[87]. Hydrogen gas density was modeled using Leachman et al.[88]. Other properties are modeled the same way as in CO2STORE but modified for hydrogen-brine (a similar approach was recently reported by Raad et al.[89]). A full description of the underlying PVT models used by CO2STORE is described by Sandve et al.[90]. This means that the normal PVT keywords like DENSITY, PVTO, PVDG etc. are not required by OPM Flow when this model is activated, and if entered will be ignored by the simulator. Note that the H_2-Brine PVT properties depend on the temperature and salinity and these must therefore be entered in the PROPS section. The reservoir temperature can be defined using, e.g., the RTEMP keyword. Region based salinity can be provided using the SALINITY keyword.

The H2STORE keyword must be used with either: (1) the GAS and WATER keywords (or alternatively the GASWAT keyword), or (2) the GAS and OIL keywords in the RUNSPEC section. It is recommended that the standard option (1) is used.

The DISGASW keyword can also be used with option (1) to model dissolution of H_2 in the Brine.

Option (1) has the advantage that it can be used with the VAPWAT and PRECSALT keywords to model the impact of both vaporization of residual water and salt precipitation in the near wellbore region on injectivity of H_2 injection wells.

In option (1), the GAS and WATER (or GASWAT) keywords declare that the gas and water phases are present in the model. When the H2STORE option is used the water phase represents the brine and the gas phase represents H_2. Note that the input and output keywords need to be consistent with this assumption, e.g., GSF (gas saturation function) and WSF (water saturation function) should be used for the H_2-Brine relative permeability, etc.

Although, the DISGAS and VAPOIL keywords can be used with option (2) to model water vaporization and H_2 dissolution, salt precipitation is not currently supported with option (2).

Option (2) currently has the advantage that it can be used with the DRSDTCON keyword to control convective dissolution of H_2 into in-situ brine. Support for the use of DRSDTCON with option (1) is planned to be added in the next release.

[87] *Dedong Li, Christof Beyer, Sebastian Bauer, A unified phase equilibrium model for hydrogen solubility and solution density, International Journal of Hydrogen Energy, Volume 43, Issue 1, 2018, Pages 512-529.*

[88] *J. W. Leachman, R. T Jacobsen, S. G. Penoncello, E. W. Lemmon; Fundamental Equations of State for Parahydrogen, Normal Hydrogen, and Orthohydrogen. J. Phys. Chem. Ref. Data 1 September 2009; 38 (3): 721–748.*

[89] *Seyed Mostafa Jafari Raad, Ehsan Ranjbar, Hassan Hassanzadeh, Yuri Leonenko, Hydrogen-brine mixture PVT data for reservoir simulation of hydrogen storage in deep saline aquifers, International Journal of Hydrogen Energy, Volume 48, Issue 2, 2023, Pages 696-708.*

[90] *Tor Harald Sandve, Sarah E. Gasda, Atgeirr Rasmussen, and Alf Birger Rustad. Convective dissolution in field scale CO_2 storage simulation using the OPM Flow simulator. Submitted to TCCS 11 – Trondheim Conference on CO_2 Capture, Transport and Storage Trondheim, Norway – June 21-23, 2021.*

In option (2), the GAS and OIL keywords declare that the gas and oil phases are present in the model. Internally when H2STORE is used the oil phase refers to the brine and the gas phase to H_2. Again, the input and output keywords need to be consistent with this assumption, e.g., SGOF (gas-oil relative permeability) is used for the H_2-Brine relative permeability, FOIP (Field Oil-In-Place) shows the total amount of brine in the reservoir, etc.

There is no data required for this keyword and there is no terminating "/" for this keyword.

Examples

The first example shows the standard useage of H2STORE with Option (1) the Gas-Water model (GASWAT). Here we also activate the dissolved gas in water (DISGASW) and vaporized water in gas (VAPWAT) options.

```
-- ==========================================================================
--
-- RUNSPEC SECTION
--
-- ==========================================================================
RUNSPEC
-- --------------------------------------------------------------------------
-- FLUID TYPES AND TRACER OPTIONS
-- --------------------------------------------------------------------------
--
--       ACTIVATE H2 STORAGE IN THE MODEL (OPM FLOW H2 STORAGE KEYWORD)
--
H2STORE
--
--       ACTIVATE GAS-WATER THE MODEL (OPM FLOW KEYWORD)
--
GASWAT
--
--       DISSOLVED GAS IN WATER IS PRESENT IN THE RUN (OPM FLOW KEYWORD)
--
DISGASW
--
--       VAPORIZED WATER IN DRY/WET GAS IS PRESENT IN THE RUN (OPM FLOW KEYWORD)
--
VAPWAT
```

The second part of the example covers the data required for the PROPS section, in which the two-phase relative permeability functions are set using GSF and WSF keywords.

```
-- ==========================================================================
--
-- PROPS SECTION
--
-- ==========================================================================
PROPS
--
--       RESERVOIR
--       TEMPERATURE
--       -----------
RTEMP
         80.0                                          / RESERVOIR TEMP

--
--       ROCK COMPRESSIBILITY
--
--       REFERENCE PRESSURE IS TAKEN FROM THE HCPV WEIGHTED RESERVOIR PRESSURE
```

```
--
--        REF PRES  CF
--        BARSA     1/BARSA
--        --------  --------
ROCK
          200.0     5.0E-05                                  / ROCK COMPRESSIBILITY
--
--        GAS RELATIVE PERMEABILITY TABLES (OPM FLOW KEYWORD)
--
GSF
--        SGAS       KRG        PCGW
--        FRAC                  PSIA
--        --------  --------   -------
             0.000    0.000       0.0
             0.080    0.001       0.0
             0.170    0.010       0.0
             0.350    0.050       0.0
             0.530    0.200       0.0
             0.620    0.350       0.0
             0.650    0.390       0.0
             0.710    0.560       0.0
             0.800    1.000       0.0   / TABLE NO. 01
--
--        WATER RELATIVE PERMEABILITY TABLES (OPM FLOW KEYWORD)
--
WSF
--        SWAT       KRW
--        FRAC
--        --------   --------
             0.200   0.0000
             0.400   0.1000
             0.800   0.5000
             1.000   1.0000             / TABLE NO. 01
--
--        SET SALINITY FOR ALL CELLS (OPM FLOW KEYWORD)
--
SALINITY
0.7                                                       /
-- ===========================================================================
```

No other data is required to define the fluid and rock properties in the PROPS section as the data is generated from internal analytic correlations and models by the simulator. Finally, note that units for salinity are to the 10^{-3}, thus for metric units we have 10^{-3} x kg-M/kg.

The third part of the example covers initializing the model in the SOLUTION section. Here we use the EQUIL(EQLOPT6) parameter equal to one, to use table number one of the RVWVD keyword, in order to set the vaporized water versus depth distribution for the model.

```
-- ================================================================================
--
-- SOLUTION SECTION
--
-- ================================================================================
SOLUTION
--
--       SYSTEM IS SATURATED WITH WATER
--
--       DATUM   DATUM   OWC     PCOW   GOC     PCGO   RS   RV   N    E300  RVW
--       DEPTH   PRESS   DEPTH   ----   DEPTH   ----   OPT  OPT  OPT  OPT   OPT
EQUIL
         2000.0  200.0   1800.0  0.00  1800.0  0.00   1*   1*   1*   2*    1   /
--
--       WATER VAPOR RATIO VS DEPTH (OPM FLOW KEYWORD)
--
--       DEPTH    RVW
--                STB/MSCF
--       ------   --------
RVWVD
         1000.0   0.000
         3000.0   0.000                                / RVW VS DEPTH EQUIL REGN 01
```

For the SUMMARY section, the simulator supports, several summary vectors specific to H_2 storage, as shown below.

```
-- ================================================================================
--
-- SUMMARY SECTION
--
-- ================================================================================
SUMMARY
--
--       EXPORT STANDARD SUMMARY VARIABLE VECTORS TO FILE
--
ALL
--
--       FIELD H2 DISOLVED IN WATER PHASE
FWCD
--
--       FIELD H2 TRAPPED IN GAS PHASE
FGCDI
--
--       FIELD H2 MOBILE IN GAS PHASE
FGCDM
```

The final part of the example covers the SCHEDULE section. The standard WCONINJE keyword is then used to set the gas injection rate, in this case 100,000 sm^3/day of H_2.

```
-- ================================================================================
--
-- SCHEDULE SECTION
--
-- ================================================================================
SCHEDULE
--
--       RESTART CONTROL BASIC = 4 (ALL=2, YEARLY=4, MONTHLY=5, TSTEP=6)
--
RPTRST
        'BASIC=1' 'ALLPROPS'                                                    /
--
--       WELL SPECIFICATION DATA
--
```

```
-- WELL  GROUP     LOCATION  BHP    PHASE  DRAIN  INFLOW  OPEN  CROSS  PVT
-- NAME  NAME        I    J  DEPTH  FLUID  AREA   EQUANS  SHUT  FLOW   TABLE
WELSPECS
INJ1     'G1'       25   25  1*     GAS                                    /
/
--
--       WELL CONNECTION DATA
--
-- WELL  --- LOCATION ---  OPEN   SAT   CONN   WELL   KH    SKIN   D     DIR
-- NAME   II  JJ  K1  K2   SHUT   TAB   FACT   DIA    FACT  FACT   FACT  PEN
COMPDAT
INJ1      25  25   1  10  OPEN    1*    1*    0.3                          /
/
--
--       WELL INJECTION CONTROLS
--
-- WELL  FLUID  OPEN/  CNTL  SURF   RESV   BHP   THP   VFP
-- NAME  TYPE   SHUT   MODE  RATE   RATE   PRES  PRES  TABLE
WCONINJE
INJ1     GAS    OPEN   RATE 100000  1*     300                            /
/
--
--       ADVANCE SIMULATION BY REPORTING TIME
--
TSTEP
         1   36*30
/
```

Output of the liquid phase mole fractions of H2, that is, the mole factions of H2 in the water phase, and the vapor phase mole factions to the restart file, is only supported for CO2STORE.

The second example shows how to use H2STORE with the alternative option (2). The example below declares that the hydrogen storage model is active for the run to account for both hydrogen and water phase solubility using OPM Flow's H_2-Brine PVT model. Option (2) is used where the OIL phase refers to the brine and the GAS phase to H_2.

```
-- ======================================================================
--
-- RUNSPEC SECTION
--
-- ======================================================================
RUNSPEC
-- ----------------------------------------------------------------------
-- FLUID TYPES AND TRACER OPTIONS
-- ----------------------------------------------------------------------
--
--       ACTIVATE H2 STORAGE IN THE MODEL (OPM FLOW H2 STORAGE KEYWORD)
--
H2STORE
--
--       OIL PHASE IS PRESENT IN THE RUN BUT IS THE BRINE PHASE FOR H2STORE
--
OIL
--
--       GAS PHASE IS PRESENT IN THE RUN BUT IS THE H2 PHASE FOR H2STORE
--
GAS
--
--       DISSOLVED GAS IN LIVE OIL IS PRESENT IN THE RUN (H2 IN BRINE)
--
DISGAS
```

```
--
--       VAPORIZED OIL IN WET GAS IS PRESENT IN THE RUN (WATER IN H2)
--
VAPOIL
```

The second part of the example covers the data required for the PROPS section, in which the input keywords need to be consistent with the OIL phase referring to the Brine and the GAS to H_2; that is SGOF (gas-oil relative permeability) is used to define the H_2-Brine relative permeability table.

```
-- ======================================================================
--
-- PROPS SECTION
--
-- ======================================================================
PROPS
--
--       RESERVOIR
--       TEMPERATURE
--       -----------
RTEMP
90.0                                              / RESERVOIR TEMPERATURE
--
--       ROCK COMPRESSIBILITY
--
--       REFERENCE PRESSURE IS TAKEN FROM THE HCPV WEIGHTED RESERVOIR PRESSURE
--
--       REF PRES  CF
--       BARSA     1/BARSA
--       --------  --------
ROCK
         1.0000    1.0E-06                                 / ROCK COMPRESSIBILITY
--
--       GAS-OIL RELATIVE PERMEABILITY TABLES (SGOF) - H2STORE PHASES
SGOF
--       SG          KRG       KROG      PCOG
--       FRAC                            PSIA
--       -------   --------  -------   -------
         0.00000   0.000000  1.00000    0.0000
         1.00000   1.000000  0.00000    0.0000             / TABLE No. 01
--
--       SET SALINITY FOR ALL CELLS (OPM FLOW KEYWORD)
--
SALINITY
0.7                                                     /
-- ======================================================================
```

The third part of the example covers initializing the model in the SOLUTION section. Here we set the EQUIL(EQLOPT1 and EQLOPT2) parameters equal to one, to use table number one of the RSVD and RVVD keywords, in order to set the initial dissolved H_2 and vaporised water versus depth distribution for the model.

```
-- ======================================================================
--
-- SOLUTION SECTION
--
-- ======================================================================
SOLUTION
--
--       SYSTEM IS SATURATED WITH OIL (BRINE)
--
--       DATUM   DATUM   OWC     PCOW   GOC     PCGO   RS   RV   N    E300  RVW
--       DEPTH   PRESS   DEPTH   ----   DEPTH   ----   OPT  OPT  OPT  OPT   OPT
```

```
EQUIL
        2000.0  200.0   2200.0  0.00  1800.0  0.00   1    1    1*   2*    1*   /
--
--      DEPTH    RS
--               MSCF/STB
--      ------   --------
RSVD
        1000.0   0.000
        3000.0   0.000                                      /
--
--      DEPTH    RV
--               STB/MSCF
--      ------   --------
RVVD
        1000.0   0.000
        3000.0   0.000                                       /
```

The forth and final part of the example sets the maximum dissolution rate for convective H_2 mixing via the DRSDTCON keyword in the SCHEDULE section.

```
-- ======================================================================
--
-- SCHEDULE SECTION
--
-- ======================================================================
SCHEDULE
--
--      H2 CONVECTIVE DISSOLUTION PARAMETER
--
DRSDTCON
--      H2 CONV
--      DISSOLN
--      --------
        0.04                                          /
```

See the DRSDTCON keyword in the SCHEDULE section for further information on this keyword.

5.3.58 HMDIMS – DEFINE HISTORY MATCH GRADIENT PARAMETER DIMENSIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, HMDIMS, defines the maximum parameter dimensions for the History Match Gradient option.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

5.3.59 HYST – ACTIVATE THE HYSTERESIS OPTION (RETIRED)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The HYST keyword activates the hysteresis option, the keyword should be avoided and the hysteresis option should be enabled by the HYSER parameter on the SATOTPS keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

5.3.60 IMPES – ACTIVATE IMPLICIT PRESSURE EXPLICIT SATURATION SOLUTION OPTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The IMPES keyword activates the Implicit Pressure Explicit Saturation formulation and solution options, commonly know as IMPES. OPM Flow uses a different numerical scheme which makes this keyword redundant; hence, OPM Flow ignores this keyword. It is documented here for completeness.

There is no data required for this keyword and there is no terminating "/" for this keyword.

See section *2.2 Running OPM Flow 2023-10 From The Command Line* on how to invoke various numerical schemes via the OPM Flow command line interface.

Example

```
--
--       ACTIVATE THE IMPES SOLUTION OPTION
--
IMPES
```

The above example switches on the IMPES solution option; however, this has no effect in OPM Flow input decks.

5.3.61 IMPLICIT – ACTIVATE FULLY IMPLICIT SOLUTION OPTION

Description

The IMPLICIT keyword activates the Fully Implicit Solution formulation and solution options. OPM Flow uses a different numerical scheme which makes this keyword redundant; hence, OPM Flow ignores this keyword. It is documented here for completeness. The keyword as the same function as the FULLIMP keyword in the RUNSPEC section.

There is no data required for this keyword and there is no terminating "/" for this keyword.

See section *2.2 Running OPM Flow 2023-10 From The Command Line* on how to invoke various numerical schemes via the OPM Flow command line interface.

Example

```
--
--       ACTIVATES THE FULLY IMPLICIT SOLUTION OPTION
--
IMPLICIT
```

The above example switches on the fully implicit solution option; however, this has no effect in OPM Flow input decks.

5.3.62 INCLUDE – LOAD ANOTHER DATA FILE AT THE CURRENT POSITION

The INCLUDE keyword informs OPM Flow to continue reading input data from the specified INCLUDE file. When the end of the INCLUDE file is reached, or the ENDINC keyword is encountered in the included file, input data is read from the next keyword in the current file.

See *INCLUDE – Load Another Data File at the Current Position* in the GLOBAL section for a full description.

5.3.63 INSPEC – ACTIVATE THE INSPEC FILE OPTION

Description

This keyword switches on the writing of the INIT Index file that specifies and defines the format and data type written to the *.INIT data file. The *.INIT data file contains the static data specified in the GRID, PROPS and REGIONS sections. For example, the PORO, PERM and NTG arrays from the GRID section. The data is used in post-processing software, for example OPM ResInsight, to visualize the static grid properties.

The INIT Index file can either be written out in formatted form as ASCII i.e. text files, if the FMTOUT keyword has been activated (*.FINSPEC), or binary format (*.INSPEC) if the FMTOUT keyword has not been activated. If the INIT keyword in the GRID section has been used to switch on the writing of the *. INIT data file then a binary INIT Index file is automatically written out as well, unless the NOINSPEC keyword in the RUNSPEC section has been used to switch off the writing of the INIT Index file.

Note that most post-processing software require the *.INSPEC file to load the *.INIT data set, although OPM ResInsight does not require this file to be able to load the *.INIT data file.

There is no data required for this keyword and there is no terminating "/" for this keyword.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

Example

```
--
--       ACTIVATE WRITING THE INIT INDEX FILE FOR POST-PROCESSING
--
INSPEC
```

The above example switches on the writing of the INIT Index file for post-processing in ResInsight.

5.3.64 LAB - ACTIVATE THE LABORATORY SYSTEM OF UNITS FOR THE MODEL

Description

This keyword switches on the LABORATORY system of units for the model.

OPM Flow has three sets of units, namely: METRIC, FIELD and LAB and one of these keyword should be invoked in the RUNSPEC section to avoid any ambiguity. Both the simulator input and output units are controlled by including one of the METRIC, FIELD or LAB keywords in the RUNSPEC section of the input file.

There is no data required for this keyword and there is no terminating "/" for this keyword.

Example

```
--
--        SWITCH ON THE LABORATORY SYSTEM OF UNITS FOR BOTH INPUT AND OUTPUT
--
LAB
```

The above example switches on the LABORATORY system of units for the model.

5.3.65 LGR – DEFINE LOCAL GRID REFINEMENT DIMENSIONS AND PARAMETERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, LGR, defines maximum dimensions and parameters for the Local Grid Refinement ("LGR") option.

Currently, OPM Flow does not support the local grid refinement feature and therefore this keyword is ignored by the simulator, but is documented here for completeness.

No.	Name	Description	Default
1	MAXLGR	A positive integer value that defines the maximum number of LGRs in the model.	0
2	MAXCLS	A positive integer value that defines the maximum number of grid blocks in all the LGRs.	0
3	MCOARS	A positive integer value that defines the maximum number of amalgamated coarse grid blocks in the model.	0
4	MAMALG	A positive integer value that defines the maximum number of LGR amalgamations in the model.	0
5	MXLALG	A positive integer value that defines the maximum number of LGRs in any amalgamation in the model.	0
6	LSTACK	A positive integer that defines the maximum number of previous search directions stored by the linear solver for the LGR. See the NSTACK keyword in the RUNSPEC section for a full description.	10
7	INTOPT	A character string set to either INTERP to activate the Quandalle[91] pressure correction, or NOINTERP to deactivate this option. The option applies bi-linear interpolation to the global cells surrounding an LGR in order to improve the accuracy of the flow calculations between the LGR and the host cells.	NOINTERP
8	NCHCOR	A positive integer value that defines the maximum number of grid blocks within a coarsened grid that overlap parallel domain boundaries for when the Parallel option has been invoked by the PARALLEL keyword in the RUNSPEC section. OPM Flow uses a different numerical scheme which makes this parameter redundant, see section *2.2 Running OPM Flow 2023-10 From The Command Line* on how to run OPM Flow in parallel mode.	0
Notes: 1) The keyword is terminated by a "/".			

Table 5.19: LGR Keyword Description

[91] *Quandalle, Philippe & Besset, P. (1985). Reduction of Grid Effects Due to Local Sub-Gridding in Simulations Using a Composite Grid. 10.2118/13527-MS.*

Example

```
--
--        LOCAL GRID REFINEMENT DIMENSIONS AND PARAMETERS
--
--        LGR      LGR      LGR      LGR      LGR      LGR     LGR      LGR
--        MAXLGR   MAXCLS   MCOARS   MAMALG   MXLALG   LSTACK  INTOPT   NCHCOR
LGR
          10       1000     1*       1*       1*       1*      INTERP   1*              /
```

The above example sets the maximum number of LGRs to 10 and the maximum number of grid blocks a LGR may contain to 1,000, and that Quandalle pressure correction should be used to improve the flow calculation.

5.3.66 LGRCOPY – ACTIVATE LOCAL GRID REFINEMENT INHERITANCE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The LGRCOPY keyword actives Local Grid Refinement ("LGR") Inheritance option that allows the LGR to inherit the properties of the global or host cell containing a LGR grid block at the start of the GRID section, as oppose to the normal process of applying this transform at the end of the GRID section. LGRCOPY can be used in the RUNSPEC, GRID and EDIT sections. If used in the RUNSPEC section then the option is applied to all LGRs defined in the input file, whereas if used in the GRID and EDIT sections the keyword must be placed inside a LGR definition section, that is between a CARFIN (Cartesian LGR grid) or RADIN/RADIN4 (radial LGR grid) and the ENDFIN keyword. In the latter case inheritance is applied on an individual LGR basis.

Currently, OPM Flow does not support the local grid refinement feature and therefore this keyword is ignored by the simulator.

Example

The following example activates the LGR Inheritance option for all LGRs in the model.

```
-- ======================================================================
--
-- RUNSPEC SECTION
--
-- ======================================================================
RUNSPEC
--
--       ACTIVATE LOCAL GRID REFINEMENT INHERITANCE
--
LGRCOPY
```

5.3.67 LICENSES – DEFINE REQUIRED LICENSES FOR RUN

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword defines the additional software licenses that are required to invoke various licensed options in the commercial simulator at the start of the run. The commercial simulator requests a license when keywords associated with a licensed option is encountered in the input deck, this may result in the license being unavailable at the time of request and after the simulation has been initiated, resulting in the run terminating. This keyword avoids this scenario by reserving the license at the start of the run.

OPM Flow is an open source project and therefore there is no license management of the various implemented options; hence this keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

5.3.68 LIVEOIL – ACTIVATE THE LIVE OIL PHASE (OIL WITH FREE AND DISSOLVED GAS)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword activates oil, free and dissolved gas in the model and therefore makes the oil phase live oil[92] in the black-oil formulation, and is equivalent to setting the phases present in the model to be oil, dissolved gas, gas and water phases. Note if water is present in the model this needs to be explicitly stated via the WATER keyword in the RUNSPEC section (see also the BLACKOIL and DEADOIL keywords in the RUNSPEC section). The keyword is used by the commercial simulator's compositional THERMAL option to set the phases present in the model.

There is no data required for this keyword and there is no terminating "/" for this keyword.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

Example

The following example activates the black-oil phases in the model.

```
--
--       ACTIVATE LIVE-OIL PHASE
--
LIVEOIL
```

Alternatively one could explicitly declare the phases using the following keywords in the RUNSPEC section.

```
--
--       OIL PHASE IS PRESENT IN THE RUN
--
OIL
--
--       DISSOLVED GAS IN LIVE OIL IS PRESENT IN THE RUN
--
DISGAS
--
--       GAS PHASE IS PRESENT IN THE RUN
--
GAS
--
--       WATER PHASE IS PRESENT IN THE RUN
--
WATER
```

The above example switches on the oil, dissolved gas, gas and water phases in the model.

[92] *"Live" oil is oil that contains gas in solution, which is normally the case for most conventional oil reservoirs. However, for oil reservoirs classified as heavy oil reservoirs, the in situ dissolved gas may be negligible and oil would then be classified as gas-free oil which is commonly referred to as "dead" oil.*

5.3.69 LOAD – LOAD A SAVE FILE FOR A FAST RESTART

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The LOAD keyword loads a previously generated SAVE file to enable a fast restart. A SAVE file contains all the data from a previous run's RUNSPEC, GRID, EDIT, PROPS and REGIONS sections, and thus there is no need for the simulator to calculate various parameters, including grid block transmissibilities etc. This allows for the current run to restart quicker than a conventional restart run using the RESTART keyword in the SOLUTION section via a RESTART file (*.UNRST or *.FUNRST etc.). The keyword should be the first keyword in the input deck and the RUNSPEC, GRID, EDIT, PROPS and REGIONS sections should be deleted from the input deck.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

5.3.70 LOWSALT – ACTIVATE THE LOW SALT BRINE PHASE IN THE BRINE MODEL

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, LOWSALT, activates the low salt brine phase for the Brine option and also activates the Brine option. See also the BRINE keyword in the RUNSPEC section.

There is no data required for this keyword and there is no terminating "/" for this keyword.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

Example

```
--
--       ACTIVATE THE LOW SALT BRINE PHASE FOR THE BRINE OPTION
--
LOWSALT
```

The above example declares that the low salt brine phase is active in the model for the Brine option.

5.3.71 MEMORY – DEFINE ALLOCATED MEMORY (RETIRED)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword defines the memory allocation for the run.

OPM Flow and now the commercial simulator uses dynamic memory allocation and therefore the keyword has no effect and is ignored by both simulators.

5.3.72 MESSAGE – OUTPUT USER MESSAGE

The MESSAGE keyword outputs a user message to the terminal, as well as to the print (*.PRT) and debug (*.DBG) files. Note this is different to the MESSAGES keyword, that defines OPM Flows message print limits and stop limits generated by the simulator.

See *MESSAGE – Output User Message* in the GLOBAL section for a full description.

5.3.73 MESSAGES – DEFINE MESSAGE PRINT LIMITS AND STOP LIMITS

The MESSAGES keyword defines the print and stops levels for various messages. The "print limits" set the maximum number of messages that will be printed, after which no more messages will be printed and the "stop limits" terminate the run when these limits are exceeded. There are six levels of message that increase in severity from informative all the way to programming errors, as outlined in Error: Reference source not found.

See *MESSAGES – Define Message Print Limits and Stop Limits* in the GLOBAL section for a full description.

5.3.74 MESSSRVC - ACTIVATE OR DEACTIVATE DATABASE MESSAGE FILE OUTPUT

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The MESSSRVC keyword activates or deactivates output to the database message file (*.DBPRTX). The file contains all the messages from run in binary format and is used in some post-processing software to annotate production line plots from the run.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

5.3.75 METRIC – ACTIVATE THE METRIC SYSTEM OF UNITS FOR THE MODEL

Description

This keyword switches on the METRIC system of units for the model.

OPM Flow has three sets of units, namely: METRIC, FIELD and LAB and one of these keyword should be invoked in the RUNSPEC section to avoid any ambiguity. Both the simulator input and output units are controlled by including one of the METRIC, FIELD or LAB keywords in the RUNSPEC section of the input file.

There is no data required for this keyword and there is no terminating "/" for this keyword.

Example

```
--
--       SWITCH ON THE METRIC SYSTEM OF UNITS FOR BOTH INPUT AND OUTPUT
--
METRIC
```

The above example switches on the METRIC system of units for the model.

5.3.76 MICP – ACTIVATE THE MICROBIALLY INDUCED CALCITE PRECIPITATION MODEL

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The MICP keyword activates the Microbially Induced Calcite Precipitation ("MICP") model for the run. MICP is a new and sustainable technology which utilizes biochemical processes to create barriers by calcium carbonate cementation, the technology has the potential to be used for sealing leakage zones in geological formations. Further information on the mathematical model can be found in the open-access publications Landa-Marbán et al[93] and [94].

Note

This is an OPM Flow specific keyword used to investigate CO2 leakage remediation. The module requires that both the MICP and WATER keywords in the RUNSPEC to be active.

There is no data required for this keyword and there is no terminating "\" for this keyword.

Example

```
--
--       ACTIVATE THE MICROBIAL INDUCED CALCITE PRECIPITATION MODEL
--
MICP
--
--       WATER PHASE IS PRESENT IN THE RUN
--
WATER
```

The above example declares that the MICP model is active for the run and activates the water phase for the MICP model.

[93] *Landa-Marbán, D., Tveit, S., Kumar, K., Gasda, S.E., 2021. Practical approaches to study microbially induced calcite precipitation at the eld scale. Int. J. Greenh. Gas Control 106, 103256. https://doi.org/10.1016/j.ijggc.2021.103256.*

[94] *Landa-Marbán, D., Kumar, K., Tveit, S., Gasda, S.E., 2021. Numerical studies of CO2 leakage remediation by micp-based plugging technology. In: Røkke, N.A. and Knuutila, H.K. (Eds) Short Papers from the 11th International Trondheim CCS conference, ISBN: 978-82-536-1714-5, 284-290.*

5.3.77 MINNPCOL - DEFINE THE MINIMUM NUMBER OF NEWTON ITERATIONS USED TO UPDATE WELL TARGETS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The MINNPCOL keyword defines the minimum number of Newton iterations within a time step that may be used to update the well production and injection targets, after which the well targets will be frozen until the time step calculations have converged and the time step is complete.

Note

This is an OPM Flow specific keyword that sets the minimum number of Newton iterations, as opposed to the commercial simulator's NUPCOL keyword that defines the maximum number of Newton iterations within a time step, after which well targets are frozen.

Wells under group control may suffer from some dependency with other wells in the same group that are under group control. This may cause some oscillation in the production and injection well rates within the group. In order to avoid this, the NUPCOL keyword in the RUNSPEC section can be used to set the maximum number of Newton iterations within a time step, after which the group well rates are frozen until the time step has converged. Reducing the potential of well rate oscillations within the time step may result in the group targets and limits not being exactly met in this case. Increasing the value of NUPCOL, will improve the accuracy of the group targets and limits at the expense of computational efficiency. Here, the MINNPCOL set the minimum number of Newton iterations.

See also section *2.2 Running OPM Flow 2023-10 From The Command Line* on how to set various other numerical control parameters for OPM Flow.

No.	Name	Description	Default
1	MINNPCOL	A positive integer that defines the minimum number of Newton iterations used to update well targets within a time step. Note that default value of six is different to the commercial simulator's default value of three used on the NUPCOL keyword in the RUNSPEC section.	6
Notes: 1) The keyword is terminated by a "/".			

Table 5.20: MINNPCOL Keyword Description

Example

```
--
--       DEFINE THE MIN NUMBER OF ITERATIONS TO UPDATE WELL FLOW TARGETS
--
MINNPCOL
         6                                                                  /
```

The above example sets the default MINNPCOL value to the default value of six.

5.3.78 MISCIBLE – DEFINE MISCIBILITY TODD-LONGSTAFF PARAMETERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The MISCIBLE keyword defines the options associated with the Todd-Longstaff[95] mixing parameters used for when polymer flooding or CO2 EOR simulation cases are being run.

No.	Name	Description	Default
1	NTMISC	A positive integer value that declares the number miscible residual oil saturations versus water saturations tables for SORWMIS keyword and the number Todd-Longstaff mixing parameters entries on the TLMIXPAR keyword.	1
2	NSMISC	A positive integer value that sets the maximum number of entries (or rows) for each SORWMIS table defined by the SORWMIS keyword.	20
3	MISOPT	A character string that defines the numerical dispersion control options for the oil and gas relative permeability curves, set to either NONE or TWOPOINT: 1) NONE – standard single point up streaming, that is using the immediate neighbor 2) TWOPOINT – two-point up streaming, that is using the immediate neighbor plus one cell for better numerical dispersion control but with a higher computational cost. Only the default value of NONE is supported.	NONE
Notes: 1) The keyword is terminated by a "/".			

Table 5.21: MISCIBLE Keyword Description

Example

```
--
--        NTAB     MAX      UPSTRM
--        NTMISC   NSMISC   MISOPT
MISCIBLE
          1        20       NONE                                    /
```

The above example defines the default values for the MISCIBLE keyword, that is one table with a maximum of 20 rows per table using the standard one cell upstream option.

[95] *M. R. Todd and W. J Longstaff, The Development, Testing, and Application Of a Numerical Simulator for Predicting Miscible Flood Performance". In: J. Petrol. Tech. 24.7 (1972), pages 874-882.*

5.3.79 MONITOR – ACTIVATE OUTPUT OF THE MONITORING DATA AND FILE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The MONITOR keyword activates the writing out of the run time monitoring information used by post-processing graphics software to display run time information, for example the simulated production and injection rates and cumulative values. OPM Flow does not have this functionality.

Hence, OPM Flow ignores this keyword but it is documented here for completeness.

There is no data required for this keyword and there is no terminating "/" for this keyword.

Example

```
--
--       ACTIVATE MONITORING OUTPUT DATA AND FILES
--
MONITOR
```

The above example switches on the output required for run time monitoring required by post-processing graphics software to review the simulation results in real time as the run progresses; however, this has no effect in OPM Flow input decks.

5.3.80 MSGFILE – ACTIVE OR DEACTIVATE MESSAGE FILE OUTPUT

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

MSGFILE keyword activates or deactivates the message file output used by pre- and post-processing software. Note that message file processing is not available in OPM Flow.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

No.	Name	Description	Default
1	MSGOPT	A positive integer set to 0 for to deactivate message file output or 1 to activate message file output.	1
Notes: 1) The keyword is terminated by a "/".			

Table 5.22: MSGFILE Keyword Description

Example

```
--
--        OUTPUT
--        OPTN
MSGFILE
          0                                                                    /
```

The above example deactivates the message file output, but the keyword is ignored by OPM Flow.

5.3.81 MULTIN – ACTIVATE THE NON-UNIFIED MULTIPLE INPUT FILE OPTION

Description

This keyword switches on the Multiple Input Files option for all input files. Similar to the commercial simulator, OPM Flow has various options for reading various input files and writing the resulting OPM Flow output files as described in Table 5.23.

Process	Keyword	Description	Files
Input	FMTIN	A character string that defines the input files to be formatted as ASCII i.e. text files, as oppose to binary files. The input deck file is always of this type. The option relates to the OPM Flow derived files that used as input, for for example when restarting from another case. If the keyword is omitted then the default is for binary file input.	*.FEGRID *.FINSPEC *.FINIT *.FRSSPEC *.FUNRST *.FSMSPEC *.FUNSMRY
	MULTIN	A character string that defines the input files to be non-unified multiple files, as opposed to unified files. In this case, one file is read in per reporting time step, as opposed to all time steps reports being read from one file. If the keyword is omitted then the default is for one file per reporting time step.	*.RSSPEC *.X0001 *.SMSPEC *.S0001
	UNIFIN	A character string that defines the input files to be unified files, as opposed to non-unified multiple files. A unified file is a single file containing output for each reporting time step. For this option a single summary file and a single restart file will be read. If the keyword is omitted then the default is for one file per reporting time step.	*.RSSPEC *.UNRST *.SMSPEC *.UNSMRY
Output	FMTOUT	A character string that sets all output files to be formatted as ASCII i.e. text files, as opposed to binary files. The *.PRT, *.LOG and *.DBG files are always of this type. The option relates to the OPM Flow output files only. In this case the files will be portable across operating systems, but will also be very large in terms of hard disk space. For this reason it is recommend that the default option is used so that binary files are outputted. If the keyword is omitted then the default is for binary file input.	*.FEGRID *.FINSPEC *.FINIT *.FRSSPEC *.FUNRST *.FSMSPEC *.FUNSMRY
	MULTOUT	A character string that defines the output files to be non-unified multiple files, as opposed to unified files. In this case, one file is written for each reporting time step, as opposed to all time steps reports being written in one file. If the keyword is omitted then the default is for one file per reporting time step.	*.RSSPEC *.X0001 *.SMSPEC *.S0001

Process	Keyword	Description	Files
	UNIFOUT	A character string that defines the output files to be unified files, as opposed to non-unified multiple files. A unified file is a single file containing output for each reporting time step. Here a single summary file and a single restart file will be generated, as opposed to one file per report time step. If the keyword is omitted then the default is for one file per reporting time step.	*.RSSPEC *.UNRST *.SMSPEC *.UNSMRY

Notes:

1) A binary file is computer-readable but not human-readable. All executable programs are stored in binary files, as are most numeric data files. In contrast, text files are stored in a form (usually ASCII) that is human-readable.
2) For unified files if the run terminates unexpectedly, or there is sufficient disk space, then the last report output is not stored. Their main advantage is that if a number of simulations reside in one directory, their output is organized. There is no limit on the number of reporting steps that a unified file can store.

Table 5.23: MULTIN Keyword Description

There is no data required for this keyword.

See also *OPM FLOW OUTPUT FILE FORMATS* for a more detailed description of the various file types (ASCII or binary) and file structure formats (unified or non-unified formats).

Example

```
--
--       ACTIVATE THE MULTIPLE INPUT FILES OPTION
--
MULTIN
```

The above example switches on the multiple input file option.

5.3.82 MULTOUT – ACTIVATE THE NON-UNIFIED MULTIPLE OUTPUT FILE OPTION

Description

This keyword switches on the Multiple Output Files option for all output files. Similar to the commercial simulator, OPM Flow has various options for reading various input files and writing the resulting OPM Flow output files as described in Table 5.24.

Process	Keyword	Description	Files
Input	FMTIN	A character string that defines the input files to be formatted as ASCII i.e. text files, as oppose to binary files. The input deck file is always of this type. The option relates to the OPM Flow derived files that used as input, for for example when restarting from another case. If the keyword is omitted then the default is for binary file input.	*.FEGRID *.FINSPEC *.FINIT *.FRSSPEC *.FUNRST *.FSMSPEC *.FUNSMRY
	MULTIN	A character string that defines the input files to be non-unified multiple files, as opposed to unified files. In this case, one file is read in per reporting time step, as opposed to all time steps reports being read from one file. If the keyword is omitted then the default is for one file per reporting time step.	*.RSSPEC *.X0001 *.SMSPEC *.S0001
	UNIFIN	A character string that defines the input files to be unified files, as opposed to non-unified multiple files. A unified file is a single file containing output for each reporting time step. For this option a single summary file and a single restart file will be read. If the keyword is omitted then the default is for one file per reporting time step.	*.RSSPEC *.UNRST *.SMSPEC *.UNSMRY
Output	FMTOUT	A character string that sets all output files to be formatted as ASCII i.e. text files, as opposed to binary files. The *.PRT, *.LOG and *.DBG files are always of this type. The option relates to the OPM Flow output files only. In this case the files will be portable across operating systems, but will also be very large in terms of hard disk space. For this reason it is recommend that the default option is used so that binary files are outputted. If the keyword is omitted then the default is for binary file input.	*.FEGRID *.FINSPEC *.FINIT *.FRSSPEC *.FUNRST *.FSMSPEC *.FUNSMRY
	MULTOUT	A character string that defines the output files to be non-unified multiple files, as opposed to unified files. In this case, one file is written for each reporting time step, as opposed to all time steps reports being written in one file. If the keyword is omitted then the default is for one file per reporting time step.	*.RSSPEC *.X0001 *.SMSPEC *.S0001

Process	Keyword	Description	Files
	UNIFOUT	A character string that defines the output files to be unified files, as opposed to non-unified multiple files. A unified file is a single file containing output for each reporting time step. Here a single summary file and a single restart file will be generated, as opposed to one file per report time step. If the keyword is omitted then the default is for one file per reporting time step.	*.RSSPEC *.UNRST *.SMSPEC *.UNSMRY

Notes:

1) A binary file is computer-readable but not human-readable. All executable programs are stored in binary files, as are most numeric data files. In contrast, text files are stored in a form (usually ASCII) that is human-readable.
2) For unified files if the run terminates unexpectedly, or there is sufficient disk space, then the last report output is not stored. Their main advantage is that if a number of simulations reside in one directory, their output is organized. There is no limit on the number of reporting steps that a unified file can store.

Table 5.24: MULTOUT Keyword Description

There is no data required for this keyword.

See also *OPM FLOW OUTPUT FILE FORMATS* for a more detailed description of the various file types (ASCII or binary) and file structure formats (unified or non-unified formats).

Example

```
--
--       ACTIVATE THE MULTIPLE OUTPUT FILES OPTION
--
MULTOUT
```

The above example switches on the multiple output file option.

5.3.83 MULTOUTS – ACTIVATE NON-UNIFIED MULTIPLE SUMMARY OUTPUT FILE OPTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword switches on the Multiple Output Files option for SUMMARY files only, and overwrites the UNIFOUT keyword in the RUNSPEC section that activates the Unified Output Files option for all output files.

There is no data required for this keyword and there is no terminating "/" for this keyword.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

See also *OPM FLOW OUTPUT FILE FORMATS* for a more detailed description of the various file types (ASCII or binary) and file structure formats (unified or non-unified formats).

Example

```
--
--       ACTIVATE MULTIPLE OUTPUT SUMMARY FILES ONLY OPTION
--
MULTOUTS
```

The above example switches on the multiple output file option.

5.3.84 MULTREAL – ACTIVATE COMMERCIAL SIMULATOR'S MULTI-REALIZATION LICENSE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The MULTREAL keyword activates the commercial simulator's Multi-Realization License option.

OPM Flow is an open source project and therefore there is no license management of the various implemented options; hence this keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

5.3.85 NETWORK – ACTIVATE THE EXTENDED NETWORK OPTION AND DEFINE PARAMETERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword activates the Extended Network option and defines the maximum number of nodes and links (branches) in the network. The Extended Network option is a different facility to the Standard Network facility, as such, this keyword should only be used if the former network is required for the run.

No.	Name	Description	Default
1	NODMAX	NODMAX is a positive integer that defines the maximum number of nodes in the Extended Network model.	None
2	NBRMAX	NBRMAX is a positive integer that defines the maximum number of links in the Extended Network model.	None
3	NBCMAX	NBCMAX is a positive integer that defines the maximum number of branches that can be connected to a node in the Extended Network model, used in the commercial compositional simulator. The parameter is ignored by OPM Flow and should be defaulted or set to the default value of 20.	20
Notes: 1) The keyword is terminated by a "/".			

Table 5.25: NETWORK Keyword Description

Example

```
--
--        ACTIVATE THE EXTENDED NETWORK OPTION AND DEFINE PARAMETERS
--
--        MAX.     MAX      NOT
--        NODE     LINK     USED
NETWORK
          10        12      1*                                          /
```

In the above example the maximum number of nodes is set equal to ten and the maximum number of links (or branches) is set equal to 12, for the Extended Network option.

5.3.86 NINEPOIN – ACTIVATE THE NINE-POINT DISCRETIZATION OPTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The NINEPOIN keyword activates the Nine-Point Discretization formulation for the whole grid. If the keyword is absent from the run then the conventional standard five-point discretization formulation is used for the model. The nine-point scheme is based on adding additional non-neighbor connections between the diagonal neighbors in the areal plane, in order to reduce grid orientation effects[96].

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

In none Local Grid Refinement runs the NINENUM keyword in the GRID section may be use to optionally set parts of the grid to use nine-point discretization and the remaining regions to use the conventional standard five-point discretization formulation.

There is no data required for this keyword and there is no terminating "/" for this keyword.

Example

```
--
--       ACTIVATE THE NINE-POINT DISCRETIZATION OPTION
--
NINEPOIN
```

The above example switches on the Nine-Point Discretization option for the whole grid.

[96] *Yanosik, J. L. and McCracken, T. A. "A Nine-Point, Finite-Difference Reservoir Simulator for Realistic Prediction of Adverse Mobility Ratio Displacements," paper SPE 5734, Society of Petroleum Engineers Journal (1979) 19, No. 4, 253-262.*

5.3.87 NMATRIX – ACTIVATE THE DISCRETIZED MATRIX DUAL POROSITY OPTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The NMATRIX keyword activates the Discretized Matrix Dual Porosity option and specifies the number of sub-grid blocks in the actual matrix grid blocks. See also the NMATOPS keyword in the GRID section that defines various parameters for this option.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description	Default
1	NMATRIX	A positive integer value that specifies the number of sub-grid blocks in the actual matrix grid blocks.	1
Notes: 1) The keyword is terminated by a "/".			

Table 5.26: NMATRIX Keyword Description

Note the keyword cannot be used in conjunction with the TRPLPORO keyword, which is also in the RUNSPEC section.

Example

```
--
--        SUB-GRIDS
--        NMATRIX
NMATRIX
          4                                                    /
```

The above example activates the Discretized Matrix Dual Porosity option and specifies the number of sub-grid blocks in the actual matrix grid block to be four.

5.3.88 NNEWTF – ACTIVATE THE NON-NEWTONIAN FLUID MODEL

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword activates the Non-Newtonian Fluid phase and model for when the polymer phase is present in the model, as indicated by the POLYMER keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description	Default
1	NTHRBL	A positive integer that defines the maximum number of Herschel-Bulkley versus polymer concentration tables to be used with the polymer model, as entered via the FHERCHBL keyword in the PROPS section. The tables are allocated to different parts of the grid by the HBNUM keyword in the REGION section	NTPVT
2	NLNHBL	A positive integer that defines the maximum number of rows for each table entered by the FHERCHBL keyword in the PROPS section.	2

Notes:

1) The exact number of NTHRBL tables are required to completed the data set. For example, if NTHRBL is set equal to three, then there must be three tables entered for the FHERCHBL keyword.
2) The keyword is terminated by a "/".

Table 5.27: NNEWTF Keyword Description

Example

```
--
--        MAX      MAX
--        NTHRBL   NLNHBL
NNEWTF
           3       5                                                    /
```

The above example defines maximum number of Herschel-Bulkley tables to be three with a maximum number of rows for each table set to five.

5.3.89 NOCASC – ACTIVATE LINEAR SOLVER TRACER ALGORITHM

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

NOCASC keyword activates the linear solver tracer algorithm for single phase tracers.

OPM Flow uses a different numerical scheme which makes this keyword redundant; hence, OPM Flow ignores this keyword. It is documented here for completeness.

There is no data required for this keyword and there is no terminating "/" for this keyword.

Example

```
--
--        TRACER SOLVER OPTION
--
NOCASC
```

The above example switches on the linear solver tracer algorithm; however, this has no effect in OPM Flow input decks.

5.3.90 NODPPM – DEACTIVATE FRACTURE POROSITY-PERMEABILITY CALCULATION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The NODPPM keyword deactivates the default behavior of multiplying the fracture porosity by the fracture permeability to calculate the effective fracture permeability in dual porosity and dual permeability runs. Either the DUALPORO or DUALPERM keywords in the RUNSPEC section must be declared in the input file in order to use this keyword. If the default calculation is switched off by this keyword, then the effective fracture permeability is taken to be those entered for the fracture using the PERMX, PERMY and PERMZ keywords in the GRID section. If the keyword is absent from the input deck, then the entered PERMX, PERMY and PERMZ arrays for the fractures are multiplied by fracture PORO array values in order to obtain the effective fracture permeability.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

There is no data required for this keyword and there is no terminating "/" for this keyword.

Example

```
--
--       DEACTIVATE FRACTURE POROSITY-PERMEABILITY CALCULATION
--
NODPPM
```

The above example switches off the default behavior of multiplying the fracture porosity by the fracture permeability to calculate the effective fracture permeability.

5.3.91 NOECHO – DEACTIVATE ECHOING OF USER INPUT FILES TO THE PRINT FILE

Turns off echoing of all the input files to the print file. Note by default echoing of the inputs files is active. but can subsequently be switched off by the NOECHO activation keyword.

See *NOECHO – Deactivate Echoing of User Input Files to the Print File* in the GLOBAL section for a full description.

5.3.92 NOHYST - DEACTIVATE THE HYSTERESIS OPTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The NOHYST keyword deactivates the Hysteresis option and informs the simulator to ignore the IMBNUM array in the REGIONS section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

There is no data required for this keyword and there is no terminating "/" for this keyword.

Example

```
--
--       DEACTIVATE THE HYSTERESIS OPTION
--
NOHYST
```

The above example switches off the default behavior of multiplying the fracture porosity by the fracture permeability to calculate the effective fracture permeability.

5.3.93 NOINSPEC – DEACTIVATE OUTPUT OF THE INIT INDEX FILE

Description

The NOINSPEC keyword deactivates the writing out of the INIT index file (*.INSPEC). The initialization data (or static data) is written out to two files one file contains the data, *.INIT, and the second file contains an index of the data (*.INSPEC) stored in the *.INIT file. This functionality is redundant as most post-processing software require the *.INSPEC file to load the *.INIT data set.

Hence, OPM Flow ignores this keyword. It is documented here for completeness.

There is no data required for this keyword and there is no terminating "/" for this keyword.

Example

```
--
--       DEACTIVATE OUTPUT OF THE INIT INDEX FILE *.INSPEC
--
NOINSPEC
```

The above example switches off the writing of the INIT index file (*.INSPEC); however, this has no effect in OPM Flow input decks.

5.3.94 NOMONITO – DEACTIVATE OUTPUT OF THE MONITORING DATA AND FILE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The NOMONITO keyword deactivates the writing out of the run time monitoring information used by post-processing graphics software to display run time information, for example the simulated production and injection rates and cumulative values. OPM Flow does not have this functionality.

Hence, OPM Flow ignores this keyword but it is documented here for completeness.

There is no data required for this keyword and there is no terminating "/" for this keyword.

Example

```
--
--       DEACTIVATE MONITORING OUTPUT DATA AND FILES
--
NOMONITO
```

The above example switches off the output required for run time monitoring required by post-processing graphics software to review the simulation results in real time as the run progresses; however, this has no effect in OPM Flow input decks.

5.3.95 NONNC – DEACTIVATE NON-NEIGHBOR CONNECTIONS

Description

The NONNC keyword deactivates non-neighbor connections ("NNCs") in the current run. NNCs create off-diagonal elements in the Jacobi matrix that impact the numerical efficiency of the solution algorithms, and thus if the run does not contain NNC's then there is the potential for greater computation efficiency. Unfortunately, nearly all models, except for the most simple models, generate NNCs via for example:

1) aquifer connections,
2) faults, and
3) manually entered NNCs, including those automatically generate by pre-processing software.

Due to the limited application of this option, the feature has not been implemented in OPM Flow and hence OPM Flow ignores this keyword. It is documented here for completeness.

There is no data required for this keyword and there is no terminating "/" for this keyword.

Example

```
--
--       DEACTIVATE NON-NEIGHBOR CONNECTIONS
--
NONNC
```

The above example switches off the NNCs; however, this has no effect in OPM Flow input decks.

5.3.96 NORSSPEC – DEACTIVATE OUTPUT OF THE RESTART INDEX FILE

Description

The NORSSPEC keyword deactivates the writing out of the RESTART index file (*.RSSPEC). The restart data (pressure, saturations etc. through time for each active cell) are written out to two files one file contains the data, *.UNRST for example, and the second file contains an index of the data (*.RSSPEC) stored in the *.UNRST file. This functionality is redundant as most post-processing software require the *.RSSPEC file to load the *.UNRST data set.

Hence, OPM Flow ignores this keyword. It is documented here for completeness.

There is no data required for this keyword and there is no terminating "/" for this keyword.

Example

```
--
--       DEACTIVATE OUTPUT OF THE RESTART INDEX FILE *.RSSPEC
--
NORSSPEC
```

The above example switches off the writing of the restart index file (*.RSSPEC); however, this has no effect in OPM Flow input decks.

5.3.97 NOSIM – ACTIVATE THE NO SIMULATION MODE FOR DATA FILE CHECKING

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

NOSIM switches the mode of OPM Flow to data input checking mode. In this mode the input file is read and all messages and print instructions are sent to the respective output files. The SCHEDULE section is read but the simulation is not performed.

There is no data required for this keyword and there is no terminating "/" for this keyword.

Example

The example below switches OPM Flow to no simulation mode for data checking of the input deck.

```
--
--       SWITCH NO SIMULATION MODE FOR DATA CHECKING COMMENT OUT TO RUN THE MODEL
--
NOSIM
```

And the next example shows how to commented out the NOSIM activation keyword so that the simulation will proceed.

```
--
--       SWITCH NO SIMULATION MODE FOR DATA CHECKING COMMENT OUT TO RUN THE MODEL
--
-- NOSIM
```

Note

Simulation input decks are complex and are therefore prone to typing errors, thus before submitting a run that will take over 15 minutes or so, it is a good idea to run the model with the NOSIM option. If no errors are found then the NOSIM keyword should be commented out by placing "--" before the keyword, and then re-running the model.

Alternatively, one could use OPMRUN to run all the jobs in the queue in NOSIM mode and have the software re-run jobs in simulation mode if there are no errors.

5.3.98 NOWARN – DEACTIVATE WARNING MESSAGES

Turns off warning messages to be printed to the print file; note that this keyword is deactivated by default and can subsequently be switched off by the WARN activation keyword. The warning messages may be turned on and off using keywords WARN and NOWARN.

See *NOWARN – Deactivate Warning Messages* in the GLOBAL section for a full description.

5.3.99 NRSOUT – DEFINED MAXIMUM NUMBER OF RESTART ELEMENTS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The NRSOUT keyword specifies the maximum number of elements that can be written to the RESTART file at each reporting time step.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description	Default
1	NRSOUT	A positive integer value that specifies the maximum number of elements that can be written to the RESTART file at each reporting time step.	3600

Notes:

1) The keyword is terminated by a "/".

Table 5.28: NRSOUT Keyword Description

Example

```
--
--        MAX
--        NRSOUT
NRSOUT
          6000                                                    /
```

The above example sets the maximum number of elements that can be written to the RESTART file at each reporting time step to 6000.

5.3.100 NSTACK – DEFINE THE STACK LENGTH FOR THE ITERATIVE LINEAR SOLVER

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The NSTACK keyword defines the maximum number of previous search directions stored by the linear solver. Increasing the value of NSTACK may improve the efficiency of the solver on difficult problems, but will increase the memory requirements of the simulator. The default value of 10 should be sufficient for most problems; however, if OPM Flow is having issues with the convergence of the linear questions then increasing NSTACK and LITMAX on the TUNING keyword may improve performance.

OPM Flow uses a different numerical scheme which makes this keyword redundant; see section *2.2 Running OPM Flow 2023-10 From The Command Line* on how to invoke various numerical schemes via the OPM Flow command line interface.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

No.	Name	Description	Default
1	NSTACK	A positive integer that defines the maximum number of previous search directions stored by the linear solver.	10

Notes:

1) NSTACK and LITMAX on the TUNING keyword are related such that NSTACK should always be less than or equal to LITMAX.

2) The keyword is terminated by a "/".

Table 5.29: NSTACK Keyword Description

Example

```
--
--          SET STACK SIZE FOR LINEAR SOLVER
--
NSTACK
            30                                                              /
```

The above example sets maximum number of previous search directions stored by the linear solver to 30, this has no effect in OPM Flow input decks.

Note

If the run is suffering from linear convergence problems, then check the data first for any data issues before manipulating the numerical control parameters. For example, if OPM Flow has written some WARNING messages with respect to end-point scaling, etc., then resolve these messages first before adjusting the numerical controls.

5.3.101 NUMRES – DEFINE THE NUMBER OF RESERVOIR GRIDS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The NUMRES keyword defines the number of reservoir grids (COORD data sets) that the simulator should process. Currently, this should only be set to one in OPM Flow

No.	Name	Description	Default
1	NUMRES	A positive integer greater than one that defines the maximum number COORD data sets to be processed by OPM Flow. This should be set to one.	1
Notes: 1) The keyword is terminated by a "/".			

Table 5.30: NUMRES Keyword Description

Example

```
--
--       DEFINE THE NUMBER OF RESERVOIR GRIDS (COORD DATA SETS)
--
NUMRES
         1                                                                  /
```

The above example sets the maximum number of COORD data sets to be processed to one, this is the only value that can currently be used in OPM Flow.

5.3.102 NUPCOL – DEFINE THE MAXIMUM NUMBER OF NEWTON ITERATIONS USED TO UPDATE WELL TARGETS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The NUPCOL keyword defines the maximum number of Newton iterations within a time step that may be used to update the well production and injection targets, after which the well targets will be frozen until the time step calculations have converged and the time step is complete.

Wells under group control may suffer from some dependency with other wells in the same group that are under group control. This may cause some oscillation in the production and injection well rates within the group. In order to avoid this, after the number Newton iterations within a time step surpasses NUPCOL, the group well rates are frozen until the time step has converged. Reducing the potential of well rate oscillations within the time step may result in the group targets and limits not being exactly met in this case. Increasing the value of NUPCOL to greater than the default value of six, will improve the accuracy of the group targets and limits at the expense of computational efficiency.

See also section *2.2 Running OPM Flow 2023-10 From The Command Line* on how to set various other numerical control parameters for OPM Flow.

No.	Name	Description	Default
1	NUPCOL	A positive integer that defines the maximum number of Newton iterations used to update well targets within a time step. Note that default value of six is different to the commercial simulator's default value of three.	6
Notes: 1) The keyword is terminated by a "/".			

Table 5.31: NUPCOL Keyword Description

See also the MINNPCOL keyword in the RUNSPEC section that sets the minimum number of Newton iterations within a time step that may be used to update the well production and injection targets, after which the well targets will be frozen until the time step calculations have converged and the time step is complete. Note that MINNPCOL is an OPM Flow specific keyword.

Example

```
--
--       DEFINE THE MAX NUMBER OF ITERATIONS TO UPDATE WELL FLOW TARGETS
--
NUPCOL
        6                                                                  /
```

The above example sets the default NUPCOL value to the default value of six.

5.3.103 OIL – ACTIVATE THE OIL PHASE IN THE MODEL

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword indicates that the oil phase is present in the model and must be used for oil-gas, oil-water, oil-water-gas input decks that contain the oil phase. The keyword will also invoke data input file checking to ensure that all the required oil phase input parameters are defined in the input deck.

There is no data required for this keyword and there is no terminating "/" for this keyword.

Example

```
--
--       OIL PHASE IS PRESENT IN THE RUN
--
OIL
```

The above example declares that the oil phase is active in the model.

5.3.104 OPTIONS – ACTIVATE VARIOUS PROGRAM OPTIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The OPTIONS keyword activates various program options in the commercial simulator. Currently, none of the options available in the commercial simulator are implemented in OPM Flow, and it is unlikely that this keyword will be supported in the future releases of OPM Flow.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Description	Default
1 - 273	Commercial simulator options keyword, used to to switch on or off specific features. The keyword is commonly used to revert the simulator's behavior to past functionality that has been depreciated in the current version of the commercial simulator, for backward compatibility with previous models.	0

Notes:

1) The keyword is terminated by a "/".

Table 5.32: OPTIONS Keyword Description

Examples

```
--
--        SKIP       ACTIVATE
--        OPTIONS    OPTION
OPTIONS
          77*0       1                                                  /
```

The above example activates the use of scratch files for pre-processing grid geometry data for non-neighbor connections. Note if multiple options are required then one can just repeat the format of the example to activate multiple options as the keyword does not overwrite previous entries. So for example:

```
--
--        SKIP       ACTIVATE
--        OPTIONS    OPTION
OPTIONS
           7*0       1                                                  /
--
--        SKIP       ACTIVATE
--        OPTIONS    OPTION
OPTIONS
          77*0       1                                                  /
--
--        SKIP       ACTIVATE
--        OPTIONS    OPTION
OPTIONS
          177*0      1                                                  /
```

Could be used to activate the 8, 78 and 178 options if they were available.

5.3.105 PARALLEL – DEFINE PARALLEL RUN CONFIGURATION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The PARALLEL keyword defines the run to use parallel processing and sets the domain decomposition options. See section *2.2 Running OPM Flow 2023-10 From The Command Line* on how to run OPM Flow in parallel mode.

No.	Name	Description	Default
1	NPROCS	A positive integer that defines the number of domains or parallel processors to use for this run.	1
2	RTYPE	A character string set to either SERIAL to run the parallel code in serial mode for testing the code, or DISTRIBUTED to full utilize parallel processing.	PARALLEL
Notes: 1) The keyword is terminated by a "/".			

Table 5.33: PARALLEL Keyword Description

OPM Flow uses a different numerical scheme which makes this keyword redundant; hence, OPM Flow ignores this keyword. It is documented here for completeness.

Example

```
--
--       PARALLEL MULTI-CORE OPTIONS
--       NDMAIN     MACHINE TYPE
PARALLEL
           2        DISTRIBUTED                                              /
```

The above example sets the number of domains (or processors) to two and for the simulation to run in parallel mode. This has no effect in OPM Flow input decks.

5.3.106 PARTTRAC – ACTIVATE AND DEFINE PARTITIONED TRACER OPTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The PARTTRAC keyword activates the Partitioned Tracer option and defines the maximum number of partitioned tracers, the number of TRACERKP or TRACERKM partitioning tables in the PROPS section, and the maximum number of number of rows in the TRACERKP or TRACERKM partitioning tables.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

5.3.107 PATHS – DEFINE FILENAME DIRECTORY PATH ALIASES

Description

PATHS allows the user to define alias directory filenames to avoid long filenames with the INCLUDE, IMPORT, RESTART or GDFILE keywords. To use the alias with the aforementioned keywords PATHS should be prefixed with the $ symbol.

No.	Name	Description	Default
1	ALIAS	A character string enclosed in quotes defining the alias.	None
2	DIRC	A character string enclosed in quotes defining the directory filename.	None

Notes:

1) Multiple entries must be on separate lines (see the example).
2) The keyword is terminated by a "/".

Table 5.34: PATHS Keyword Description

Examples

```
--
--       PATH       PATH
--       ALIAS      DIRECTORY FILENAME
PATHS
        'GRID'      '/DISK1/NORNE/2017/GRID-INCLUDES'                    /
        'SCHD'      '/DISK1/NORNE/2017/SCHD-INCLUDES'                    /
/
```

The above example defines "GRID" and "SCHD" aliases in the RUNSPEC section than can be used in the GRID and SCHEDULE sections of the input deck. The next example shows how to use the "GRID" alias with the INCLUDE keyword in the GRID section.

```
--
--       LOAD INCLUDE FILES
--
INCLUDE
         '$GRID/PORO.INC'                                                /

INCLUDE
         '$GRID/PERMX.INC'                                               /

INCLUDE
         '$GRID/NTG.INC'                                                 /
```

Here the porosity, permeability and net-to-gross arrays are loaded in the GRID section using the directory filename aliases declared in the RUNSPEC section.

5.3.108 PEDIMS – DEFINE PETRO-ELASTIC MODEL REGIONS AND TABLE DIMENSIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The PEDIMS keyword defines the number of petro-elastic regions to be used with the PENUM keyword in the REGIONS section and the number of rows in the PEGTAB0 to PEGTAB7 keywords, as well as the number of rows in the PEKTAB0 to PEKTAB7 keywords in the PROPS section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

5.3.109 PETOPTS – DEFINE PETREL AND GENERIC SIMULATION FILE OPTIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The PETOPTS keyword defines various Petrel and Generic Simulation (*.GSG) file options.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

5.3.110 PIMTDIMS – DEFINE WELL PRODUCTIVITY SCALING TABLE DIMENSIONS

Description

PIMTDIMS keyword defines the maximum number of PIMULTAB tables and the maximum number of entries (or rows) per PIMULTAB table. The PIMULTAB keyword is used to define a well's productivity index factor as a function of a well's producing water cut.

No.	Name	Description	Default
1	NTPIMT	A positive integer value that defines the maximum number of PIMULTAB keywords defined in the input deck.	0
2	NRPIMT	A positive integer value defining the maximum number of entries (rows) in the PIMULTAB keyword.	0
Notes: 1) The keyword is terminated by a "/".			

Table 5.35: PIMTDIMS Keyword Description

Example

```
--
--        MAX     MAX
--        TABLES  ENTRIES
PIMTDIMS
          1       51                                                    /
```

The above example defines that there is one PIMULTAB table with a maximum number of 51 rows.

5.3.111 PINTDIMS – DEFINE POLYMER MOLECULAR WEIGHT MODEL TABLE DIMENSIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The PINTDIMS keyword defines the number of property tables used in the OPM Flow's Polymer Molecular Weight Transport option, that uses the polymer molecular weight in calculating the polymer viscosity, as well as accounting for formation damage due to the water and polymer injection, by adjusting the wellbore skin pressure. This keyword should only be used if the POLYMER and POLYMW keywords in the RUNSPEC section are also activated. The PINTDIMS keyword defines the maximum number of tables for the SKPRWAT, SKPRPOLY, and PLYMWINJ keywords, and the number of entries in the PLYVMH keyword. All the aforementioned keywords are in the PROPS section.

Note

This is an OPM Flow specific keyword that employs an alternative polymer flood model based on a Polymer Molecular Weight Transport equation, that is not available in the commercial simulator.

No.	Name	Description	Default
1	NTSKWAT	NTSKWAT is a positive integer that defines the number of SKPRWAT tables in the PROPS section, used to describe the relationship of wellbore skin pressure as a function of water throughput and water velocity, for the simulator's Polymer Molecular Weight Transport option.	1
2	NTSKPOLY	NTSKPOLY is a positive integer that defines the number of SKPRPOLY tables in the PROPS section, used to describe the relationship of wellbore skin pressure as a function of polymer throughput and polymer velocity, for the simulator's Polymer Molecular Weight Transport option.	1
3	NTPMWINJ	NTPMWINJ is a positive integer that defines the number of PLYMWINJ tables in the PROPS section, used to describe the relationship of the injected polymer molecular weight as a function of polymer throughput and polymer velocity, for the simulator's Polymer Molecular Weight Transport option.	1
4	NPLYVMH	NPLYVMH is a positive integer that defines the maximum number of entries (rows) in the PLYVMH table in the PROPS section, used to describe the relationship of the injected polymer viscosity as a function of polymer molecular weight and polymer concentration, for the simulator's Polymer Molecular Weight Transport option.	1
Notes: 1) The keyword is terminated by a "/".			

Table 5.36: PINTDIMS Keyword Description

The SKPRWAT, SKPRPOLY, PLYMWINJ, and PLYVMH keywords in the PROPS section, are the additional keywords required for the Polymer Molecular Weight Transport option. Note that the standard polymer property data keywords: PLYROCK, PLYADS, PLYMAX, etc., are still required to fully describe the polymer fluid.

Example

```
--
--       POLYMER MOLECULAR WEIGHT TRANSPORT TABLES (OPM FLOW RUNSPEC KEYWORD)
--
--       NO.       NO.       NO.       NO.
--       NTSKWAT   NTSKPOLY  NTPMWINJ  NPLYVMH
PINTDIMS
         2         2         2         1                                     /
```

The above example declares two SKPRWAT, SKPRPOLY, and PLYMWINJ keywords in the PROPS section will be used, as well as the default value of one for the number of rows in the PLYVMH keyword.

5.3.112 POLYMER – ACTIVATE THE POLYMER PHASE IN THE MODEL

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword indicates that the polymer phase is present in the model and to activate the polymer flooding model. The keyword will also invoke data input file checking to ensure that all the required polymer phase input parameters are defined in the input deck.

There is no data required for this keyword and there is no terminating "/" for this keyword.

Example

```
--
--       ACTIVATE THE POLYMER PHASE IN THE MODEL
--
POLYMER
```

The above example declares that the polymer phase is active in the model.

5.3.113 POLYMW – ACTIVATE THE POLYMER MOLECULAR WEIGHT TRANSPORT OPTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword activates OPM Flow's Polymer Molecular Weight Transport option that uses the polymer molecular weight in calculating the polymer viscosity, as well as accounting for formation damage due to the water and polymer injection, by adjusting the wellbore skin pressure. This keyword should only be used if the POLYMER keyword in the RUNSPEC section is also activated. The keyword will also invoke data input file checking to ensure that all the required polymer phase input parameters are defined in the input deck.

In the PROPS section the PLYVMH, PLYMWINJ, SKPRWAT and SKPRPOLY keywords are used to define the additional polymer and water properties for the model. And the SPOLYMW keyword in the SOLUTION section may be used to set the initial molecular weight for each grid block in the model, in order to initialize the model. In the SCHEDULE section, the WPIMITAB keyword should be used to associate a water injection well with the appropriate PLYMWINJ table. Finally, the WSKPTAB keyword, also in the SCHEDULE section, may be used to define the SKPRPOLY and SKPRWAT tables associated with the skin pressure for polymer and water injection. All the aforementioned keywords are OPM Flow specific keywords, that are not available in the commercial simulator.

Note

This is an OPM Flow specific keyword that employs an alternative polymer flood model based on a Polymer Molecular Weight Transport equation, that is not available in the commercial simulator.

There is no data required for this keyword and there is no terminating "/" for this keyword.

Example

```
--
--       ACTIVATE THE POLYMER PHASE IN THE MODEL
--
POLYMER
--
--       ACTIVATE THE POLYMER MOLECULAR WEIGHT TRANSPORT OPTION
--
POLYMW
```

The above example declares that the polymer phase is present, and that the simulator's Polymer Molecular Weight Transport option should be used instead of the standard polymer model.

5.3.114 PRECSALT – ACTIVATE THE OPM FLOW SALT PRECIPITATION MODEL

Description

This keyword activates the OPM Flow Salt Precipitation model that accounts for salt precipitating out of the water phase when the water is being vaporized into the gas phase and the dissolved salt reaches the solubility limit as the pressure in the reservoir is being depleted (see the VAPWAT keyword in the RUNSPEC section). This facility is an extension to the standard Brine model, and as such the BRINE keyword in the RUNSPEC must also be present in the input deck. In general, if the PRECSALT keyword has been activated in the input deck then the VAPWAT keyword should also be activated. The keyword should only be used if both water and gas phases are active in the model.

Note

This is an OPM Flow specific keyword for the simulator's Salt Precipitation model, note that this is an extension to the commercial simulator's Brine model.

If the keyword is present in the input deck then the SALTSOL keyword in the PROPS section also needs to be present in the input deck to define the salt solubility. In addition, either the SALTPVD or SALTP keywords in the SOLUTION section should be used to define the initial salt precipitated saturation.

There is no data required for this keyword and there is no terminating "/" for this keyword.

Example

The first part of the example shows the keywords for the RUNSPEC section.

```
-- ==================================================================================
--
-- RUNSPEC SECTION
--
-- ==================================================================================
RUNSPEC
-- ----------------------------------------------------------------------------------
-- FLUID TYPES AND TRACER OPTIONS
-- ----------------------------------------------------------------------------------
--
--       OIL PHASE IS PRESENT IN THE RUN
--
OIL
--
--       WATER PHASE IS PRESENT IN THE RUN
--
WATER
--
--       GAS PHASE IS PRESENT IN THE RUN
--
GAS
--
--       DISSOLVED GAS IN LIVE OIL IS PRESENT IN THE RUN
--
DISGAS
```

```
--
--       ACTIVATE STANDARD BRINE MODEL IN THE RUN
--
BRINE
--
--       ACTIVATE THE OPM FLOW SALT PRECIPITATION MODEL (OPM FLOW KEYWORD)
--
PRECSALT
```

The above example declares that the oil, water, gas, dissolved gas, and brine phases are present in the model, and activates the Brine (BRINE keyword) and Salt Precipitation (PRECSALT keyword) models.

The second part of the example shows the additional PVT fluid keywords require for the Salt Precipitation model in the PROPS section. Note that in addition to the Salt Precipitation model specific keywords, the standard DENSITY (or GRAVITY), PVDG, and PVTO keywords are required.

```
-- ======================================================================
--
-- PROPS SECTION
--
-- ======================================================================
PROPS
--
--       WATER SALT PVT TABLE
--
PVTWSALT
--       REF PRES  REF SALT
--       PSIA      LB/STB
--       --------  --------
         3000.0    0.000                                  / REFERENCE DATA
--
--       SALTCONC  BW         CW        VISC     VISC
--       LB/STB    RB/STB     1/PSIA    CPOISE   GRAD
--       --------  --------   -------   ------   ------
            0.0    1.0        3.0E-6    1.0      1.0E-6
           10.0    0.8        3.0E-6    1.0      1.0E-6    / SALT CONCENTRATION
--
--       SET SALT SOLUBILITY LIMIT FOR ALL CELLS (OPM FLOW KEYWORD)
--
SALTSOL
--       MAX       SALT
--       SALTSOL   DENSITY
         0.500     135.46867445023383                     /
```

In addition, to the above the PERMFACT keyword may be used to account for the reduction in porosity and permeability due to salt precipitating into the pore space. Again, in addition to the Salt Precipitation model specific keywords, the standard ROCK, SGOF, and SWOF keywords are required.

```
--
--       PERMEABILITY FACTOR REDUCTION DUE TO SALT (OPM FLOW KEYWORD)
--
PERMFACT
--       PORO       PERM
--       FACTOR     FACTOR
--       -------    --------
         0.0         0.000
         0.4         0.005
         0.6         0.010
         0.9         0.100
         1.0         1.000                                  / TABLE NO. 01
/
```

In order to initialize the model in the SOLUTION section, apart from the standard equilibration keywords, the following Salt Precipitation model specific keywords should be included.

```
-- ======================================================================
--
-- SOLUTION SECTION
--
-- ======================================================================
SOLUTION
--
--       PRECIPITATED SALT VOLUME FRACTION VERSUS DEPTH (OPM FLOW KEYWORD)
--
--       DEPTH    SALTPSAT
--       ------   --------
SALTPVD
         8300.0    0.010
         8450.0    0.010                          / SALT VS DEPTH EQUIL REGN 01
--
--       DEPTH    SALT-1    SALT-2    SALT-3
--                SALTCON   SALTCON   SALTCON
--       ------   -------   -------   -------
SALTVD
         8300.0    0.500
         8450.0    0.500                           / SALT VS DEPTH EQUIL REGN 01
```

Note that OPM Flow does not support Multi-Component Brine model, thus there should only one column of salt concentrations.

Finally, in the SCHEDULE section, one may optionally one can add the injections well's injection salt concentrations, as shown below.

```
-- ======================================================================
--
-- SCHEDULE SECTION
--
-- ======================================================================
SCHEDULE
--
--       DEFINE WATER INJECTION WELL SALT CONCENTRATIONS
--
-- WELL  SALT-1     SALT-2     SALT-3     SALT-4
-- NAME  SALTCON    SALTCON    SALTCON    SALTCON
--       -------    -------    -------    --------
WSALT
'INJ'    0.0000                                        /
/
```

Normally, with this option one would also include vaporized water via the VAPWAT keyword in the RUNSPEC section, as well as the PVTGW keyword to define gas PVT properties for dry gas, or the PVTGWO keyword that declares the gas PVT properties for wet gas. Both keywords are in the PROPS

section. The RWGSALT keyword in the PROPS section, may be used to define the relationship of water vaporization versus pressure and salt concentration. Again, the keyword is in the PROPS section. Secondly, in the SOLUTION section, the RVW keyword can be used to set the initial equilibration vaporized water in gas ratio values for all grid cells in the model for Enumeration Initialization, or the RVWVD keyword that declares the vaporized water-gas ratio (Rvw) versus depth tables for each equilibration region, for Equilibrium Initialization.

5.3.115 PSTEADY – ACTIVATE PSEUDO STEADY STATE FLOW CALCULATION OPTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The PSTEADY keyword activates Pseudo Steady State Flow Calculation option by advancing the simulator until it reaches a pseudo steady state flow and then sets the date to the date defined on this keyword, that is written to the RESTART file. Keyword also includes parameters defining the conditions for pseudo steady flow state.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

5.3.116 RADIAL – RADIAL GRID ACTIVATION OPTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

RADIAL activates the radial grid geometry option for the model, if this keyword and the SPIDER keyword are omitted then Cartesian geometry is assumed by OPM Flow.

See also the SPIDER keyword in the RUNSPEC section (*SPIDER – Spider Grid Activation Option*). This is an OPM Flow specific keyword for the simulator's Spider Grid option, that emulates a Radial Grid via corner-point geometry. Both Spider and Radial grids will be displayed the same in OPM ResInsight, as they both use Irregular Corner-Point Grid geometry. The difference is that the radial model's pore volumes are adjusted to match the radial geometry pore volumes, whereas, the Spider grid volumes are not adjusted. See the examples in the section on Radial Grids (*Radial Grids*) and Spider Grids (*Spider Grids*) for further information.

Example

```
--
--       DEFINE RADIAL GRID GEOMETRY
--
RADIAL
```

The example activates the radial grid geometry option.

5.3.117 REGDIMS – DEFINE THE MAXIMUM NUMBER OF REGIONS FOR A REGION ARRAY

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The REGDIMS keyword defines the maximum number of regions for various region arrays used in the model. Note that the maximum number of FIPNUM regions can be defined both on this keyword and the TABDIMS keyword, if it set in both locations the maximum value is used. The reason for this type of inconsistency is due to the commercial simulator evolving with time as new features were added, but at the same time having to maintain backward input deck compatibility.

No.	Name	Description	Default
1	NTFIP	A positive integer defining the maximum number of regions in the FIPNUM region array. If additional sets of fluid in-place regions have been defined, as per the FIPxxxxx series of fluid in-place region keywords, then NTFIP should be set to the maximum number of regions in either the FIPNUM or FIPxxxxx associated arrays. Thus, if the maximum number of regions in the FIPNUM array is 12, and the maximum value in the FIPxxxxx series of arrays is 20, then 20 should be entered for NTFIP. Note that this parameter may also be set on the TABDIMS keyword as well. If NTFIP is set in both places, then the maximum value is used.	1
2	NMFIPR	A positive integer defining the total maximum number of fluid in-place regions. The number of FIPNUM regions are defined by NTFIP. However, if additional sets of fluid in-place regions are required, as per the FIPxxxxx series of fluid in-place region keywords, then these are to be defined here by adding the number of FIPxxxxx arrays to the value NTFIP. So for example, if NTFIP equals five and the number of distinct FIPxxxxx regions is three, then the value to enter for NMFIPR is eight.	1
3	NRFREG	A positive integer defining the maximum number of independent reservoir regions in the ISOLNUM region array.	0
4	MXNFLX	A positive integer defining the maximum number of flux regions in the FLUXNUM region array. MXNFLN can also be defined on the TABDIMS keywords as well. If MXNFLX is defined both here and on the TABDIMS keyword then the maximum value of the two is used.	0
5	NUSREG	A positive integer defining the maximum user defined regions in a commercial simulator's compositional model. This parameter is included for compatibility and should be defaulted as it is not used in OPM Flow.	0
6	NTCREG	A positive integer defining the maximum number of regions in the COALNUM region array. This parameter is included for compatibility and should be defaulted as it is not used in OPM Flow.	1
7	NOPREG	A positive integer defining the maximum number of regions in the OPERNUM region array.	0

No.	Name	Description	Default
8	NWKDREG	A positive integer defining the maximum number of real double-precision work arrays for use with the OPERATE and OPERATER keywords. This parameter is included for compatibility and should be defaulted as it is not used in OPM Flow.	0
9	NWKIREG	A positive integer defining the maximum number of integer work arrays for use with the OPERATE and OPERATER keywords. This parameter is included for compatibility and should be defaulted as it is not used in OPM Flow.	0
10	NPLMIX	A positive integer defining the maximum number of regions in the PLMIXNUM region array.	1
Notes: 1) The keyword is terminated by a "/".			

Table 5.37: REGDIMS Keyword Description

Example

```
--
--        MAX     TOTAL   INDEP   FLUX    TRACK   CBM     OPERN   WORK    WORK    POLY
--        FIPNUM  REGNS   REGNS   REGNS   REGNS   REGNS   REGNS   REAL    INTG    REGNS
REGDIMS
          9       12      1*      1*      1*      1*      1*      1*      1*      1*    /
```

The above example defines the number of FIPNUM regions to be nine and the number of FIPxxx type of regions to be three (12 – 9), the rest of the region sizes are set to the default values.

5.3.118 RIVRDIMS – DEFINE THE RIVER DIMENSIONS AND ASSOCIATED DATA

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

RIVRDIMS defines the river system array dimensions used with the REACHES keyword and other river keywords in the SOLUTION and SCHEDULE sections. The keyword also enables the River option.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

5.3.119 ROCKCOMP – ACTIVATE ROCK COMPACTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The ROCKCOMP keyword activates rock compaction and defines various rock compaction options for the run. By default OPM Flow models rock compaction via pore volume compressibility as entered on the ROCK keyword in the PROPS section. This keyword enables pressure dependent pore volume and transmissibility multipliers for rock compaction that are entered in the PROPS section using the ROCKTAB keyword. Currently OPM Flow only supports the default options for rock compaction.

No.	Name	Description	Default
1	ROCKOPT	A character string that defines the rock compaction option based on one of the following character strings: 1) REVERS: Rock compaction is reversible with increasing pressure. The rock compaction multipliers should be entered via the ROCKTAB keyword in the PROPS section. This is the default value. 2) IRREVERS: Rock compaction is irreversible, that is the rock expansion does not occur when the pressure subsequently increases. 3) HYSTER: Invokes the hysteresis rock compaction option. 4) BOBERG: Rock compaction hysteresis is modeled using the Boberg formulation[97]. 5) REVLIMIT: Activates the reversible hysteresis rock compaction option that limits the pore volume subject to reversibility based on the minimum pressure in a grid block and the initial water saturation. This option is only intended to be used with the water induced compaction model, neither of which are currently supported by OPM Flow.. 6) PALM-MAN: Rock compaction hysteresis is modeled using the Palmer-Mansoori[98] and [99] formulation for coal bed methane reservoirs, neither of which are supported by OPM Flow. 7) NONE: Deactivates rock compaction, unless the water induced compaction model has been invoked. Only the REVERS and IRREVERS options are supported by OPM Flow.	REVERS
2	NTROCC	A positive integer that defines the number of rock compaction tables, that is the number of ROCKTAB tables to be used by OPM Flow.	1
3	WATINOPT	A character string that states if the water induced rock compaction option should be used (YES) or not (NO). If set to YES then either the ROCKTABW or the ROCK2D and ROCKWNOD keywords should be entered in the PROPS section.	NO

[97] *Beattie, C.I., Boberg, T.C., and McNab, G.S. "Reservoir Simulation of Cyclic Steam Stimulation in the Cold Lake Oil Sands," paper SPE 18752, Society of Petroleum Engineers Journal, (1991) 6, No. 2, 200-206.*

[98] *Palmer, I. and Mansoori, J. "How Permeability Depends on Stress and Pore Pressure in Coalbeds: A New Model," paper SPE 52607, SPE Reservoir Evaluation & Engineering (1998) 1, No. 6, 539-544.*

[99] *Clarkson, C.R., Pan, Z., Palmer, I. and Harpalani, S. "Predicting Sorption-Induced Strain and Permeability Increase With Depletion for Coalbed-Methane Reservoirs", SPE 114778-PA, SPE Journal (2010) 15, No. 1, 152–159.*

No.	Name	Description	Default
4	PORTXROP	A character string that specifies the model to be used for when transmissibility is dependent on porosity, and should be set to either: 1) EXP: An exponential porosity-transmissibility relationship should be used. 2) CZ: The Carmen-Kozeny[100, 101] and [102] porosity-transmissibility relationship should be used. This option is used in the commercial compositional simulator and is therefore ignored by OPM Flow.	1*
5	CARKZEXP	The exponent constant in the Carmen-Kozeny porosity-transmissibility equation for when PORTXROP has been set to CZ. This option is used in the commercial compositional simulator and is therefore ignored by OPM Flow.	0.0
Notes: 1) The keyword is terminated by a "/".			

Table 5.38: ROCKCOMP Keyword Description

Example

```
--
--        ROCK    NUMBER   WAT      POR-TRAN
--        OPTN    TABLES   INDUCE   OPTION
ROCKCOMP
          REVERS  5        NO       1*                                    /
```

The above example defines the default values for the ROCKCOMP keyword with five rock compaction tables.

[100] *J. Kozeny, "Ueber kapillare Leitung des Wassers im Boden." Sitzungsber Akad. Wiss., Wien, 136(2a): 271-306, 1927.*

[101] *P.C. Carman, "Fluid flow through granular beds." Transactions, Institution of Chemical Engineers, London, 15: 150-166, 1937.*

[102] *P.C. Carman, "Flow of gases through porous media." Butterworths, London, 1956*

5.3.120 RPTCPL – ACTIVATE COUPLE SIMULATION REPORTING

FRUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword activates the couple simulation reporting, that results in the simulator writing out various initialization data and simulation data in order for external "controlling programs" to interactively manage the simulation. There is no data required for this keyword but the keyword should be terminated by a "/".

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

Example

```
--
--       ACTIVATE COUPLE SIMULATION REPORTING
--
RPTCPL
/
```

The above example switches on couple simulation reporting; however, this has no effect in OPM Flow input decks.

5.3.121 RPTHMD - DEFINE WELL HISTORY MATCH GRADIENT REPORTING OPTIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, RPTHMD, defines the options and level of history match output that should be written to history match file (*.HMD), for when the History Match Gradient option has been activated by the HMDIMS keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

5.3.122 RPTRUNSP – ACTIVATE RUNSPEC REPORTING

Description

This keyword activates reporting of all the RUNSPEC options utilized in the run. There is no data required for this keyword.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

Example

```
--
--       ACTIVATE RUNSPEC SECTION REPORTING
--
RPTRUNSP
```

The above example switches on RUNSPEC reporting; however, this has no effect in OPM Flow input decks.

5.3.123 RSSPEC – ACTIVATE OUTPUT OF THE RESTART INDEX FILE

Description

The RSSPEC keyword activates the writing out of the RESTART index file (*.RSSPEC). The restart data (pressure, saturations etc. through time for each active cell) are written out to two files one file contains the data, *.UNRST for example, and the second file contains an index of the data (*.RSSPEC) stored in the *.UNRST file. This keyword is somewhat redundant as the RESTART index file is written out by default. See the NORSSPEC keyword in the RUNSPEC section that deactivates the writing out of the file.

Hence, OPM Flow ignores this keyword. It is documented here for completeness.

There is no data required for this keyword and there is no terminating "/" for this keyword.

Example

```
--
--       ACTIVATE OUTPUT OF THE RESTART INDEX FILE *.RSSPEC
--
RSSPEC
```

The above example switches on the writing of the restart index file (*.RSSPEC); however, this has no effect in OPM Flow input decks.

5.3.124 RUNSPEC - DEFINE THE START OF THE RUNSPEC SECTION OF KEYWORDS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The RUNSPEC activation keyword marks the start of the RUNSPEC section that defines the key parameters for the simulator including the dimensions of the model, phases present in the model (oil, gas and water for example), number of tables for a given property and the maximum number of rows for each table, the maximum number of groups, wells and well completions, as well as various options to be invoked by OPM Flow.

Apart from COMMENTS entered by "--" in columns one and two, this keyword should be the first keyword in the input deck.

There is no data required for this keyword and there is no keyword terminating "/"

Example

```
-- ================================================================================
--
-- RUNSPEC SECTION
--
-- ================================================================================
RUNSPEC
```

The above example marks the start of the RUNSPEC section in the OPM Flow data input file.

5.3.125 SAMG – ACTIVATE ALGEBRAIC MULTI-GRID LINEAR SOLVER

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword actives the algebraic multi-grid linear solve; note this solver is not available to the general public in the commercial simulator.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

See section *2.2 Running OPM Flow 2023-10 From The Command Line* on how to invoke various numerical schemes via the OPM Flow command line interface.

5.3.126 SATOPTS – ACTIVATE RELATIVE PERMEABILITY ASSIGNMENT OPTIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

SATOPTS keyword activates OPM Flow's relative permeability assignment options. The relative permeability functions are defined using the either the:

1) SWOF, SGOF, SLGOF series of saturation functions, or the
2) SWFN, SGFN, SGWFN, SOF2, SOF3, SOF32D series of functions.

The allocation of the relative permeability tables to the grid cells is dependent on the options selected on this keyword (SATOPTS).

If the DIRECT option as been activated and the IRREVERS has not been invoked on the SATOPTS keyword, then different relative permeability functions are used for each x, y, and z directions. Here the KRNUMX, KRNUMY and KRNUMZ keywords are used for Cartesian grids to allocate the relative permeability tables to the cells. For Radial grids the KRNUMR, KRNUMT and KRNUMZ keywords should be used. This results in the same relative permeability curves being used in both the xi to xi – 1 and the xi to xi + 1 flow directions. Similarly for the y direction the same curves are used for the yi to yi – 1 and the yi to yi + 1 flow directions. And again for the z direction, the same relative permeability function is used for flow in the zi to zi – 1 and the zi to zi + 1 flow directions.

If the DIRECT option as been activated and the IRREVERS has been invoked on the SATOPTS keyword, then KRNUMX, KRNUMY and KRNUMZ keywords are used for Cartesian grids to allocate the relative permeability tables in the xi to xi + 1, yi to yi + 1, zi to zi + 1, flow directions, respectively. For Radial grids the KRNUMR, KRNUMT and KRNUMZ keywords should be used. For flow in the xi to xi – 1 flow directions, etc., the KRNUMX-, KRNUMY- and KRNUMZ- keywords are used for Cartesian grids and the KRNUMR-, KRNUMT- and KRNUMZ- are used for radial grids.

The HYSTER option activates the relative permeability hysteresis option of the non-wetting phases (liquid and vapour) may show hysteretic behavior in that their values depend on whether the non-wetting phase saturation is increasing or decreasing. For this option the user specifies two sets of saturation functions, one for a drainage process (decreasing wetting phase saturation) and one for an imbibition process (increasing wetting phase saturation). For a process starting at the maximum wetting phase saturation, on the drainage curve, with the wetting phase saturation decreasing, the drainage curve is followed. Similarly for a process starting at the minimum wetting phase saturation with the wetting phase saturation increasing, the imbibition curve is followed. If the drainage or imbibition process is reversed at some point, then the data does not necessarily run back over its previous values. In OPM Flow the Carlson[103] is used to describe relative permeability hysteresis and the Killough[104] model is used for capillary pressure.

If the DIRECT option as been activated and the IRREVERS has not been invoked on the SATOPTS keyword, then the same set are keywords as for the DIRECT only option are used to assign the drainage relative permeability curves, that is: KRNUMX, KRNUMY, etc., plus the IMBNUMX, IMBNUMY, and IMBNUMZ, keywords for the imbibition curves. If the DIRECT option as been activated and the IRREVERS has been invoked on the SATOPTS keyword, then the same set are keywords as for the DIRECT and IRREVERS option are used to assign the drainage relative permeability curves, that is: KRNUMX, KRNUMX-, etc., plus the IMBNUMX, IMBNUMY, MBNUMZ, IMBNUMX-, IMBNUMY-, IMBNUMZ- keywords for the imbibition curves. See Table 5.40 for the various relative permeability table allocation keywords for the various combination of DIRECT, IRREVERS and HYSTER command options.

The keyword should be followed by one or more of the following keyword options.

[103] *Carlson, F. M. (1981) SPE 10157, presented at the 56th Annual SPE Fall Meeting, San Antonio, 1981*

[104] *Killough, J. E. "Reservoir Simulation with History-dependent Saturation Functions," paper SPE 5106, Society of Petroleum Engineers Journal (1976) 16, No. 1, 37-48.*

No.	Name	Description	Default
1	DIRECT	A character string that activates the directional relative permeability assignment option. If the DIRECT command is stated then directional relative permeability assignment is activated and different relative permeability function are assigned to the x, y and z directions. In this case the KRNUMX, KRNUMY and KRNUMZ keywords are used for Cartesian grids to allocate the relative permeability tables. For Radial grids the KRNUMR, KRNUMT and KRNUMZ keywords should be used.	None
2	IRREVERS	A character string the activates reversible directional relative permeability assignment option. If IRREVERS is stated then the relative permeability assignment is set to non-reversible and results in different sets of relative permeability tables being applied for flow from the xI to xI + 1 direction and the xI to the xI – 1 direction, for all directions (x, y, z). in this case the KRNUMX, KRNUMY and KRNUMZ keywords are used for Cartesian grids to allocate the relative permeability tables in the xi to xi + 1 flow directions etc. For Radial grids the KRNUMR, KRNUMT and KRNUMZ keywords should be used. For flow in the xi to xi – 1 flow directions, etc., the KRNUMX-, KRNUMY- and KRNUMZ- keywords are used for Cartesian grids and the KRNUMR-, KRNUMT- and KRNUMZ- are used for radial grids. Only the default option is supported by OPM Flow, that is non-reversible permeability tables are not supported.	None
3	HYSTER	A character string that activates the hysteresis option. If the HYSTER and DIRECT options have activated and the IRREVERS has not been invoked on the SATOPTS keyword, then different relative permeability functions are used for each x, y, and z directions and for the drainage and imbibition processes. Here the drainage relative permeability curves are allocated via the KRNUMX, KRNUMX and KRNUMZ keywords for Cartesian grids and the KRNUMR, KRNUMT and KRNUMZ keywords for radial grids. The imbibition relative permeability curves are allocated via the IMBNUMX, IMBNUMY and IMBNUMZ keywords for Cartesian grids and the IMBNUMR, IMBNUMT and IMBNUMZ keywords for radial grids. If the HYSTER, DIRECT and IRREVERS options have activated, then different relative permeability functions are used for each x, y, and z directions, flow direction and for the drainage and imbibition processes. Then in addition to aforementioned relative permeability curves allocation keywords for the xi to xi + 1 flow direction etc., the xi to xi – 1 flow directions keywords, KRNUMX-, KRNUMY- and KRNUMZ- are used for Cartesian grids and the KRNUMR-, KRNUMT- and KRNUMZ- are used for radial grids. The imbibition relative permeability curves are allocated via the IMBNUMX-, IMBNUMY- and IMBNUMZ keywords for Cartesian grids and the IMBNUMR, IMBNUMT and IMBNUMZ keywords for radial grids.	None

No.	Name	Description	Default
4	SURFTENS	A character string that activates the capillary pressure surface tension pressure dependency option. Only the default option is supported by OPM Flow, that is capillary pressure surface tension pressure dependency option is not supported.	None

Notes:

1) Note that the IRREVERS command can only been activated if the DIRECT command is activated at the same time. See Table 5.40 for the various relative permeability table allocation keywords.
2) The keyword is terminated by a "/".

Table 5.39: SATOPTS Keyword Description

For clarity the following table outlines the keywords that should be used in allocating the relative permeability tables for the various SATOPTS options.

Option	Cartesian		Radial	
DIRECT Flow in all directions	KRNUMX KRNUMY KRNUMZ		KRNUMR KRNUMT KRNUMZ	
DIRECT and IRREVERS Flow in the i to i +1 directions.	KRNUMX, KRNUMY KRNUMZ		KRNUMR KRNUMT KRNUMZ	
Flow in the i to i -1 directions.	KRNUMX- KRNUMY- KRNUMZ-		KRNUMR- KRNUMT- KRNUMZ-	
DIRECT and HYSTER Flow in all directions.	**Drainage** KRNUMX KRNUMY KRNUMZ	**Imbibition** IMBNUMX IMBNUMY IMBNUMZ	**Drainage** KRNUMR KRNUMT KRNUMZ	**Imbibition** IMBNUMR IMBNUMT IMBNUMZ
DIRECT, IRREVERS and HYSTER Flow in the i to i +1 directions.	**Drainage** KRNUMX KRNUMY KRNUMZ	**Imbibition** IMBNUMX IMBNUMY IMBNUMZ	**Drainage** KRNUMR KRNUMT KRNUMZ	**Imbibition** IMBNUMR IMBNUMT IMBNUMZ
Flow in the i to i -1 directions.	KRNUMX-, KRNUMY- KRNUMZ-	IMBNUMX- IMBNUMY- IMBNUMZ-	KRNUMR- KRNUMT- KRNUMZ-	IMBNUMR- IMBNUMT- IMBNUMZ-

Notes:

1) Note the drainage and imbibition classification is related to the wetting phase, that may be either oil or water; however, water is normally assumed in most cases but there are exceptions to this, especially for heavy oils.

2) Keywords colored orange indicate keywords that are not currently supported in OPM Flow.

Table 5.40: SATOPTS Relative Permeability Function Allocation Keywords

Examples

The first example actives the directional relative permeability assignment option only and hence the following keywords are used to allocate the relative permeability arrays for Cartesian grids: KRNUMX, KRNUMY, and KRNUMZ.

```
--
--       ACTIVATE RELATIVE PERMEABILITY ASSIGNMENT HYSTERESIS OPTIONS
--       DIRECTTIONAL(DIRECT) IRREVERSIBLE(IRREVERS) HYSTERESIS(HYSTER)
SATOPTS
         'DIRECT'                                                        /
```

The next example actives the directional irreversible relative permeability assignment options, and hence the following keywords are used to allocate the relative permeability arrays for Cartesian grids: KRNUMX, KRNUMY, KRNUMZ, KRNUMX-, KRNUMY- and KRNUMZ-.

```
--
--       ACTIVATE RELATIVE PERMEABILITY ASSIGNMENT HYSTERESIS OPTIONS
--       DIRECTTIONAL(DIRECT) IRREVERSIBLE(IRREVERS) HYSTERESIS(HYSTER)
SATOPTS
         'DIRECT'   'IRREVERS'                                           /
```

Finally, the last option invokes all three assignment options.

```
--
--       ACTIVATE RELATIVE PERMEABILITY ASSIGNMENT HYSTERESIS OPTIONS
--       DIRECTTIONAL(DIRECT) IRREVERSIBLE(IRREVERS) HYSTERESIS(HYSTER)
SATOPTS
         'DIRECT'   'IRREVERS'   'HYSTER'                                /
```

In this case the drainage relative permeability curves are allocated by the KRNUMX, KRNUMY, KRNUMZ, KRNUMX-, KRNUMY-, KRNUMZ- keywords, and the imbibition relative permeability curves are allocated by the IMBNUMX, IMBNUMY, IMBNUMZ, IMBNUMX-, IMBNUMY-, IMBNUMZ- keywords.

Note

This keyword activates how relative permeability curves are assigned in the model. The ENDSCALE keyword allows the end-point scaling also to vary with direction, flow direction and hysteresis process, resulting in a great deal of flexibility.

Whether or not all these features should be used though is another question.

5.3.127 SAVE – ACTIVATE OUTPUT OF A SAVE FILE FOR FAST RESTARTS

Description

This keyword activates output of a SAVE file for fast restarts which are read in by the LOAD keyword in the RUNSPEC section. No data is required for this keyword.

OPM Flow does not support the SAVE file format for fast restarts like the commercial simulator, but instead writes a standard RESTART file at the requested time step in the SCHEDULE section, which can then be used to restart the simulation at a given point in time via the RESTART keyword in the RUNSPEC section.

Example

The first example requests that a SAVE file be written out in the RUNSPEC section; however, OPM Flow will not write a RESTART record if the SAVE keyword is encountered in the RUNSPEC section.

```
-- ==================================================================================
--
-- RUNSPEC SECTION
--
-- ==================================================================================
RUNSPEC
--
--       WRITE OUT SAVE FILE FOR FAST RESTARTS
--
SAVE
```

The second example shows how the keyword is used in the SCHEDULE section.

```
-- ==================================================================================
--
-- SCHEDULE SECTION
--
-- ==================================================================================
SCHEDULE

-- ----------------------------------------------------------------------------------
-- SCHEDULE SECTION - 2020-01-01
-- ----------------------------------------------------------------------------------
RPTSCHED
          'WELLS=2'    'WELSPECS'    'CPU=2'     FIP=2'                          /

DATES
          1  APR   2021  /
/
--
--       WRITE OUT SAVE FILE FOR FAST RESTARTS
--
SAVE


DATES
          1  JLY   2021  /
          1  OCT   2021  /
/
```

Here OPM Flow will write out a RESTART file instead of a SAVE file at April 1st, 2021.

5.3.128 SCDPDIMS – DEFINE SCALE DEPOSITION AND DAMAGE TABLE DIMENSIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The SCDPDIMS keyword defines the number of tables used in the Scale Deposition option and the maximum number of entries for the various tables.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description	Default
1	NTSCDP	NTSCDP is a positive integer that defines the number of SCDPTAB scale deposition tables used in the Scale Deposition option.	0
2	NPSCDP	NPSCDP is a positive integer that defines the maximum number of entries (or rows) in any one SCDPTAB scale deposition table defined in the input deck.	0
3	NTSCDA	NTSCDA is a positive integer that defines the number of SCDATAB scale damage tables used in the Scale Deposition option.	0
4	NPSCDA	NPSCDA is a positive integer that defines the maximum number of entries (or rows) in any one SCDATAB scale damage table defined in the input deck.	0
5	Not Used		1*
6	Not Used		1*
7	NTSCDE	NTSCDE is a positive integer that defines the number of SCDETAB karst aquifer dissolution tables used in the Scale Deposition option.	0
Notes: 1) The keyword is terminated by a "/".			

Table 5.41: SCDPDIMS Keyword Description

Example

```
--
--        NO.     MAX     NO.     MAX     NOT   NOT   NO.
--        NTSCDP  NPSCDP  NTSCDA  NPSCDA  USED  USED  NTSCDE
SCDPDIMS
          5       10      4       10      1*    1*    3                    /
```

The above example defines the number of SCDPTAB scale deposition tables to be five with a maximum number of rows for each table set to 10, the maximum number of SCDATAB scale damage tables to be four with a maximum number of 10 rows per table, and the maximum number of SCDETAB karst aquifer dissolution tables to be three.

5.3.129 SKIP – ACTIVATE SKIPPING OF ALL KEYWORDS AND INPUT DATA

The SKIP keyword turns on skipping of keywords until the ENDSKIP activation keyword is encountered. All keywords and between the SKIP and ENDSKIP keywords are ignored by OPM Flow.

See *SKIP – Activate Skipping of All Keywords and Input Data* in the GLOBAL section for a full description.

5.3.130 SKIP100 – ACTIVATE SKIPPING OF BLACK-OIL KEYWORDS AND INPUT DATA

This keyword turns on skipping of black-oil keywords until the ENDSKIP activation keyword is encountered. All black-oil keywords between the SKIP100 and ENDSKIP keywords are ignored by OPM Flow.

See *SKIP100 – Activate Skipping of Black-Oil Keywords and Input Data* in the GLOBAL section for a full description.

5.3.131 SKIP300 – ACTIVATE SKIPPING OF "COMPOSITIONAL" KEYWORDS AND INPUT DATA

Turns on skipping of "compositional" keywords until the ENDSKIP activation keyword is encountered. All "compositional" keywords between the SKIP300 and ENDSKIP keywords are ignored by OPM Flow.

See *Error: Reference source not found* in the GLOBAL section for a full description.

5.3.132 SMRYDIMS – DEFINE MAXIMUM NUMBER OF SUMMARY VECTORS TO BE WRITTEN

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The SMRYDIMS keyword defines the maximum number of summary vectors to be written out to the SUMMARY file (*.SUMMARY).

OPM Flow uses dynamic memory allocation and therefore the keyword has no effect and is ignored by the simulator, but is documented here for completeness.

No.	Name	Description	Default
1	NSUMMX	A positive integer that defines the maximum number of summary vectors to be written out to the SUMMARY file (*.SUMMARY).	10000
Notes: 1) The keyword is terminated by a "/".			

Table 5.42: SMRYDIMS Keyword Description

Example

```
--
--       SET THE MAXIMUM NUMBER OF SUMMARY VECTORS THAT CAN BE WRITTEN OUT
--
SMRYDIMS
         10000                                                              /
```

The above example sets maximum number of summary vectors that can be written out to the SUMMARY file to the default value of 10,000; however, this has no effect in OPM Flow input decks.

5.3.133 SOLVDIMS – DEFINE PEBI GRID NESTED FACTORIZATION SOLVER DIMENSIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The SOLVDIMS defines the unstructured Perpendicular Bisector ("PEBI")[105] and [106] grid nested factorization solver dimensions. This keyword is generated by an external pre-processing program for generating simulation grids.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

[105] *Heinemann, Z.E. and Brand, C.W. 1988. Gridding Techniques in Reservoir Simulation. Proc., First Intl. Forum on Reservoir Simulation, Alpbach, Austria, 339.*

[106] *Heinemann, Z.E., Brand, C.W., Munka, M. et al. 1991. Modeling Reservoir Geometry With Irregular Grids. SPE Res Eng 6 (2): 225–232. SPE-18412-PA. http://dx.doi.org/10.2118/18412-PA*

5.3.134 SOLVENT – ACTIVATE THE SOLVENT PHASE IN THE MODEL

Description

This keyword indicates that the solvent phase is present in the model and to activate the four component solvent model for this run. In addition to this keyword, the oil, water and gases phases should also be declared for the run using the OIL, WATER and GAS keywords. The keyword will also invoke data input file checking to ensure that all the required Solvent phase input parameters are defined in the input deck.

There is no data required for this keyword and there is no terminating "/" for this keyword.

Example

```
--
--       SOLVENT PHASE IS PRESENT IN THE RUN
--
SOLVENT
```

The above example declares that the solvent phase is active in the model.

5.3.135 SPIDER – SPIDER GRID ACTIVATION OPTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

SPIDER activates the OPM Flow spider grid geometry option for the model, if this keyword and the RADIAL keyword are omitted then Cartesian geometry is assumed by OPM Flow. Note that is an OPM Flow specific keyword and option.

Spider Grids basically take the standard radial geometry keywords in the grid section and transform the grid specification to an Irregular Corner-Point Grid which can then be viewed in OPM ResInsight in a more intuitive form. The only difference is that in the RUNSPEC section the SPIDER keyword is used instead of the RADIAL keyword. One drawback of the approach that only "cake-slides" can be used for Spider Grids; that is theta must be less than 360 degrees and the dimension must be one, not the full radial geometry.

See also the RADIAL keyword in the RUNSPEC section (*RADIAL – Radial Grid Activation Option*). Both Spider and Radial grids will be displayed the same in OPM ResInsight, as they both use Irregular Corner-Point Grid geometry. The difference is that the radial model's pore volumes are adjusted to match the radial geometry pore volumes, whereas, the Spider grid volumes are not adjusted. See the examples in the section on Radial Grids (*Radial Grids*) and Spider Grids (*Spider Grids*) for further information

Example

```
--
--       DEFINE SPIDER GRID GEOMETRY (OPM FLOW RADIAL GRID KEYWORD)
--
SPIDER
```

The above example actives OPM Flow's spider grid geometry option.

5.3.136 START – SIMULATION START DATE

Description

This keyword sets the start date for the simulation switches. If the DATES keyword is to be used during the simulation, then a start date should be entered.

No.	Name	Description	Default
1	DAY	A positive integer that defines the day of the month, the value should be greater than or equal to one and less than or equal to 31.	None
2	MONTH	Character string for the month and should be one of the following 'JAN', 'FEB', 'MAR', 'APR', 'MAY', 'JUN', 'JUL' (or 'JLY'), 'AUG', 'SEP', 'OCT', 'NOV', or 'DEC'	None
3	YEAR	A positive four digit integer value of the start year, which must be specified fully by four digits, that is 1986.	None
Notes: 1) The keyword is terminated by a "/".			

Table 5.43: START Keyword Description

Example

```
--
--        DEFINE THE START DATE FOR THE RUN
--
START
          01 'JAN' 2014                                                  /
```

The above example sets the start date for the run to be January 1, 2014.

Note

Whenever possible it is a good idea to always set the start date to be at the beginning of the year as per the example. As like most simulators, OPM Flow reports are always stated at the number of days from the start date (and sometimes at a given date). If the start date is at the beginning of the year, then calculating the actual date is relatively straight forward and simple.

5.3.137 SURFACT – ACTIVATE THE SURFACTANT PHASE IN THE MODEL

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword indicates that the surfactant phase is present in the model and to activate the surfactant flooding model. The keyword will also invoke data input file checking to ensure that all the required surfactant phase input parameters are defined in the input deck. See also the SURFACTW keyword in the RUNSPEC section that actives the surfactant phase, but with the changes to the wettability option activated as well.

There is no data required for this keyword and there is no terminating "/" for this keyword.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

Example

```
--
--       ACTIVATE THE SURFACTANT PHASE IN THE MODEL
--
SURFACT
```

The above example declares that the surfactant phase is active in the model.

5.3.138 SURFACTW – ACTIVATE THE SURFACTANT PHASE WITH WETTABILITY CHANGES IN THE MODEL

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword indicates that the surfactant phase is present in the model and to activate the surfactant flooding mode with Changes to Wettability option activated as well. The keyword will also invoke data input file checking to ensure that all the required surfactant phase input parameters are defined in the input deck. See also the SURFACT keyword in the RUNSPEC section that actives the surfactant phase only, that is without the Changes to the Wettability option.

There is no data required for this keyword and there is no terminating "/" for this keyword.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

Example

```
--
--       ACTIVATE THE SURFACTANT PHASE WITH WETTABILITY CHANGES IN THE MODEL
--
SURFACTW
```

The above example declares that the surfactant phase is active in the model together with the wettability changes.

5.3.139 TABDIMS – DEFINE THE NUMBER OF TABLES AND THE TABLE DIMENSIONS

Description

The TABDIMS keyword defines the maximum number of tables for a given table type dataset and the maximum number of entries for the various tables. The commercial simulator combines both the black-oil and compositional simulator variables on this keyword; however, although all the parameters are explained below only the black-oil parameters are used by OPM Flow.

No.	Name	Description	Default
1	NTSFUN	A positive integer that defines the number of relative permeability table sets defined in the input deck. The tables are allocated to different parts of the grid by the SATNUM keyword.	1
2	NTPVT	A positive integer that defines the number of fluid property table sets defined in the input deck. The tables are allocated to different parts of the grid by the PVTNUM keyword.	1
3	NSSFUN	A positive integer that defines the maximum number of saturation entries in the relative permeability tables defined in the input deck.	20
4	NPPVT	A positive integer that defines the maximum number of pressure entries in the PVT tables.	20
5	NTFIP	A positive integer defining the maximum number of regions in the FIPNUM region array. Note that this parameter may also be set on the REGDIMS keyword as well. If NTFIP is set in both places then the maximum value is used.	1
6	NRPVT	A positive integer that defines the maximum number of Rs and Rv entries in the PVT tables. If the DISGAS and VAPOIL options have not been activated then this parameter is ignored.	20
7	NRVPVT	A positive integer that defines the maximum number of Rv entries in the PVT tables for the commercial compositional simulator.	20
8	NTENDP	A positive integer that defines the maximum number of saturation end-point depth tables. The end-point depth tables are used to re-scale the saturation tables as a function of depth as oppose to being a grid block property. NTENDP may also be specified on the ENDSCALE keyword, and if specified on both here and on the ENDSCALE keyword the maximum value of the two is used.	1
9	NMEOSR	A positive integer that defines the maximum number of reservoir equations of states for the commercial compositional simulator.	1
10	NMEOSS	A positive integer that defines the maximum number of separator or surface equations of states for the commercial compositional simulator.	1
11	MXNFLN	A positive integer defining the maximum number flux regions in the FLUXNUM region array. MXNFLN can also be defined on the REGDIMS keywords as well. If MXNFLX is defined both here and on the REGDIMS keyword then the maximum value of the two is used.	10

No.	Name	Description	Default
12	MXNTHR	A positive integer that defines the maximum number of thermal regions for the commercial compositional simulator.	1
13	NTROCC	A positive integer that defines the number of rock compressibility entries enter by the ROCK keyword defined in the input deck. The ROCK data is allocated to different parts of the grid by the either the PVTNUM or ROCKNUM keywords in the REGIONS section. If NTROCC is defaulted then the PVTNUM array will be used to allocate the ROCK keyword data to the grid blocks; whereas, if a value is entered then the ROCKNUM array will be used instead. See also the ROCKOPTS keyword in the PROPS section that can be used to redefine if the PVTNUM, ROCKNUM or SATNUM arrays should be employed to allocate the ROCK keyword data.	1*
14	MXNPMR	A positive integer that defines the maximum number of pressure maintenance regions for the commercial compositional simulator.	0
15	NTABKT	A positive integer that defines the maximum number of temperature dependent K-value tables for the when the thermal option is activated in the commercial compositional simulator.	0
16	NTALPHA	A positive integer that defines the maximum number of transport coefficient tables for the commercial compositional simulator.	0
17	NASPKA	A positive integer that defines the maximum number of maximum number of entries in the ASPKDAM keyword tables for the commercial compositional simulator.	10
18	MXRAWG	A positive integer that defines the maximum number of maximum number of entries in the ASPREWG keyword tables for the commercial compositional simulator.	10
19	MXRASO	A positive integer that defines the maximum number of pressure maintenance regions for the commercial compositional simulator.	10
20		Not Used	1*
21	MCASPP	A positive integer that defines the maximum number of column entries in the ASPPW2D keyword tables for the commercial compositional simulator.	5
22	MRASPP	A positive integer that defines the maximum number of row entries in the ASPPW2D keyword tables for the commercial compositional simulator.	5
23	MXRATF	A positive integer that defines the maximum number of entries in the ASPWETF table for the commercial compositional simulator.	5
24	MXNKVT	A positive integer that defines the maximum number of composition dependent K-value tables for the commercial compositional simulator.	0
25	RESVED	Not Used	1*

No.	Name	Description	Default
Notes: 1) Table sets are groups of keywords that need to be defined to complete a table set. For example if NTPVT is set to three, then there must be three PVTO tables and three PVTG tables to complete the three PVT data set. 2) The keyword is terminated by a "/".			

Table 5.44: TABDIMS Keyword Description

Example

```
--
--        NO.     NO.     MAX     MAX     MAX     MAX     E300
--        NTSFUN  NTPVT   NSSFUN  NPPVT   NTFIP   NRPVT   BLANK   NTEND
TABDIMS
          15      9       40      30      1*      1*      1*      1               /
```

The above example defines number of relative permeability tables to be 15 with a maximum number of rows for each table set to 40, and the number of PVT tables to be nine with a maximum number of 30 rows per table.

5.3.140 TEMP – ACTIVATE THE TEMPERATURE MODELING OPTION

Description

This keyword activates the temperature modeling option in the commercial simulator. There is no data required for this keyword.

The temperature option (TEMP keyword) and the thermal option (THERMAL keyword) are two separate modeling facilities in the commercial simulator, although some keywords can be used by both options, for example the RTEMP keyword. OPM Flow's thermal implementation is based on solving the energy equation fully coupled with the black-oil equations so the results are not directly equivalent to the commercial simulator's black-oil TEMP or compositional THERMAL formulations. To activate OPM Flow's thermal implementation use the THERMAL keyword in the RUNSPEC section.

OPM Flow treats the TEMP keyword as an alias for the THERMAL keyword, although it is recommended that the THERMAL keyword is used instead.

Example

```
--
--       ACTIVATE THE TEMPERATURE MODELING OPTION (NOT SUPPORTED BY OPM FLOW)
--
TEMP
```

The above example activates the temperature modeling option in the commercial simulator, however the keyword is treated as an alias for the THERMAL keyword by OPM Flow and activates OPM Flow's thermal option.

5.3.141 THERMAL– ACTIVATE THE THERMAL MODELING OPTION

Description

This keyword activates the thermal modeling option. There is no data required for this keyword. The energy black-oil implementation in OPM Flow is a mixture of the commercial simulators black-oil and the commercial simulators "compositional thermal" keywords, as well as some OPM Flow specific keywords.

The temperature option (TEMP keyword) and the thermal option (THERMAL keyword) are two separate modeling facilities in the commercial simulator, although some keywords can be used by both options, for example the RTEMP keyword. OPM Flow's thermal implementation is based on solving the energy equation fully coupled with the black-oil equations so the results are not directly equivalent to the commercial simulator's black-oil TEMP or compositional THERMAL formulations.

Keywords specifically associated with both OPM Flow's THERMAL and the commercial simulators TEMP and THERMAL options are listed in Table 5.45 for ease of reference.

Section	Keyword	Function	OPM Flow	Commercial	
			THERMAL MODEL	TEMP MODEL	THERMAL MODEL
RUNSPEC	ROCKDIMS	Thermal Rock Dimensions of Over and Underburden Rock Types			
	TEMP	Activate the Temperature Modeling Option	Alias for THERMAL		Black-Oil
	THERMAL	Activate the Thermal Modeling Option			
GRID	ROCKCON	Thermal Rock Over and Underburden Connection Data			
	ROCKPROP	Thermal Rock Over and Underburden Property Data			
	HEATCR	Rock Heat Capacity.			
	HEATCRT	Rock Heat Capacity Temperature.			
	THCGAS	Gas Phase Thermal Conductivity.			
	THCOIL	Oil Phase Thermal Conductivity.			
	THCONR	Thermal Conductivity of liquids and reservoir rock.			
	THCONSF	Thermal Conductivity of liquids and reservoir rock scaling factor applied to THCONR to account for gas saturation.			
	THCROCK	Rock Thermal Conductivity.			
	THCSOLID	Solid Phase Thermal Conductivity.			
	THCWATER	Water Thermal Conductivity.			

Section	Keyword	Function	OPM Flow	Commercial	
			THERMAL MODEL	TEMP MODEL	THERMAL MODEL
PROPS	HEATVAP	Thermal Oil Component Heat of Vaporization			
	GASDENT	Gas Density Temperature Coefficients (OPM Flow keyword).			
	GASJT	Gas Joule-Thomson Coefficient (OPM Flow keyword).			
	GASVISCT	Gas Viscosity versus Temperature Functions (OPM Flow black-oil keyword).			
	OILDENT	Oil Density Temperature Coefficients (OPM Flow keyword).			
	OILJT	Oil Joule-Thomson Coefficient (OPM Flow keyword).			
	OILVISCT	Oil Viscosity versus Temperature Functions (OPM Flow black-oil keyword).			
	RTEMP	Constant Initial Reservoir Temperature.			
	RTEMPA	Constant Initial Reservoir Temperature.			
	RTEMPVD	Initial Reservoir Temperature versus Depth.			
	SPECHA	Thermal Specific Heat of Oil Component			
	SPECHEAT	Specific Heat of Oil, Water and Gas			
	SPECROCK	Specific Heat of the Reservoir Rock			
	TEMPVD	Initial Reservoir Temperature versus Depth.			
	THANALB	Activate Thermal Analytic Water Density Option			
	THERMEX1	Liquid Components Thermal Expansion Coefficient			
	WATDENT	Oil Density Temperature Coefficients.			
	WATJT	Water Joule-Thomson Coefficient (OPM Flow keyword).			
	WATVISCT	Oil Viscosity versus Temperature Function.			
REGION	THERMNUM	Thermal Region Numbers.			
SOLUTION	RTEMP	Constant Initial Reservoir Temperature.			

Section	Keyword	Function	OPM Flow	Commercial	
			THERMAL MODEL	TEMP MODEL	THERMAL MODEL
	RTEMPA	Constant Initial Reservoir Temperature.			
	RTEMPVD	Initial Reservoir Temperature versus Depth.			
	TEMPI	Initial Reservoir Temperature for All Cells.			
	TEMPVD	Initial Reservoir Temperature versus Depth.			
SCHEDULE	WTEMP	Set An Injection Well's Fluid Temperature			
	WINJTEMP	Define Injection Fluid Thermal Properties			

Notes:

1) Cells colored green implies the keyword can be used with this model formulation.
2) Cells colored orange means the keyword is recognized by OPM Flow's parser but is ignored and not used by OPM Flow. The TEMP keyword is treated as an alias for the THERMAL keyword by OPM Flow, although it is recommended that the THERMAL keyword is used instead.
3) Cells colored red should not be used with this model formulation.
4) The list is focused on the OPM Flow implementation of the energy and black-oil formulation and therefore does not necessary include all the commercial simulator's compositional keywords.

Table 5.45: OPM Flow's THERMAL Option Associated Keywords

In thermal runs a producing well's bottom-hole temperature is calculated based on a weighted average of the temperature in the grid cell connections open to flow in the producing well, that is the reported bottom-hole temperature, TBHT, is calculated as:

$$T_{BHT} = \frac{\sum_{i=1}^{M} W_i T_i}{\sum_{i=1}^{M} W_i} \tag{5.1}$$

with

$$W_i = \sum_{p}^{N} \left(\rho^r_{pi}\right) \left(q^r_{pi}\right) C_p \tag{5.2}$$

The term $\left(\rho^r_{pi}\right) \left(q^r_{pi}\right) C_p$ is the energy rate density (J/(K s)) of phase p,

where:

N = number of phases,
M = number of open connections in the well,
i = the open connection index,
p = the phase index, oil, water and gas.
r = indicating that the parameter is evaluated at reservoir conditions,

T = temperature (K),
q = connection flow rate (m^3/s),
ρ = fluid density (kg/m^3), and
C = specific heat capacity ((J(K kg))
(see the SPECHEAT keyword in the PROPS section).

The current implementation makes use of the internal energy:

$$E_{pi} = C_p \cdot T_i \tag{5.3}$$

derived from the enthalpy, Hpi

$$E_{pi} = H_{pi} + \frac{P_{pi}}{\rho^r{}_{pi}} \tag{5.4}$$

where P_{pi} is connection grid block pressure of phase p.

The phase rates at surface conditions $\left(q^s{}_{p1}\right)$ are converted to reservoir in situ rates $\left(q^r{}_{p1}\right)$ using the phase formation volume factor, Bpi, via:

$$q^r{}_{pi} = \frac{q^s{}_{pi}}{B^{-1}{}_{pi}} \tag{5.5}$$

And thus equation (5.2) can be simplified to:

$$W_i = \sum_p^N \left(\rho^r{}_{pi}\right) \left(\frac{q^s{}_{pi}\, E_{pi}}{B^{-1}{}_{pi} \cdot T_i}\right) \tag{5.6}$$

Example

```
--
--       ACTIVATE THE THERMAL MODELING OPTION (OPM FLOW THERMAL OPTION ONLY)
--
THERMAL
```

The above example activates the thermal modeling option.

5.3.142 TITLE – DEFINE THE TITLE FOR THE INPUT DECK

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The TITLE keyword defines the title for the input deck. The title text will be printed on all reports so as to act as a reference for the run.

No.	Name	Description	Default
1	TITLE	A character string that defines the TITLE for the input deck	None

Notes:

1) All the characters on the line are processed as a string and therefore there is no need to enclose the TITLE in quotes.
2) There is no terminator "/" for the keyword.

Table 5.46: TITLE Keyword Description

Note

It is good practice to include the name of the input file in the tittle (without the extension) for when cross checking results from multiple cases.

Example

```
--
--       DEFINE THE TITLE FOR THE RUN
TITLE
SPE01-THEM01-OPM1810-R01 - OPM THERMAL OPTION RUN
```

The above example defines the title for the run to be "SPE01-THEM01-OPM1810-R01 - OPM THERMAL OPTION RUN".

5.3.143 TRACERS – ACTIVATE TRACER OPTIONS AND SET TRACER ARRAY DIMENSIONS

Description

The TRACERS keyword defines the number of tracers in the model and the various passive tracer tracking options.

No.	Name	Description	Default
1	MXOILTR	A positive integer defining the maximum number of passive oil tracers defined using the TRACER keyword.	0
2	MXWATTR	A positive integer defining the maximum number of passive water tracers defined using the TRACER keyword.	0
3	MXGASTR	A positive integer defining the maximum number of passive gas tracers defined using the TRACER keyword.	0
4	MXENVTR	A positive integer defining the maximum number of passive environmental tracers defined using the TRACER keyword. Environmental tracers are used with the Environment Tracer model that takes into account tracer adsorption and decay. Only the default value of zero is currently supported.	0
5	DIFFOPT	A character string defining the numerical diffusion option for tracer tracking runs that should be set to: 1) DIFF activates the numerical diffusion control options. 2) NODIFF deactivates the numerical diffusion control options. Only the default value of NODIFF is supported	NODIFF
6	MXITRTR	A positive integer defining the maximum number of non-linear iterations to be used when the tracer option is activated.	12
7	MNITRTR	A positive integer defining the minimum number of non-linear iterations to be used when the tracer option is activated.	1
8	NONLIN	A character string stating if passive tracers as should be linear (NO) or non-linear (YES). Only the default value of NO is supported.	No
9	LNCONFAC	A real value defining the initial linear convergence factor. The default value of 1* means the parameter will not be utilized.	1*
10	NLCONFAC	A real value defining the initial non-linear convergence factor. The default value of 1* means the parameter will not be utilized.	1*
11	CONFAC	A real value defining the LNCONFAC and NLCONFAC convergence factors to be used after the initial convergence factor has been applied.	1.0
12	NUMCONF	A positive integer defining the maximum number of times CONFAC can be used.	0

No.	Name	Description	Default
Notes: 1) The keyword is terminated by a "/".			

Table 5.47: TRACERS Keyword Description

See also the TRACER keyword in the PROPS section that defines the individual tracers.

Example

```
--
--       NO OIL  NO WAT  NO GAS  NO ENV  DIFF    MAX    MIN    TRACER
--       TRACERS TRACERS TRACERS TRACERS CONTL   NONLIN NONLIN NONLIN
TRACERS
         0       7       1       0      'NODIFF' 1*     1*     1*             /
```

The above example defines seven tracers in the water phase and one tracer in the gas phase.

5.3.144 TRPLPORO – ACTIVATE THE TRIPLE POROSITY MODEL OPTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The TRPLPORO keyword activates the Triple Porosity Model option that models matrix, fractures and vuggy porosity for carbonate reservoirs, and specifies the number of matrix porosity systems

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description	Default
1	TRPLPORO	A positive integer value that specifies the number of matrix porosity systems in the model. TRPLPORO should be set to either: 1) TRPLPORO set equal to 2, if the vugs are only connected to the fractures, so that the porosity system is matrix and connected vugs, or, 2) TRPLPORO set equal to 3, if the vugs are connected to the fractures and the matrix, so that the porosity system is matrix, connected vugs, and isolated vugs.	1
Notes: 1) The keyword is terminated by a "/".			

Table 5.48: TRPLPORO Keyword Description

Note the keyword cannot be used in conjunction with the NMATRIX keyword, which is also in the RUNSPEC section.

Example

```
--
--       TRPLPORO
--       OPTION
TRPLPORO
         3                                                    /
```

The above example activates the Triple Porosity Model option and specifies the porosity system is matrix, connected vugs, and isolated vugs.

5.3.145 UDADIMS – DEFINE THE DIMENSIONS OF THE USER DEFINED ARGUMENTS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword defines the dimensions of the User Defined Arguments ("UDA") used by OPM Flow that can be applied to various connection, group, and well keywords in the SCHEDULE section. UDAs are defined by the UDQ keyword that is used to specify values to be constants, SUMMARY variables, as defined in SUMMARY section, or a formula using various mathematical functions together with constants and SUMMARY variables.

Although this keyword is read by OPM Flow and the ACTION and UDQ computational logic and calculations have been implemented, one should used caution using this facility as it may result in OPM Flow aborting.

No.	Name	Description	Default
1	NMUDA	NMUDA is a positive integer that defines the number of arguments in a SCHEDULE section keyword, that are replaced by numeric UDQ values.	0
2	IGNORED	Not used and should be defaulted.	1*
3	MXUDA	MXUDA is a positive integer that defines the maximum number of unique arguments in a keyword that are replaced by numeric UDA values. Note that MXUDA differs from NMUDA, for example: 1) If only the oil rate argument of, say the WCONPROD keyword is specified by a UDA, then both NMUDA and MXUDA equal one. 2) However, if a second WCONPROD uses a different UDA, then NMUDA equals two, but MXUDA would still be one. 3) Finally, if the same two distinct UDAs are used separately in two lines of WCONPROD data, then both NMUDA and MXUDA must be set to two. As MXUDA's default value is 100 then this only needs to be increased where the same UDA is used more than 100 times.	100
Notes: 1) The keyword is terminated by a "/".			

Table 5.49: UDADIMS Keyword Description

Note that OPM Flow has a more restricted UDQ feature set than the commercial simulator, so not all options and functions are currently available.

Example

```
--
--        USER DEFINED ARGUMENT DIMENSIONS
--        NO.      NOT      TOTAL
--        ARGS     USED     UDQ
UDADIMS
          10        1*      10                                          /
```

In the above example both NMUDA and MXUDA are set equal to ten.

5.3.146 UDQPARAM – DEFINE PARAMETERS FOR THE USER DEFINED QUANTITY FEATURE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword defines the dimensions of the User Defined Arguments ("UDA") used by OPM Flow that can be applied to various connection, group, and well keywords in the SCHEDULE section. UDAs are defined by the UDQ keyword that is used to specify values to be constants, SUMMARY variables, as defined in SUMMARY section, or a formula using various mathematical functions together with constants and SUMMARY variables.

Although this keyword is read by OPM Flow and the ACTION and UDQ computational logic and calculations have been implemented, one should used caution using this facility as it may result in OPM Flow aborting.

No.	Name	Description	Default
1	RSEED	RSEED is a positive integer greater than zero that sets a new random number seed for use in the UDQ functions RANDN, RANDU RRNDN and RRNDU. See also the RSEED character variable on the UDQDIMS keyword in the RUNSPEC section to default the random number seed for a restart run. This feature is not supported by OPM Flow.	1
2	RANGE	RANGE is a real positive value greater than or equal to one and less than or equal to 1.0×10^{20}, that sets the absolute range for the user defined quantities. The default value of 1×10^{20} sets the range from -1×10^{20} to $+1 \times 10^{20}$.	1×10^{20}
3	DEFAULT	DEFAULT is real value that is the default numerical value given to undefined UDQ variables and should be in the same range as RANGE.	0.0
4	TOLUDQ	TOLUDQ a real positive number greater than zero and less than one that defines the tolerance used to determine if two real values are equal. Floating-point numbers (as implemented in computers) are never exact, one cannot compare floating point numbers for exact equality. Thus, TOLUDQ defines a tolerance. For example, the default value of 1 x 10-4 means that if the difference between two real values is less than 1 x 10-4 then the values are considered equal.	1 x 10-4
Notes: 1) The keyword is terminated by a "/".			

Table 5.50: UDQPARAM Keyword Description

Note that OPM Flow has a more restricted UDQ feature set than the commercial simulator, so not all options and functions are available.

Example

```
--
--        USER DEFINED DEFAULT VALUES
--        SEED    RANGE    UNDEFINED   COMPARISON
--        INTG    -AND+    VALUE       TOLERANCE
UDQPARAM
          1      1.0E20    0.0         1.0E-4                                  /
```

The example explicitly sets the default values for all four variables on the UDAPARAM keyword, namely the random seed to one, the range to 1 x 1020, the undefined UDQ variables to zero, and the comparison tolerance to 1.0 x 10-4.

5.3.147 UDQDIMS – DEFINE THE DIMENSIONS OF THE USER DEFINED UDQ FEATURE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword defines the dimensions associated with the UDQ keyword used in OPM Flow to calculate various user defined values in the SCHEDULE section. The UDQ keyword defined variables can be constants, SUMMARY variables, as defined in the SUMMARY section, or a formula using various mathematical functions together with constants and SUMMARY variables.

Although this keyword is read by OPM Flow and the ACTION and UDQ computational logic and calculations have been implemented, one should used caution using this facility as it may result in OPM Flow aborting.

No.	Name	Description	Default
1	MXFUNS	A positive integer that defines the maximum number of functions that can be included when defining a UDQ definition. This should also include any brackets that will be used in the UDQ definition.	16
2	MXITEMS	MXITEMS is a positive integer that defines the maximum number of ITEMS allowed in an UDQ definition.	16
3	MXUDC	MXUDC is a positive integer that defines the maximum number of user defined CONNECTION quantities allowed in an UDQ definition.	0
4	MXUDF	MXUDF is a positive integer that defines the maximum number of user defined FIELD quantities allowed in an UDQ definition.	0
5	MXUDG	MXUDG is a positive integer that defines the maximum number of user defined GROUP quantities allowed in an UDQ definition.	0
6	MXUDR	MXUDR is a positive integer that defines the maximum number of user defined REGION quantities allowed in an UDQ definition.	0
7	MXUDS	MXUDS is a positive integer that defines the maximum number of user defined SEGMENT quantities allowed in an UDQ definition.	0
8	MXUDW	MXUDW is a positive integer that defines the maximum number of user defined WELL quantities allowed in an UDQ definition.	0
9	MXUDA	MXUDA is a positive integer that defines the maximum number of user defined AQUIFER quantities allowed in an UDQ definition.	0
10	MXUDB	MXUDB is a positive integer that defines the maximum number of user defined BLOCK quantities allowed in an UDQ definition.	0
11	RSEED	RSEED is a character string that determines if a new random number seed should be generated for restart runs for use in the UDQ functions RANDN, RANDU RRNDN and RRNDU. If RSEED is set to Y than a new seed will be generated and if set to the default value of N or 1* then the same seed of the "base" simulation will be employed. See also the RSEED integer variable on the UDQPARAM keyword in the RUNSPEC section to set the random number seed for the current run. This feature is not supported by OPM Flow.	N

No.	Name	Description	Default
Notes: 1) The keyword is terminated by a "/".			

Table 5.51: UDQDIMS Keyword Description

Note that OPM Flow has a more restricted UDQ feature set than the commercial simulator, so not all options and functions are available.

Example

```
--
--       USER DEFINED ARGUMENT DIMENSIONS FACILITY
--       MAX     MAX     MAX    MAX    MAX    MAX   MAX   MAX   MAX  MAX    RAND
--       FUNCS   ITEMS   CONNS  FIELD  GROUP  REGS  SEGTM WELL  AQUF BLCKS  OPT
UDQDIMS
         50      25      0      50     50     0     0     0     0    0      N  /
```

In this case the maximum number of functions that can be included when defining a UDQ definition is set to 50, maximum number of items allowed in an UDQ definition is 25, the maximum number of user defined field quantities allowed in an UDQ definition is 50, and the maximum number of user defined group quantities allowed in an UDQ definition is also 50. All other parameters are defaulted including the RSEED variable (the same seed of the "base" simulation will be employed).

5.3.148 UDTDIMS – DEFINE THE DIMENSIONS OF THE USER DEFINED TABLES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword defines the dimensions of the User Defined Tables ("UDT") used by OPM Flow that can be applied to various connection, group, and well keywords in the SCHEDULE section. UDAs are defined by the UDQ keyword that is used to specify values to be constants, SUMMARY variables, as defined in SUMMARY section, or a formula using various mathematical functions together with constants and SUMMARY variables.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

No.	Name	Description	Default
1	MXUDT	MXUDA is a positive integer that defines the maximum number of User Defined Tables	0
2	NUDT	NUDT is a positive integer that defines the maximum number of rows in any given User Defined Table.	0
3	MXINTP	MXINTP is a positive integer that defines the maximum number of interpolation points allowed in any given dimension.	0
4	MXDIMS	MXDIMS is a positive integer that defines the maximum number of dimensions in any given User Defined Table.	0

Notes:

1) The keyword is terminated by a "/".

Table 5.52: UDTDIMS Keyword Description

Example

```
--
--        USER DEFINED TABLE DIMENSIONS
--
--        MAX     MAX   MAX     MAX
--        TABLES  ROWS  INTPOL  DIMS
UDTDIMS
          3       20    3       2                                    /
```

In the above example the maximum number of UDT tables is set to three and the maximum number of rows for each table is 20, the maximum number of interpolation points in any given dimension is set to three and the maximum number of dimensions is defined as two.

5.3.149 UNCODHMD – ACTIVATE HISTORY MATCH GRADIENT UNENCODED OUTPUT

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

UNCODHMD activates the history match gradient unencoded output for the history match gradient output file. Unencoded files allows external programs to read this file type.

There is no data required for this keyword and there is no terminating "/" for this keyword.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

Example

```
--
--       ACTIVATE HISTORY MATCH GRADIENT UNENCODED OUTPUT
--
UNCODHMD
```

The above example switches on the unified output file option.

5.3.150 UNIFIN – ACTIVATE THE UNIFIED INPUT FILE OPTION

Description

This keyword switches on the Unified Input Files option for all input files. Similar to the commercial simulator, OPM Flow has various options for reading various input files and writing the resulting OPM Flow output files as described in Table 5.53.

Process	Keyword	Description	Files
Input	FMTIN	A character string that defines the input files to be formatted as ASCII i.e. text files, as oppose to binary files. The input deck file is always of this type. The option relates to the OPM Flow derived files that used as input, for for example when restarting from another case. If the keyword is omitted then the default is for binary file input.	*.FEGRID *.FINSPEC *.FINIT *.FRSSPEC *.FUNRST *.FSMSPEC *.FUNSMRY
	MULTIN	A character string that defines the input files to be non-unified multiple files, as opposed to unified files. In this case, one file is read in per reporting time step, as opposed to all time steps reports being read from one file. If the keyword is omitted then the default is for one file per reporting time step.	*.RSSPEC *.X0001 *.SMSPEC *.S0001
	UNIFIN	A character string that defines the input files to be unified files, as opposed to non-unified multiple files. A unified file is a single file containing output for each reporting time step. For this option a single summary file and a single restart file will be read. If the keyword is omitted then the default is for one file per reporting time step.	*.RSSPEC *.UNRST *.SMSPEC *.UNSMRY
Output	FMTOUT	A character string that sets all output files to be formatted as ASCII i.e. text files, as opposed to binary files. The *.PRT, *.LOG and *.DBG files are always of this type. The option relates to the OPM Flow output files only. In this case the files will be portable across operating systems, but will also be very large in terms of hard disk space. For this reason it is recommend that the default option is used so that binary files are outputted. If the keyword is omitted then the default is for binary file input.	*.FEGRID *.FINSPEC *.FINIT *.FRSSPEC *.FUNRST *.FSMSPEC *.FUNSMRY
	MULTOUT	A character string that defines the output files to be non-unified multiple files, as opposed to unified files. In this case, one file is written for each reporting time step, as opposed to all time steps reports being written in one file. If the keyword is omitted then the default is for one file per reporting time step.	*.RSSPEC *.X0001 *.SMSPEC *.S0001

Process	Keyword	Description	Files
	UNIFOUT	A character string that defines the output files to be unified files, as opposed to non-unified multiple files. A unified file is a single file containing output for each reporting time step. Here a single summary file and a single restart file will be generated, as opposed to one file per report time step. If the keyword is omitted then the default is for one file per reporting time step.	*.RSSPEC *.UNRST *.SMSPEC *.UNSMRY

Notes:

1) A binary file is computer-readable but not human-readable. All executable programs are stored in binary files, as are most numeric data files. In contrast, text files are stored in a form (usually ASCII) that is human-readable.
2) For unified files if the run terminates unexpectedly, or there is sufficient disk space, then the last report output is not stored. Their main advantage is that if a number of simulations reside in one directory, their output is organized. There is no limit on the number of reporting steps that a unified file can store.

Table 5.53: UNIFIN Keyword Description

There is no data required for this keyword and there is no terminating "/" for this keyword.

See also *OPM FLOW OUTPUT FILE FORMATS* for a more detailed description of the various file types (ASCII or binary) and file structure formats (unified or non-unified formats).

Example

```
--
--       SWITCH ON THE UNIFIED INPUT FILES OPTION
--
UNIFIN
```

The above example switches on the unified input file option.

5.3.151 UNIFOUT – ACTIVATE THE UNIFIED OUTPUT FILE OPTION

Description

This keyword switches on the Unified Output Files option for all output files. Similar to the commercial simulator, OPM Flow has various options for reading various input files and writing the resulting OPM Flow output files as described in Table 5.54.

Process	Keyword	Description	Files
Input	FMTIN	A character string that defines the input files to be formatted as ASCII i.e. text files, as oppose to binary files. The input deck file is always of this type. The option relates to the OPM Flow derived files that used as input, for for example when restarting from another case. If the keyword is omitted then the default is for binary file input.	*.FEGRID *.FINSPEC *.FINIT *.FRSSPEC *.FUNRST *.FSMSPEC *.FUNSMRY
	MULTIN	A character string that defines the input files to be non-unified multiple files, as opposed to unified files. In this case, one file is read in per reporting time step, as opposed to all time steps reports being read from one file. If the keyword is omitted then the default is for one file per reporting time step.	*.RSSPEC *.X0001 *.SMSPEC *.S0001
	UNIFIN	A character string that defines the input files to be unified files, as opposed to non-unified multiple files. A unified file is a single file containing output for each reporting time step. For this option a single summary file and a single restart file will be read. If the keyword is omitted then the default is for one file per reporting time step.	*.RSSPEC *.UNRST *.SMSPEC *.UNSMRY
Output	FMTOUT	A character string that sets all output files to be formatted as ASCII i.e. text files, as opposed to binary files. The *.PRT, *.LOG and *.DBG files are always of this type. The option relates to the OPM Flow output files only. In this case the files will be portable across operating systems, but will also be very large in terms of hard disk space. For this reason it is recommend that the default option is used so that binary files are outputted. If the keyword is omitted then the default is for binary file input.	*.FEGRID *.FINSPEC *.FINIT *.FRSSPEC *.FUNRST *.FSMSPEC *.FUNSMRY
	MULTOUT	A character string that defines the output files to be non-unified multiple files, as opposed to unified files. In this case, one file is written for each reporting time step, as opposed to all time steps reports being written in one file. If the keyword is omitted then the default is for one file per reporting time step.	*.RSSPEC *.X0001 *.SMSPEC *.S0001

Process	Keyword	Description	Files
	UNIFOUT	A character string that defines the output files to be unified files, as opposed to non-unified multiple files. A unified file is a single file containing output for each reporting time step. Here a single summary file and a single restart file will be generated, as opposed to one file per report time step. If the keyword is omitted then the default is for one file per reporting time step.	*.RSSPEC *.UNRST *.SMSPEC *.UNSMRY

Notes:

1) A binary file is computer-readable but not human-readable. All executable programs are stored in binary files, as are most numeric data files. In contrast, text files are stored in a form (usually ASCII) that is human-readable.
2) For unified files if the run terminates unexpectedly, or there is sufficient disk space, then the last report output is not stored. Their main advantage is that if a number of simulations reside in one directory, their output is organized. There is no limit on the number of reporting steps that a unified file can store.

Table 5.54: UNIFOUT Keyword Description

There is no data required for this keyword and there is no terminating "/" for this keyword.

See also *OPM FLOW OUTPUT FILE FORMATS* for a more detailed description of the various file types (ASCII or binary) and file structure formats (unified or non-unified formats).

Example

```
--
--       SWITCH ON THE UNIFIED OUTPUT FILES OPTION
--
UNIFOUT
```

The above example switches on the unified output file option.

5.3.152 UNIFOUTS – ACTIVATE THE UNIFIED OUTPUT SUMMARY FILE OPTION

Description

The UNIFOUTS keyword causes the SUMMARY file output files to be a unified file, as opposed to non-unified multiple files. A unified file is a single file containing output for each reporting time step. Here a single SUMMARY file will be generated, as opposed to one file per report time step. See also the MULTOUT keyword in the RUNSPEC section that sets both the SUMMARY and RESTART files to be non-unified multiple files, as opposed to unified files. Note also that UNIFOUTS keyword has precedence over the MULTOUT keyword for SUMMARY files.

There is no data required for this keyword and there is no terminating "/" for this keyword.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

Example

```
--
--       ACTIVATE THE UNIFIED OUTPUT SUMMARY FILE OPTION
--
UNIFOUTS
```

The above example switches on writing of unified SUMMARY output files.

5.3.153 UNIFSAVE – ACTIVATE THE UNIFIED OUTPUT SAVE FILE OPTION

Description

The UNIFSAVE keyword causes the SAVE file output file to be a unified file, as opposed to non-unified multiple files. A unified file is a single file containing output for each reporting time step. Here a single SAVE file will be generated, as opposed to one file per report time step. See also the MULTOUT keyword in the RUNSPEC section that sets both the SUMMARY and RESTART files to be non-unified multiple files, as opposed to unified files.

There is no data required for this keyword and there is no terminating "/" for this keyword.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

Example

```
--
--      ACTIVATE THE UNIFIED OUTPUT SAVE FILE OPTION
--
UNIFSAVE
```

The above example switches on writing of unified SAVE output files.

5.3.154 VAPOIL – ACTIVATE THE VAPORIZED OIL IN WET GAS PHASE IN THE MODEL

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword indicates that vaporized oil (more commonly referred to as condensate) is present in wet[107] gas in the model and the keyword should only be used if the there is both oil and gas phases in the model. The keyword may be used for gas-water and oil-water-gas input decks that contain the oil and gas phases. The keyword will also invoke data input file checking to ensure that all the required oil and gas phase input parameters are defined in the input deck.

If the gas has a constant and uniform vaporized oil concentration, Condensate-Gas Ratio ("CGR"), and if the reservoir pressure never drops below the saturation pressure (dew point pressure), then the model can be run more efficiently by omitting the OIL and VAPOIL keywords from the RUNSPEC section, treating the gas as a dry gas[108], and defining a constant Rv (CGR) value with keyword RVCONST or RVCONSTT in the PROPS section. This results in the model being run with as a dry gas problem with no active oil (condensate) phase. However, OPM Flow takes into account the constant Rv in the calculations and reporting.

There is no data required for this keyword and there is no terminating "/" for this keyword.

Example

```
--
--       VAPORIZED OIL IN WET GAS IS PRESENT IN THE RUN
--
VAPOIL
```

The above example declares that the vaporized oil, i.e. condensate, in the gas phase is active in the model.

[107] *Natural gas that contains significant heavy hydrocarbons such as propane, butane and other liquid hydrocarbons is known as wet gas or rich gas. The general rule of thumb is if the gas contains less methane (typically less than 85% methane) and more ethane, and other more complex hydrocarbons, it is labeled as wet gas. Wet gas normally has GOR's less than 100,000 scf/stb or 18,000 Sm3/m3, with the condensate having a gravity greater than 50 °API.*

[108] *Natural gas that occurs in the absence of condensate or liquid hydrocarbons, or gas that had condensable hydrocarbons removed, is called dry gas. It is primarily methane with some intermediates. The hydrocarbon mixture is solely gas in the reservoir and there is no liquid (condensate surface liquid) formed either in the reservoir or at surface. The term dry indicates that the gas does not contain heavier hydrocarbons to form liquids at the surface conditions. Dry gas typically has GOR's greater than 100,000 scf/stb or 18,000 Sm3/m3.*

5.3.155 VAPWAT – ACTIVATE VAPORIZED WATER IN THE DRY AND WET GAS PHASES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword indicates that vaporized water is present in the gas phase and the keyword should only be used if both water and gas phases are present in the model. VAPWAT should also be used in conjunction with the PRECSALT keyword in the RUNSPEC section in order to activate OPM Flow's Salt Precipitation model. VAPWAT may be used for gas-water and oil-water-gas input decks that contain the oil, gas and water phases.

Note

This is an OPM Flow specific keyword for the simulator's Water Vaporization Model that is activated by declaring that vaporized water is present in the run.

Note that if the VAPWAT keyword is in the input deck then either the PVTGW or PVTGWO keywords in PROPS section should be used to defined the gas and water PVT properties.

Secondly, if both the VAPWAT keyword and the PRECSALT keyword (used to activate the OPM Flow's Salt Precipitation model) are present in the input deck, then the RWGSALT keyword in the PROPS section also needs to be present.

There is no data required for this keyword and there is no terminating "/" for this keyword.

Example

```
--
--       VAPORIZED WATER IN DRY/WET GAS IS PRESENT IN THE RUN (OPM FLOW KEYWORD)
--
VAPWAT
```

The above example declares that the vaporized water is present in the gas phase and is active in the model.

5.3.156 VE – ACTIVATE VERTICAL EQUILIBRIUM MODEL (GLOBAL)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword actives the Vertical Equilibrium ("VE") model for the global grid and optionally specifies the type of VE model. The VE model type can either be compressed for merging columns of grid blocks into a single grid block, or uncompressed for the standard VE model.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

5.3.157 VFPIDIMS – INJECTION VERTICAL FLOW PERFORMANCE TABLE DIMENSIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

VFPIDIMS keyword defines the maximum dimensions of the injection well Vertical Lift Performance ("VFP") tables defined by VFPINJ keyword. The VFP tables for the producing wells are defined by the VFPPDIMS keyword.

No.	Name	Description	Default
1	MXMFLO	A positive integer that defines the maximum number of injection rate entries for the VFPINJ keyword.	0
2	MXMTHP	A positive integer that defines the maximum number of THP entries for the VFPINJ keyword.	0
3	MXVFPTAB	A positive integer that defines the maximum number of VFPINJ tables entered through the VFPINJ keyword.	0
Notes: 1) The keyword is terminated by a "/".			

Table 5.55:VFPIDIMS Keyword Description

Example

```
--       INJECTING VFP TABLES
--       VFP      VFP      VFP
--       MXMFLO   MXMTHP   NMMVFT
VFPIDIMS
         10       10       12                                        /
```

The above example defines that the maximum number of injection rates and THP entries on the VFPINJ keyword is 10, and the number of VFPINJ tables is 12.

5.3.158 VFPPDIMS – PRODUCTION VERTICAL FLOW PERFORMANCE TABLE DIMENSIONS

Description

VFPPDIMS keyword defines the maximum dimensions of the production well Vertical Lift Performance ("VFP") tables defined by VFPPROD keyword. The VFP tables for the injection wells are defined by the VFPIDIMS keyword.

No.	Name	Description	Default
1	MXMFLO	A positive integer that defines the maximum number of production flow rate entries for the VFPPROD keyword.	0
2	MXMTHP	A positive integer that defines the maximum number of THP entries for the VFPPROD keyword.	0
3	MXMWFR	A positive integer that defines the maximum number of water fraction entries (WOR, WCUT, GWR etc.) for the VFPPROD keyword.	0
4	MXMGFR	A positive integer that defines the maximum number of gas fraction entries (GOR, GLR, OGR etc.) for the VFPPROD keyword.	0
5	MXMALQ	A positive integer that defines the maximum number of artificial lift quantity entries for the VFPPROD keyword.	0
6	MXVFPTAB	A positive integer that defines the maximum number of VFPPROD tables entered through the VFPPROD keyword.	0

Notes:

1) The keyword is terminated by a "/".

Table 5.56: VFPPDIMS Keyword Description

Example

```
--        PRODUCING VFP TABLES
--        VFP      VFP      VFP      VFP      VFP      VFP
--        MXMFLO   MXMTHP   MXMWFR   MXMGFR   MXMALQ   NMMVFT
VFPPDIMS
          20       10       10       10       6        9                    /
```

Here the example shows that there are a maximum of 20 flow rates, 10 THP entries, 10 water and gas fraction entries, and six artificial lift entries for the nine VFPPROD VFP production tables.

5.3.159 VISAGE - ACTIVATE EXTERNAL RESERVOIR GEO-MECHANICS VISAGE OPTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The VISAGE keyword activates the External Reservoir Geo-Mechanics VISAGE option. The keyword should not be used in input decks as the associated data is generated by an external program.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

5.3.160 VISCD – ACTIVATE DUAL POROSITY VISCOUS DISPLACEMENT OPTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The VISCD keyword activates the Dual Porosity Viscous Displacement option for dual porosity and dual permeability models, and therefore requires either the DUALPORO or DUALPERM keyword to be entered in the RUNSPEC section to activate either one of these options. The VISCD option is used to model the viscous displacement of fluids from the matrix by the fracture pressure gradient, for when the fracture system has a more moderate permeability, and flow to and from the matrix caused by the fracture pressure gradient acts as an additional production mechanism[109]. Normally this mechanism is ignored as the pressure gradient in the fracture system is small due to the very high permeability of the fracture system. See the LX, Lyand LZ keywords in the GRID section that define representative matrix grid block sizes.

There is no data required for this keyword and there is no terminating "/" for this keyword.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

Example

```
--
--       ACTIVATE DUAL POROSITY VISCOUS DISPLACEMENT OPTION
--
VISCD
```

The above example activates the dual porosity viscous displacement option.

[109] *Gilman, J. R. and Kazemi, H. "Improved Calculation for Viscous and Gravity Displacement in Matrix Blocks in Dual-Porosity Simulators," paper SPE 16010 (includes a number of associated papers), Journal of Petroleum Technology (1988) 40, No. 1, 60-70.*

5.3.161 WARN – ACTIVATE WARNING MESSAGES

Turns on warning messages to be printed to the print file (*.PRT); note that this keyword is activated by default and can subsequently be switched off by the NOWARN activation keyword. The warning messages may be turned on and off using keywords WARN and NOWARN. OPM Flow always prints error messages.

See *WARN – Activate Warning Messages* in the GLOBAL section for a full description.

5.3.162 WATER – ACTIVATE THE WATER PHASE IN THE MODEL

Description

This keyword indicate that the water phase is present in the model and must be used for gas-water, oil-water, oil-water-gas input decks that contain the water phase. The keyword will also invoke data input file checking to ensure that all the required water phase input parameters are defined in the input deck.

There is no data required for this keyword and there is no terminating "/" for this keyword.

Example

```
--
--       WATER PHASE IS PRESENT IN THE RUN
--
WATER
```

The above example declares that the water phase is active in the model.

5.3.163 WELLDIMS – DEFINE THE WELLS AND GROUP DIMENSIONS

Description

WELLDIMS defines various well and group dimensions for the run. The commercial simulator combines both the black-oil and compositional simulator variables on this keyword; however, although all the parameters are explained below only the black-oil parameters are used by OPM Flow.

No.	Name	Description	Default
1	MXWELS	A positive integer defining the maximum number of wells for this model.	0
2	MXCONS	A positive integer defining the maximum number of grid block connections per well for this model.	0
3	MXGRPS	A positive integer defining the maximum number of groups for this model.	0
4	MXGRPW	A positive integer defining the maximum number of wells that can belong to a group in the model and the maximum number of child groups in a group. Note that MXGRPW sets both the maximum number of wells in a group and the maximum number of child groups in a group. The former applies to groups that contain wells and the latter applies to groups that contain other groups. See also the GRUPTREE keyword in the SCHEDULE section to define group hierarchy.	0
5	MXSTAGE	A positive integer defining the maximum number of stages per separator for this model. This option is ignored by OPM Flow.	5
6	MXSTRMS	A positive integer defining the maximum number of well streams for this model. This option is ignored by OPM Flow.	10
7	MXMIXS	A positive integer defining the maximum number of mixtures for this model. This option is ignored by OPM Flow.	5
8	MXSEPS	A positive integer defining the maximum number of separators for this model. This option is ignored by OPM Flow.	4
9	MXCOMPS	A positive integer defining the maximum number of mixture components in a mixture for the model. This option is ignored by OPM Flow.	3
10	MXDOCOMP	A positive integer defining the maximum number of well completions that can cross a parallel run domain boundary when the PARALLEL option has been activated. This option is ignored by OPM Flow.	0
11	MXWSLIST	A positive integer defining the maximum number of well lists that a well may be concurrent belong to at one time for this model. This option is ignored by OPM Flow.	1

No.	Name	Description	Default
12	MXWLISTS	A positive integer defining the maximum number of dynamic well lists for this model. This option is ignored by OPM Flow.	1
13	MXWSECD	A positive integer defining the maximum number of secondary wells for this model. This option is ignored by OPM Flow.	10
14	MXNGPP	A positive integer defining the maximum number of row entries per completion in the generalized pseudo-pressure tables used for to calculate the blocking factor associated with condensate drop-out in gas condensate reservoirs. If the generalized pseudo-pressure option has not been activated then this is ignored. This option is ignored by OPM Flow.	201

Notes:

1) Only parameters (1) to (4) are used by OPM Flow.
2) The keyword is terminated by a "/".

Table 5.57: WELLDIMS Keyword Description

Example

```
--
--       WELL    WELL    GRUPS   GRUPS
--       MXWELS  MXCONS  MXGRPS  MXGRPW
WELLDIMS
         60      110     18      40                                  /
```

The above example defines the maximum number of wells to be 60 with 110 completions per well, and maximum number of groups to be 18 with maximum number of wells per group of 40. All other parameters are defaulted.

5.3.164 WPOTCALC – WELL POTENTIAL CALCULATION OPTIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

WPOTCALC defines how shut-in and stopped wells should have their well potentials calculated. Well potentials for wells under these conditions need to have their potentials calculated if they are in a Priority Drilling Queue via the WDRILPRI keyword in the SCHEDULE section, or the Prioritization option has been enabled by the PRIORITY keyword in the SCHEDULE section.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

5.3.165 WSEGDIMS – DEFINE MULTI-SEGMENT WELL DIMENSIONS

Description

The WSEGSDIMS keyword defines the multi-segment well dimensions for the multi-segment well model and the keyword is obligatory if multi-segment wells are being employed in the model.

No.	Name	Description	Default
1	MXWELS	A positive integer defining the maximum number of multi-segment wells for this model.	0
2	MXSEGS	A positive integer defining the maximum number of segments per well for this model.	1
3	MXBRAN	A positive integer defining the maximum number of branches per multi-segment well, including the main branch groups for this model.	1
4	MXLINKS	A positive integer defining the maximum number of segment links per multi-segment well. The simulator does not currently support multi-segment chord links, and therefore this parameter is ignored.	0
Notes: 1) The keyword is terminated by a "/".			

Table 5.58: WSEGDIMS Keyword Description

Example

```
--
--       WELL    WELL    BRANCH  SEGMENT
--       MXWELS  MXSEGS  MXBRAN  MXLINKS
WSEGDIMS
         5       100     10      10                                  /
```

The above example defines the maximum number of multi-segment wells to be five with up to 100 segments per multi-segment well, a maximum number of 10 branches per multi-segment well, and up to 10 segment links per multi-segment well.

CHAPTER 6: GRID SECTION

6.1 INTRODUCTION

The GRID section defines the basic grid properties, including structure, faults and various static rock properties (porosity, permeability etc.). The information in this section will be used by the software to calculate the pore volume (PORV) for each cell, the cell mid-point depths, and the regular transmissibilities (TRANX, TRANY and TRANX) between all the cells, as well as across faults. The OPM Flow calculated parameters can then be edited in the EDIT section.

All models can be classified by the number of dimensions as show in Figure 6.1 (after Mattax[110]). The zero and one dimension models are employed in analytical modeling, while the higher dimensions are used in numerical modeling. The term 4D modeling refers to a 3D model with the fourth dimension being the time domain derived from time-lapse seismic, that is the comparison of 3D seismic surveys at two or more points in time.

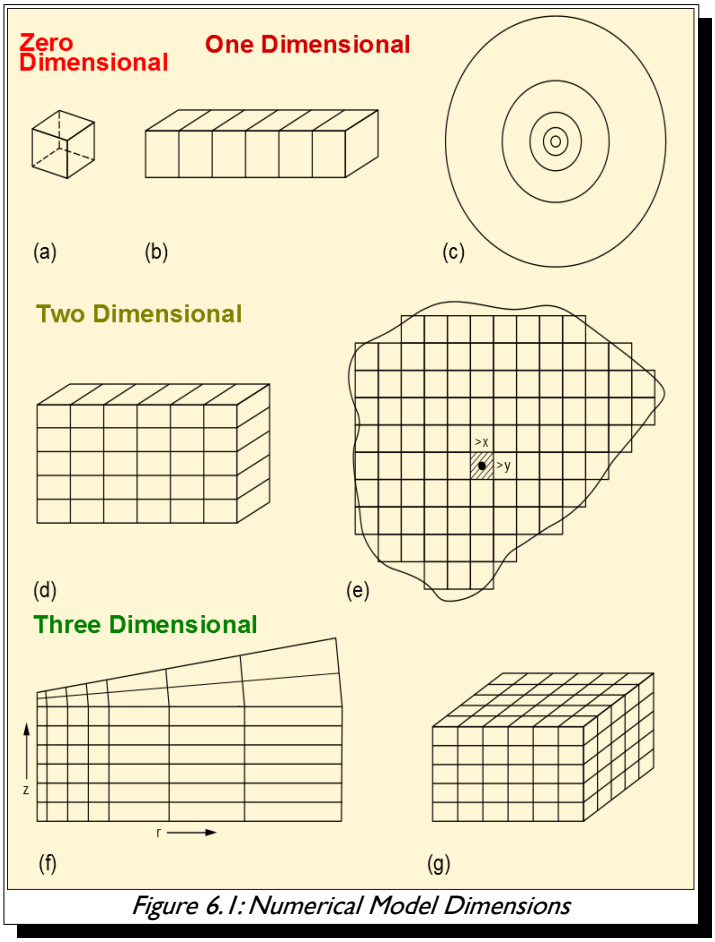


Figure 6.1: Numerical Model Dimensions

OPM Flow enables the user to define 1D, 2D and 3D models using three types of grids: Cartesian Regular Grid, Radial Grid[111], and Irregular Corner-Point Grids. The first two type of grids are rather limited in their ability to describe the structural complexity of oil and gas reservoirs; however, this simplicity allows the engineer to quickly build simple models to investigate reservoir performance. Indeed in the early days of numerical modeling back in the late 1970's two-dimensional cross-section and radial models were the main models used to predict reservoir performance due to limited computer resources at the time. That is not to say that full field models were not developed, but that these full field models were very coarse in comparison to what is designed and built today using static earth modeling software.

A brief introduction to the three types of grids and the data requirements to fully defined the structural element of the grid together with the rock properties necessary to complete the GRID section data requirements is outlined in the following section. This is then followed by the keyword definitions applicable to this section.

[110] *Mattax, C.C. and Dalton R.L. 1990. Reservoir Simulation. Society of Petroleum Engineers, Henry L. Doherty Series, Monograph Vol. 13*

[111] *Radial grids are not currently implemented in this version of OPM Flow, but is expected to be incorporated in a future release.*

6.2 DATA REQUIREMENTS

6.2.1 CARTESIAN REGULAR GRIDS

This type of grid defines a regular orthogonal grid based on defining the x, y and z dimensions of all the cells and is normally employed when a complex structural model is not required. Figure 6.2 shows the SPE Comparative Solution Project Number 1 ("SPE-CSP01") as documented by Odeh[112].

[112] *Odeh, A. "Comparison of Solutions to a Three Dimensional Black-Oil Reservoir Simulation Problem." JPT 33 (1981):13-25.*

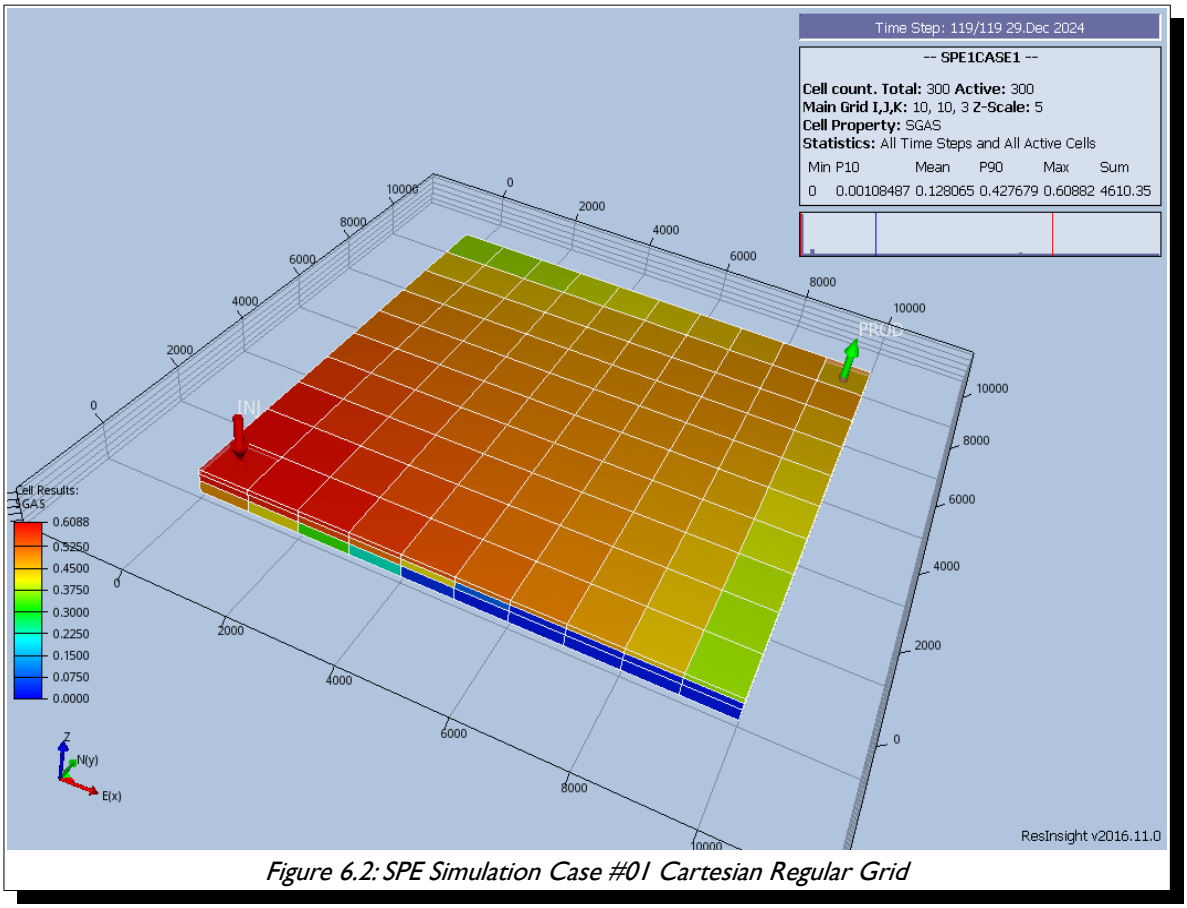


Figure 6.2: SPE Simulation Case #01 Cartesian Regular Grid

The model consists of a simple 10 x 10 x 3 (NX, NY, NZ) grid and is defined using the following GRID section keywords to define the grid geometry:

```
-- ================================================================================
--
-- GRID SECTION
--
-- ================================================================================
GRID
--
--       DEFINE GRID BLOCK SIZES IN THE X DIRECTION BASED ON NX x NY x NZ = 300)
--       (There Are In Total 300 Cells With Length 1000ft In X-Direction)
DX
         300*1000                                                              /
--
--       DEFINE GRID BLOCK SIZES IN THE Y DIRECTION (BASED ON NX x NY x NZ = 300)
--       (There Are In Total 300 Cells With Length 1000ft In Y-Direction)
DY
         300*1000                                                              /
--
--       DEFINE GRID BLOCK SIZES IN THE Z DIRECTION (BASED ON NX x NY x NZ = 300)
--
DZ
         100*20.0   100*30.0   100*50.0                                        /
--
--       DEFINE GRID BLOCK TOPS FOR THE TOP LAYER (BASED ON NX = 100, NY = 100)
--       (Layer 2 and 3 TOPS Calculated by Simulator)
TOPS
         25*3100  25*3105  25*3110                                             /
```

The rock property data required to complete the GRID section is as follows:

```
--
--      DEFINE POROSITY DATA FOR ALL CELLS (BASED ON NX x NY x NZ = 300)
--
PORO
         300*0.300                                                             /
--
--      DEFINE PERMY DATA FOR ALL CELLS (BASED ON NX x NY x NZ = 300)
--
PERMX
         100*500.0   100%50.0   100*200.0                                      /
--
--      DEFINE PERMY DATA FOR ALL CELLS (BASED ON NX x NY x NZ = 300)
--
PERMY
         100*500.0   100%50.0   100*200.0                                      /
--
--      DEFINE PERMZ DATA FOR ALL CELLS (BASED ON NX x NY x NZ = 300)
--      (Not Defined in Original Paper So Assume That PERMX = PERMY = PERMZ)
PERMZ
         100*500.0   100%50.0   100*200.0                                      /
```

The above keywords define all the properties required for the GRID section for this type of grid geometry.

6.2.2 IRREGULAR CORNER-POINT GRIDS

This type of grid is an industry standard grid used to formulate the structure of complex reservoirs [113]. Here static modeling software is used to build the model which is then exported and imported into a numerical model. Figure 6.3 illustrates the skeleton grid for the Norne Field which has dimensions of 46 x 112 x 22 in the x, y and z dimensions respectively. This results in a total number of cells of 113,344 although not all of these cells will be active in the model.

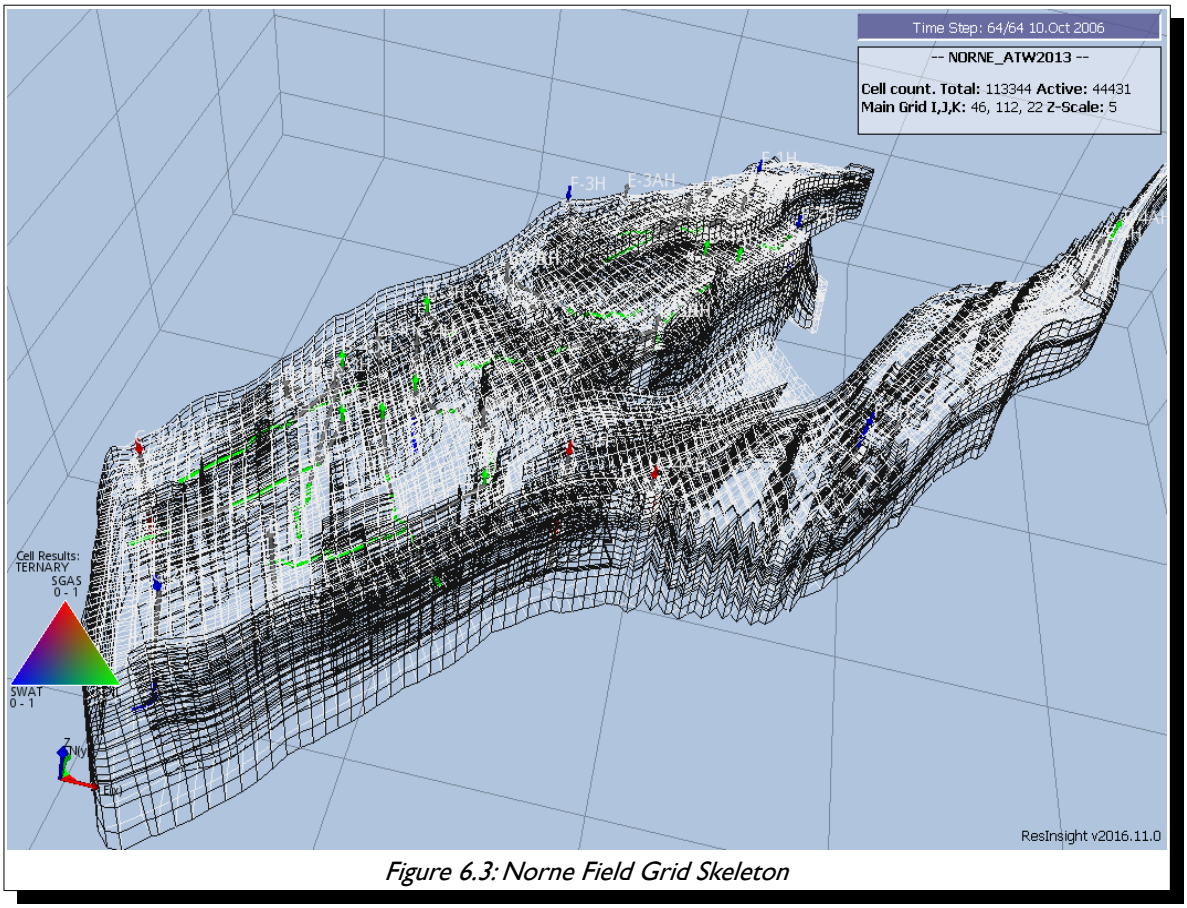


Figure 6.3: Norne Field Grid Skeleton

Similar to Cartesian Regular Grid the grid geometry must be defined for each cell and the properties for each cell defined. The formulation of the grid geometry is based on corner-point geometry, basically coordinate lines or pillars are given, then top and bottom surfaces for the cell are given by specifying the depth (z-coordinates) of the cell's corner points along each of the four adjacent pillars. The cell then forms an irregular hexahedron as depicted in Figure 6.4. Note that the figure shows a corner-point cell which is more or less orthogonal, which is ideally is what we want to minimize grid orientation effects.

The data required to define this type of grid consists of the SPECGRID to define the dimensions of the grid, that is:

```
--        MAX     MAX     MAX     MAX     GRID
--        NDIVIX  NDIVIY  NDIVIZ  NUMRES  TYPE
SPECGRID
          46      112     22      1       F                    /
```

[113] *K. Ponting, D, Corner Point Geometry in Reservoir Simulation, Conference Proceedings, ECMOR I - 1st European Conference on the Mathematics of Oil Recovery, Jul 1989, cp-234-00003 dDOI= "https://doi.org/10.3997/2214-4609.201411305", European Association of Geoscientists &; Engineers,*

A portion of the coordinate line data defined by the COORD keyword from the Norne model is shown on the next page.

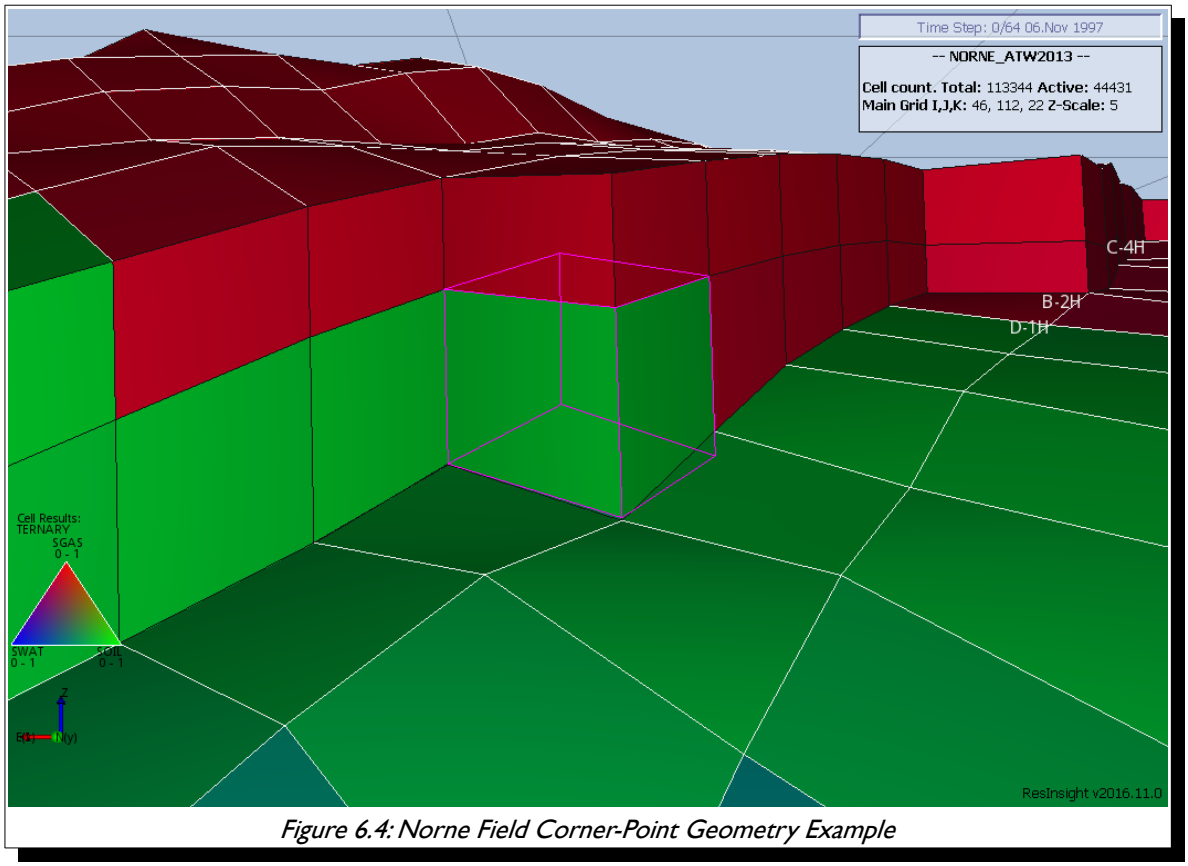


Figure 6.4: Norne Field Corner-Point Geometry Example

```
COORD
-- X1           Y1           Z1          X2           Y2           Z2
-- ---          ---          ----        ---          ---          ----
   453114.000   7319921.000  3037.473    453114.000   7319921.000  3132.831
   453155.031   7319840.000  2983.933    453142.750   7319864.000  3173.572
   453196.094   7319759.000  3005.969    453171.500   7319807.500  3215.836
   453237.156   7319678.000  3000.265    453200.250   7319751.000  3217.250
   453278.188   7319597.000  2989.348    453229.031   7319694.000  3213.951
   453319.250   7319516.500  2995.680    453257.781   7319637.500  3215.323
   453356.250   7319443.500  3000.855    453308.750   7319537.000  3220.549
   453393.250   7319370.500  3005.252    453359.688   7319436.500  3210.393
   453423.969   7319310.000  3030.862    453394.219   7319368.500  3203.438
   453454.688   7319249.500  3036.870    453428.719   7319300.500  3190.770
   453485.406   7319189.000  3038.017    453463.219   7319232.500  3190.660
   453516.125   7319128.000  3045.027    453497.750   7319164.500  3188.813
   453546.844   7319067.500  3055.410    453532.250   7319096.500  3185.966
   453577.562   7319007.000  3066.541    453566.750   7319028.500  3184.325
   453608.281   7318946.500  3076.624    453601.250   7318960.500  3183.584
   453639.000   7318886.000  3086.938    453635.781   7318892.500  3184.057
   453669.719   7318825.500  3096.153    453670.281   7318824.500  3185.988
   453700.438   7318765.000  3104.703    453704.781   7318756.500  3188.598
   453731.156   7318704.500  3097.016    453739.281   7318688.500  3180.484
   453761.875   7318644.000  3088.539    453773.812   7318620.500  3177.091
   453780.000   7318608.000  3098.118    453796.562   7318575.500  3176.401
   453798.125   7318572.500  3096.691    453819.344   7318530.500  3172.299
.....
/
```

The final keyword to define an Irregular Corner-Point geometry grid is the ZCORN keyword that defines the depths of the cell corners. A portion of the ZCORN data from the Norne model is shown below.

```
ZCORN
   3037.473    2983.933    2983.933    3005.969    3005.969    3000.265
   3000.265    2989.348    2989.348    2995.680    2995.680    3000.855
   3000.855    3005.252    3005.252    3030.862    3030.862    3036.870
   3036.870    3038.017    3038.017    3045.027    3045.027    3055.410
   3055.410    3066.541    3066.541    3076.624    3076.624    3086.938
   3086.938    3096.153    3096.153    3104.703    3104.703    3097.016
   3097.016    3088.539    3088.539    3098.118    3098.118    3096.691
   3096.691    3093.886    3093.886    3085.393    3085.393    3081.957
   3081.957    3080.645    3080.645    3115.021    3115.021    3130.474
   3130.474    3204.674    3204.674    3193.187    3193.187    3169.512
   3169.512    3101.928    3101.928    3044.277    3044.277    3023.930
   3023.930    2964.244    2964.244    2900.178    2900.178    2875.715
   2875.715    2864.913    2864.913    2855.256    2855.256    2841.119
   2841.119    2826.261    2826.261    2806.556    2806.556    2781.052
   2781.052    2791.720    2791.720    2817.940    2817.940    2813.308
   2813.308    2788.492
.....
/
```

The rock property data required to complete the GRID section is the same as for a Cartesian Regular grid, as defined in section *6.2.1 Cartesian Regular Grids* and the data is defined using the same keywords. The resulting Norne model showing the ternary solution variable is displayed in Figure 6.5.

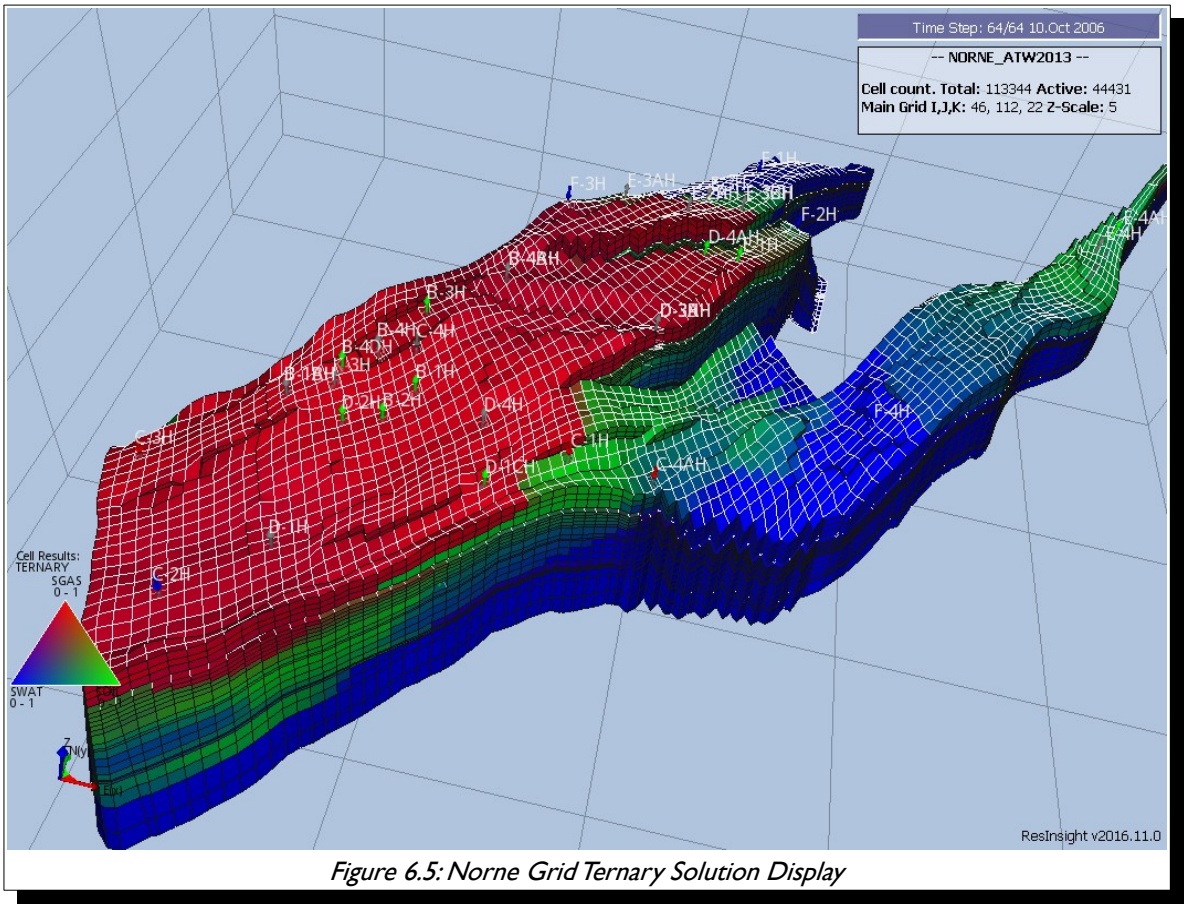


Figure 6.5: Norne Grid Ternary Solution Display

6.2.3 RADIAL GRIDS

Radial flow exists near the wellbore and linear or Cartesian flow occurs within the reservoir away from the wellbore. For field-wide studies the growth or decay of reservoir volumes activated by unsteady-state flow is small and hence linear flow is modeled via a linear grid, using either a Cartesian Regular Grid or an Irregular Corner-Point Grid, in these type of studies. On the other hand, for converging flow near wells the growth or decay of reservoir volumes activated by unsteady-state flow is large and radial flow should modeled. Thus, radial flow is important in only very localized areas around the wells and linear flow is modeled in most reservoir studies, for example in full field or sector models. Consequently for investigating near well bore effects, radial models are used to investigate the behavior around the wellbore, for example in gas cusping and water conning studies, as one can see from Figure 6.6.

In both Cartesian Regular and Irregular Corner-Point grids the dimension nomenclature is (x, y, z) for the x, y, and z planes. The corresponding nomenclature for Radial Grids is (r, theta and z) for r or radius plane, theta for the angular plane measured in degrees, and z for the z plane.

In Figure 6.6 the radial grid has been labeled as a 2-D Cylindrical grid because there is no flow in the theta direction only in the r and z planes, whereas the 3-D Reservoir grid in the figure has flow in both the x and y planes as well as the z plane. Figure 6.7 shows a typical 3-D radial grid with dimensions (10, 10, 3) in the (r, theta, z) dimensions taken from Odeh[114].

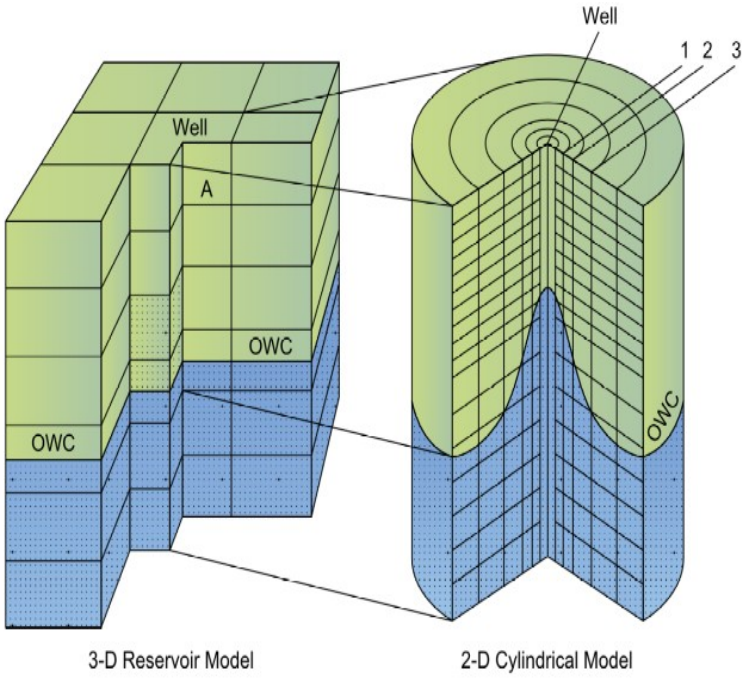


Figure 6.6: Cartesian and Radial Grids Comparison

Radial grids normally have a fine grid near the wellbore which then expands logarithmically away from the wellbore. This is illustrated in Figure 6.7 where one can clearly see the four most outer rings, but not the inner six. This type of grid makes solving the flow equations more challenging than conventional Cartesian and Irregular Corner-Point grids, were the cell pore volume distribution is relatively similar for neighboring cells, compared with Radial grids. For comparison, a typical conventional grid will have grid blocks size of 300, 300, 3 feet (or approximately 100m x 100m x 1m) spacing in the (x, y, z) plane; whereas a radial grid will utilize an inner most radius of between 0.25 to 1.0 foot in the R direction, and then logarithmically expand from the inner radius. The Odeh114 example employs 0.25 ft. for the inner radius, and 1.75, 2.32, 5.01, 10.84, 23.39, 50.55, 109.21, 235.92, 509.68, and 1101.08 ft. for the outer radii.

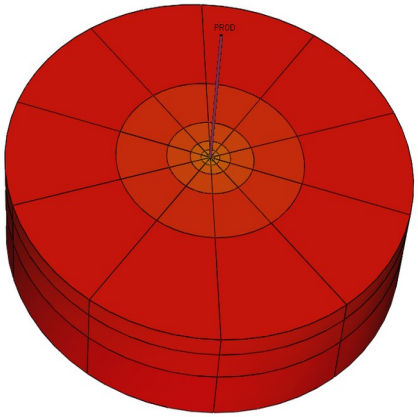


Figure 6.7: 3D Radial Grid (Odeh Example)

[114] *Comparison of Solutions to a Three-Dimensional Black-Oil Reservoir Simulation Problem by Aziz S. Odeh,-- Journal of Petroleum Technology, January 1981.*

Figure 6.7 is based on a 10 x 10 x 3 grid, where NX is the R dimensions, NY the THETA dimensions and NZ the z dimensions on the DIMENS keyword in the RUNSPEC section. Thus, in order to fully define this radial grid the following RUNSPEC and GRID section keywords are required:

```
-- ================================================================================
--
-- RUNSPEC SECTION
--
-- ================================================================================
RUNSPEC
--
--       MAX     MAX     MAX
--       NDIVIX  NDIVIY  NDIVIZ
DIMENS
         10      10      3                                                      /
--
--       DEFINE RADIAL GRID GEOMETRY
--
RADIAL
-- ================================================================================
--
-- GRID SECTION
--
-- ================================================================================
GRID
--
--       INNER RADIUS OF FIRST GRID BLOCK IN THE RADIAL DIRECTION
--
INRAD
         0.25
/
--
--       DEFINE GRID BLOCK SIZES IN THE R DIRECTION
--
DRV
         1.75  2.32  5.01  10.84  23.39  50.55  109.21  235.92  509.68  1101.08 /
--
--       DEFINE GRID BLOCK SIZES IN THE THETA DIRECTION (BASED ON NY = 10)
--
DTHETAV
         10*36                                                                  /
--
--       DEFINE GRID BLOCK Z DIRECTION CELL SIZE (BASED ON NX x NY x NZ = 300)
--
DZ
         100*20.0   100*30.0   100*50.0                                         /
--
--       DEFINE GRID BLOCK TOPS FOR THE TOP LAYER (NX=10,NY=10,and NZ=3)
--
TOPS
         100*8325                                                              /
```

The rock property data required to complete the GRID section is as follows:

```
--
--       DEFINE POROSITY DATA FOR ALL CELLS (BASED ON NX x NY x NZ = 300)
--
PORO
         300*0.300                                                             /
--
--       DEFINE GRID BLOCK PERMR DATA FOR ALL CELLS (BASED ON NR x NY x NZ = 300)
--
PERMR
         100*500.0   100*50.0   100*200.0                                      /
--
```

```
--        DEFINE GRID BLOCK PERMTHT DATA FOR ALL CELLS (NX x NY x NZ = 300)
--
PERMTHT
         100*500.0    100*50.0    100*200.0                                    /
--
--         DEFINE GRID BLOCK Z DIRECTION CELL SIZE (BASED ON NX x NY x NZ = 300)
--
DZ
         100*500.0    100*50.0    100*200.0                                    /
```

The above keywords define all the properties required for the RUNSPEC and GRID sections for this type of grid geometry.

Note

Radial grids are currently not fully implemented in OPM Flow as the completion of the circle in the THETA direction is not supported. For reference only, this feature is activated via the COMPLETE parameter on the COORDSYS keyword in the GRID section.

OPM ResInsight is the 3D visualization software that is used to view the grid block property data and OPM Flow results (grid block pressures, fluid saturations etc.); unfortunately, the software does not render radial grids correctly, and currently there is no plan to implement this feature. Figure 6.8 shows how the Odeh[115] radial model is rendered in OPM ResInsight and this should be compared to Figure 6.7 which displays the correct rendering of the grid.

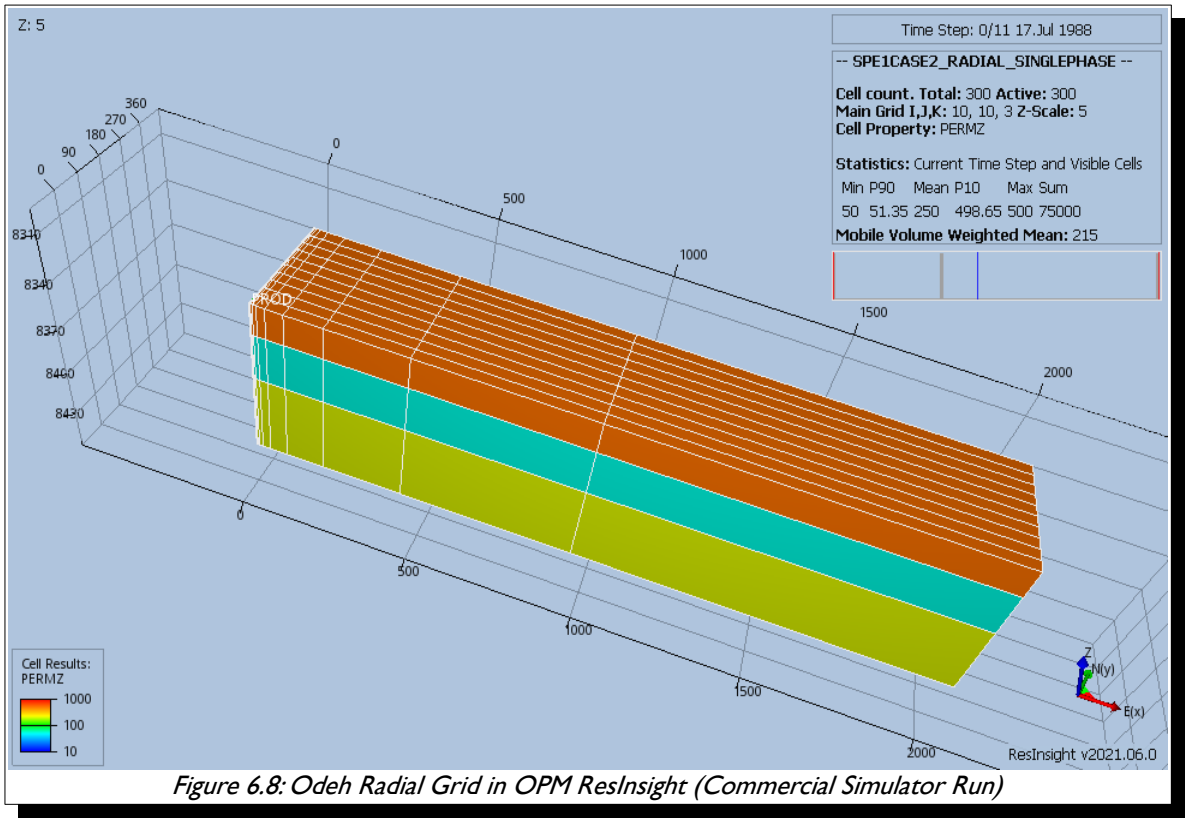


Figure 6.8: Odeh Radial Grid in OPM ResInsight (Commercial Simulator Run)

[115] *Comparison of Solutions to a Three-Dimensional Black-Oil Reservoir Simulation Problem by Aziz S. Odeh,-- Journal of Petroleum Technology, January 1981.*

The previous figure (Figure 6.8) shows how the model would be displayed if the commercial simulator run was loaded into OPM ResInsight. Previously, an OPM Flow run would also be displayed in the same manner.

To overcome this OPM ResInsight display limitation, OPM Flow now converts radial grids to Irregular Corner-Point Grids (Cartesian coordinates) and adjusts the model to reflect radial coordinates Thus, the OPM Flow radial model has the same pore volumes as the commercial simulator radial model.

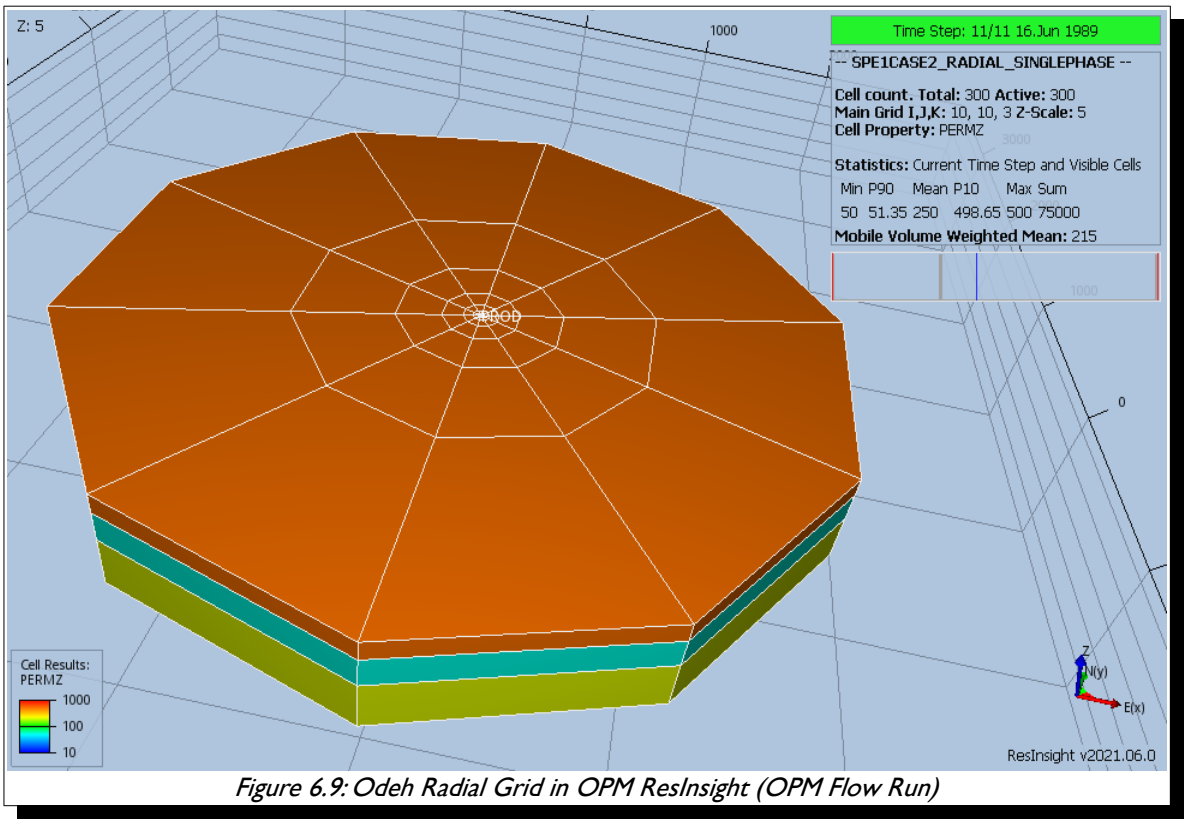


Figure 6.9: Odeh Radial Grid in OPM ResInsight (OPM Flow Run)

Figure 6.9 shows how the OPM Flow run is now displayed in OPM ResInsight. As will be shown in section 6.2.4 Spider Grids, both Spider and Radial grids will be displayed the same, as they both use Irregular Corner-Point Grid geometry. The difference is that the radial model's pore volumes are adjusted to match the radial geometry pore volumes, whereas, the Spider grid volumes are not adjusted.

6.2.4 SPIDER GRIDS

As mentioned in the previous section, OPM ResInsight does not render radial grids correctly and currently there is no plan to implement this feature. In order to overcome this limitation Spider Grids have been implemented that basically take the standard radial geometry keywords in the grid section and transform the grid specification to an Irregular Corner-Point Grid, which can then be viewed in OPM ResInsight in a more intuitive form. The only difference is that in the RUNSPEC section the SPIDER keyword is used instead of the RADIAL keyword.

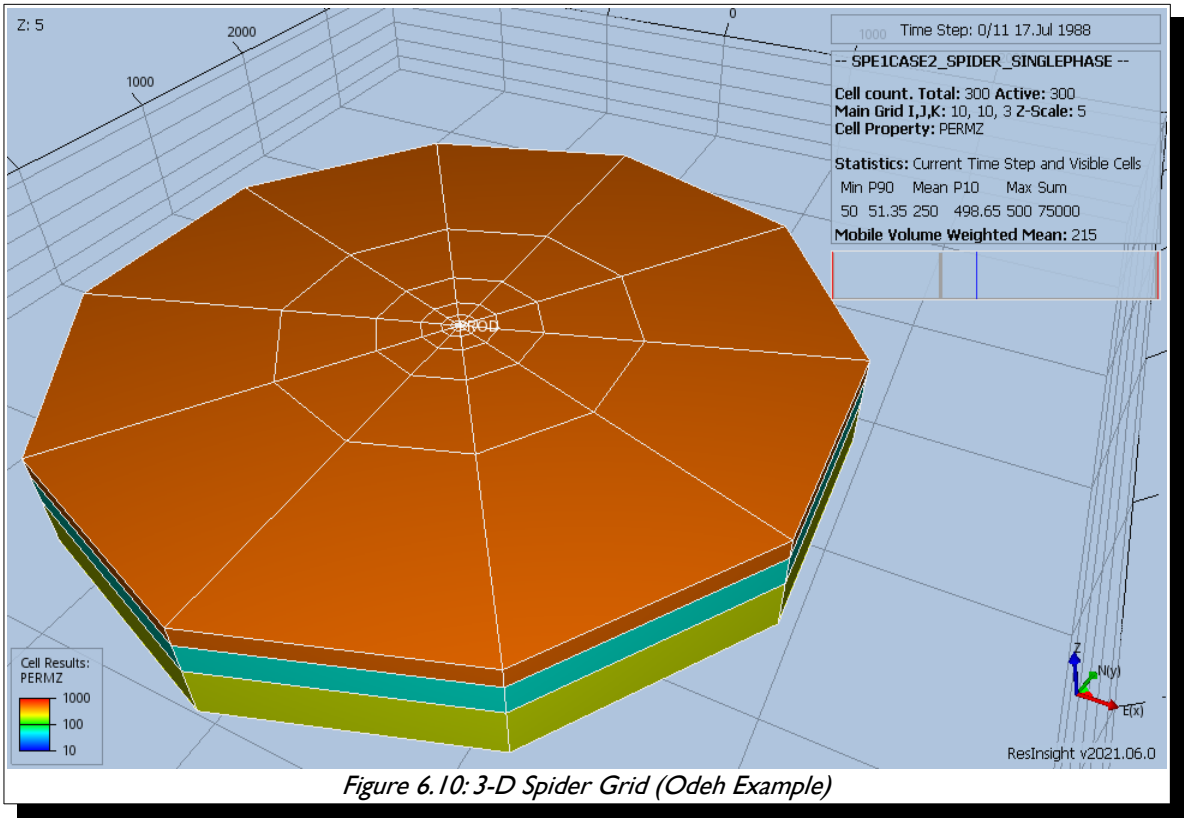


Figure 6.10: 3-D Spider Grid (Odeh Example)

Figure 6.10 illustrates the spider grid for the Odeh[115] example, which is not dissimilar to the conventional radial model shown in Figure 6.7 and is identical to OPM Flow's radial model implementation shown in Figure 6.9. Note that Figure 6.7 was not generated by OPM ResInsight,

Naturally, there will be differences in the results between radial and spider grid formulations, but the user should use their own judgement if the differences are relevant.

Again, Figure 6.10 is based on a 10 x 10 x 3 grid, where NX is the R dimensions, NY the THETA dimensions and NZ the z dimensions on the DIMENS keyword in the RUNSPEC section. The only difference between this example and the previous example for radial grids is that the RADIAL keyword has been replaced by the SPIDER keyword in the RUNSPEC section, all the other keywords are exactly the same.

```
-- ================================================================================
--
-- RUNSPEC SECTION
--
-- ================================================================================
RUNSPEC
--
--       MAX     MAX     MAX
--       NDIVIX  NDIVIY  NDIVIZ
DIMENS
         10      1       3                                                      /
--
--       DEFINE SPIDER GRID GEOMETRY (OPM FLOW RADIAL GRID KEYWORD)
--
SPIDER
-- ================================================================================
--
-- GRID SECTION
--
-- ================================================================================
GRID
--
--       INNER RADIUS OF FIRST GRID BLOCK IN THE RADIAL DIRECTION
--
INRAD
         0.25
/
--
--       DEFINE GRID BLOCK SIZES IN THE R DIRECTION
--
DRV
         1.75  2.32  5.01  10.84  23.39  50.55  109.21  235.92  509.68  1101.08 /
--
--       DEFINE GRID BLOCK SIZES IN THE THETA DIRECTION (BASED ON NY = 10)
--
DTHETAV
         1*45                                                                    /
--
--       DEFINE GRID BLOCK Z DIRECTION CELL SIZE (BASED ON NX x NY x NZ = 30)
--
DZ
         10*20.0   10*30.0   10*50.0                                             /
--
--       DEFINE GRID BLOCK TOPS FOR THE TOP LAYER (NX=10,NY=1,and NZ=3)
--
TOPS
         10*8325                                                                /
```

The rock property data required to complete the GRID section is as follows:

```
--
--       DEFINE POROSITY DATA FOR ALL CELLS (BASED ON NX x NY x NZ = 30)
--
PORO
         30*0.300                                                               /
--
--       DEFINE GRID BLOCK PERMR DATA FOR ALL CELLS (BASED ON NR x NY x NZ = 30)
--
PERMR
         10*500.0   10*50.0   10*200.0                                          /
```

```
--
--       DEFINE GRID BLOCK PERMTHT DATA FOR ALL CELLS (NX x NY x NZ = 30)
--
PERMTHT
         10*500.0   10*50.0   10*200.0                                        /
--
--        DEFINE GRID BLOCK Z DIRECTION CELL SIZE (BASED ON NX x NY x NZ = 300)
--
DZ
         10*500.0   10*50.0   10*200.0                                        /
```

The above keywords define all the properties required for the GRID section for this type of grid geometry.

6.2.5 ROCK PROPERTIES

Irrespective of the grid type used to define the structural component of the model various static properties need to be defined in order for the model to have a complete grid definition, these properties include the identification of active and inactive grid blocks, porosity, permeability, and the reservoir quality via the net-to-gross fraction ("NTG"). These parameters must be set for each cell in the model

Property	Description	Cartesian And Irregular Corner-Point Grids Keywords	Radial Grid Keywords
Active and Inactive cells	Defines if a cell in the model is active by setting the ACTNUM property for a cell to either one or inactive by setting the value to zero. Cells that are inactive in the model are ignored computationally and can act as barriers to flow. Thus, a shale in a conventional reservoir is normally treated as non-reservoir and is made inactive either by setting the ACTNUM, PORO, or NTG to zero for the cells representing the shale.	ACTNUM	
Porosity	Porosity is a measure of the space in a reservoir rock. It is defined as the fraction of the total bulk volume of the rock not occupied by solids, that is it is the fraction of the cell that is porous and contains the reservoir fluids.	PORO	
Reservoir Quality	Reservoir quality of the cell in terms of the gross volume derived from the structural grid and the net volume available for fluid flow in the model expressed as a fraction from zero to one. A zero values means the cell does contribute to flow and therefore is made inactive. A value of one means the gross and net volumes are identical for the cell	NTG	
Permeability	Permeability is a measure of the ease with which a fluid will flow through a porous medium. In numerical models permeability is dependent on the direction of flow, that is x, y and z directions in Cartesian and Irregular Corner-Point Grids, and the radial, theta and z directions in radial grids. There are various formulations for permeability, absolute permeability, effective permeability, gas permeability, liquid permeability etc., and the values are saturation dependent. Thus, values entered should be consistent with the relative permeability entered in the PROPS section. Normally Kair (Sg=1.0) should be entered for the cell permeability and the values may or not be corrected for overburden or humidity drying effects. Correcting for liquid flow and saturation end points etc., is accomplished by the relative permeability curves. For example, if Kair (Sg=1.0) has been entered for the cell permeability when Krg (Sg=1-Swc) should be less than one.	PERMX PERMY PERMZ	PERMR PERMTHT PERMZ

Table 6.1: Key Static Grid Properties

Note

Static grid properties are frequently generated from a static earth model using petrophysical evaluation of the well logs and propagated through the model based on a variety of geostatistical techniques.

Petrophysical evaluations are conducted in either in the "Total" or the "Effective" porosity domain, and is important that all the rock property data is entered into the model is of the same basis. It is not important which porosity domain is used, as long as all the data is in the same domain.

Pore volume and transmissibility are common terms in the reservoir simulation vernacular. Pore volume is self-explanatory, that is, given the grid property data the pore volume for each cell is calculated using:

$$PV = Cell\,Gross\,Volume \times PORO \times NTG \times ACTNUM \tag{6.1}$$

Where

PV	= the pore volume of a cell,
Cell Gross Volume	= the gross volume (or bulk volume) calculated from the structural parameters of the cell,
PORO	= cell porosity,
NTG	= cell net-to-gross ratio, and
ACTNUM	= active and inactive cell indicator.

Any cell with a pore volume equal to zero is made inactive automatically in the model. However, there may be some cells that have small pore volumes than may negatively impact computational performance of the model. If this is the case then the MINPV and MINPORV keywords in the GRID section can be used to make these cells inactive.

There has been a trend in the industry in recent years to not apply petrophysical cut-offs to determine net volumes in static models. This results in large models with numerous cells with very low porosity values (less than 0.01 for example) and corresponding very low permeabilities. The theory behind this approach is that the numerical model will determine the effective (or net) reservoir. However, the approach is questionable as by not applying net pay cut-offs, the in-place and recoverable volumes may not satisfy Reserve reporting requirements and guidelines to various agencies. Secondly, although this may be appropriate in unconventional reservoirs, as all the cells in the model will have similar values of porosity and permeability, but in conventional reservoirs this methodology will lead to severe computational issues when attempting to run the model, due to very tight cells being next to relative high permeability cells. Again, the MINPV and MINPORV keywords can be used to resolve this issue.

Transmissibility on the other hand is more complex as it relates the flow from one cell face to another cell face and is a function of the area open to flow, the direction of flow, the permeability, saturation and viscosity of the phases flowing between the cells. For a single phase flow in a Cartesian grid the x-direction transmissibility is of the form:

$$T_{x_{i+\frac{1}{2},\,j}} = \left[\frac{k_x\, h\,(\Delta y)}{\mu\,(\Delta x)}\right]_{i+\frac{1}{2},\,j} \tag{6.2}$$

As transmissibility is a property of the flow between two cell faces, not a block centered grid cell property like porosity or permeability, then the nomenclature for transmissibility is different. In OPM Flow, the transmissible of cell face $T_x(i, j, k)$ is the transmissibility between cells (i, j, k) and (i+1, j, k). In some simulators it would be between (i, j, k) and (i-1, j, k). This is important to note if manual modifications to cell connections are to be made in the model.

Modifications to basic grid property data, porosity and permeability etc., can only be done in the GRID section, thereafter only the calculated pore volumes and transmissibilities are available for adjustment in the EDIT section. Some engineers use the GRID section for the basic model formulation, and the EDIT section to document the changes to the "base" model using the PORV and TRANX, TRANY, TRANZ, etc. keywords.

6.3 KEYWORD DEFINITIONS

6.3.1 ACTNUM – SET THE STATUS OF A GRID BLOCK TO ACTIVE OR INACTIVE

Description

ACTNUM specifies which grid blocks are active or inactive. A value of 1 indicates the block is active and a value of 0 indicates the block is inactive. A grid block is inactive if its pore volume is less than the value entered using keyword MINPV. This keyword can be used to make blocks with a pore volume greater than MINPV inactive. Note that a value must be entered for each grid block in the model. Grid blocks are ordered with the I index cycling fastest, followed by the J and K indices. As for all array data repeat counts may be used, for example 100*1; however the full array must be specified.

Note that a cells activity can also be set using the EQUALS keyword by selection of only those cells that are required to be made inactive.

No.	Name	Description	Default
1	ACTNUM	An array of integers that define the activity of a cell by setting it to 1 for being active or 0 for inactive, for each grid block in the model.	1*

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX keyword..

2) The keyword is terminated by a "/".

Table 6.2:ACTNUM Keyword Description

Examples

The example below sets several cells to be inactive for a 4 x 5 x 2 model.

```
ACTNUM

0 0 1 1 # layer 1
0 0 1 1
1 1 1 1
1 1 1 1
1 1 1 1

1 1 1 1 # layer 2
1 1 1 1
1 1 1 1
1 1 1 1
0 0 0 0
/
```

Alternatively the EQUALS keyword could be employed to accomplish the same task, that is:

```
--        -- ARRAY    CONSTANT --  ---------- BOX ---------
--                                  I1  I2   J1  J2   K1  K2
EQUALS
          'ACTNUM'    1.0000       1*  1*   1*  1*   1*  1* / SET ACTIVE CELLS
          'ACTNUM'    0.0000       1   2    1   2    1   1  / SET INACTIVE CELLS
          'ACTNUM'    0.0000       1   4    4   4    2   2  / SET INACTIVE CELLS
/
```

6.3.2 ADD – ADD A CONSTANT TO A SPECIFIED ARRAY

Description

The ADD keyword adds a constant to a specified array or part of an array. The constant can be real or integer depending on the array type; however, the arrays that can be operated on is dependent on which section the ADD keyword is being applied.

No.	Name	Description	Default
1	ARRAY	The name of the array to be modified. This is the keyword name identifying the property and is up to eight characters in length and enclosed in quotes.	None
2	CONSTANT	An integer or real value to be added to the ARRAY in the same units as the ARRAY property.	0
3	I1	A positive integer that defines the lower bound of the array in the I-direction to be modified must be greater than or equal 1 and less than or equal to I2 and NX.	1
4	I2	A positive integer that defines the upper bound of the array in the I-direction to be modified must be greater than or equal to I1 and less than or equal to NX	NX
5	J1	A positive integer that defines the lower bound of the array in the J-direction to be modified must be greater than or equal 1 and less than or equal to J2 and NY.	1
6	J2	A positive integer that defines the upper bound of the array in the J-direction to be modified must be greater than or equal to J1 and less than or equal to NY.	NY
7	K1	A positive integer that defines the lower bound of the array in the K-direction to be modified must be greater than or equal to one and less than or equal to K2 and NZ.	1
8	K2	A positive integer that defines the upper bound of the array in the K-direction to be modified must be greater than or equal to K1 and less than or equal to NZ.	NZ

Notes:

1) Where NX, NY and NZ are the dimensions of the model as defined on the DIMENS keyword in the RUNSPEC section.
2) Each record must be terminated by a "/" and the keyword is terminated by a "/".

Table 6.3:ADD Keyword Description

The applicable arrays for each section are defined in Table 6.4 on the following page.

ADD Keyword And Variable Options By Section						
GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
DX	DEPTH	SWL	ENDNUM	PRESSURE		
DY	PORV	SWCR	EQLNUM	SWAT		
DZ	TRANX	SWU	FIPNUM	SGAS		
PERMX	TRANY	SGL	IMBNUM	RV		
PERMY	TRANZ	SGCR	MISCNUM	RS		
PERMZ	DIFFX	SGU	PVTNUM	TBLK		
MULTX	DIFFY	KRW	ROCKNUM	GI		
MULTY	DIFFZ	KRO	SATNUM	OILAPI		
MULTZ	TRANR	KRG	WH2NUM	SALT		
DR	TRANTHT	PCG		GASCONC		
THETA	DIFFR	PCW		SOLVCONC		
PERMR	DIFFTHT			SOLVFRAC		
PERMTHT				SFOAM		
DZNET				SPOLY		
PORO						
NTG						
FLUXNUM						
MULTNUM						
MPANUM						
DIFFX						
DIFFY						
DIFFZ						
DIFFR						
DIFFTHT						

Table 6.4:ADD Keyword Applicable Arrays by Section

Example

```
--
--        ARRAY       CONSTANT      ---------- BOX ---------
--                                   I1  I2   J1  J2   K1  K2
ADD
          PERMX       20.000         1*  1*   1*  1*   1*  1* / ADD 20 mD TO PERMX
/
```

The above example ADDS 20 units to the PERMX array in the GRID section to all grid blocks in the model.

6.3.3 ADDREG – ADD A CONSTANT TO AN ARRAY BASED ON A REGION NUMBER

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The ADDREG keyword adds a constant to a specified array or part of an array based on cells with a specific region number. The region number array can be FLUXNUM, MULTNUM or OPERNUM and these arrays must be defined and be available before the ADDREG keyword is read by the simulator. The constant can be real or integer depending on the property array type; however, the property arrays that can be operated on is dependent on which section the ADDREG keyword is being used.

No.	Name	Description	Default
1	ARRAY	The name of the array to be modified. This is the keyword name identifying the property and is up to eight characters in length and enclosed in quotes.	None
2	CONSTANT	An integer or real value to be added to the ARRAY in the same units as the ARRAY property for a given REGION	0
3	REGION NUMBER	REGION NUMBER is a positive integer representing the region for which the CONSTANT in (2) should be applied	None
4	REGION ARRAY	The REGION ARRAY to use for applying the CONSTANT in (2) based on the REGION NUMBER in (3). REGION ARRAY can have the following values: 1) F for the FLUXNUM array 2) M for the MULTNUM array 3) O for the OPERNUM array	M

Notes:

1) Where the REGION NUMBER should be less than or equal to the maximum number of regions as defined on the REGDIMS keyword for the FIPNUM and OPERNUM arrays or the GRIDOPTS keyword for the MULTNUM array in the RUNSPEC section.
2) Each record must be terminated by a "/" and the keyword is terminated by a "/".

Table 6.5:ADDREG Keyword Description

The applicable arrays for each section are defined in Table 6.6.

ADDREG Keyword And Variable Options By Section						
GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
DX	DEPTH	SWL	ENDNUM	PRESSURE		
DY	PORV	SWCR	EQLNUM	SWAT		
DZ	TRANX	SWU	FIPNUM	SGAS		
PERMX	TRANY	SGL	IMBNUM	RV		
PERMY	TRANZ	SGCR	MISCNUM	RS		
PERMZ	DIFFX	SGU	PVTNUM	TBLK		
MULTX	DIFFY	KRW	ROCKNUM	GI		
MULTY	DIFFZ	KRO	SATNUM	OILAPI		
MULTZ	TRANR	KRG	WH2NUM	SALT		

ADDREG Keyword And Variable Options By Section						
GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
DR	TRANTHT	PCG		GASCONC		
THETA	DIFFR	PCW		SOLVCONC		
PERMR	DIFFTHT			SOLVFRAC		
PERMTHT				SFOAM		
DZNET				SPOLY		
PORO						
NTG						
FLUXNUM						
MULTNUM						
MPANUM						
DIFFX						
DIFFY						
DIFFZ						
DIFFR						
DIFFTHT						

Table 6.6:ADDREG Keyword Applicable Arrays by Section

Example

```
--
-- FIRST DEFINE THE PROPERTY ARRAYS AND MULTNUM ARRAYS FOR 10 X 10 X 20 MODEL
--
--  ARRAY    CONSTANT --  ---------- BOX ---------
--                         I1  I2   J1  J2   K1  K2
EQUALS
    PORO      0.2000       1*  1*   1*  1*   1*  1*  / PORO  TO 0.20 IN MODEL
    PERMX     100.00       1*  1*   1*  1*   1*  1*  / PERMX TO 0.10 IN MODEL
    MULTNUM   1            1*  1*   1*  1*   1*  1*  / MULTNUM IN MODEL
    MULTNUM   2            1*  5    1   5    6   6   / MULTNUM IN MODEL
    MULTNUM   3            1*  1*   1*  1*   10  10  / MULTNUM IN MODEL
/
--
-       NOW RESET PORO AND PERMX BASED ON THE MULTNUM REGION NUMBER
--
--      ADD A CONSTANT TO AN ARRAY BASED ON A REGION NUMBER
--
--      ARRAY     CONSTANT  REGION   REGION ARRAY
--                VALUE     NUMBER    M / F / O
ADDREG
      PORO         0.050     1         M               /
      PORO         0.100     2         M               /
      PORO        -0.050     3         M               /
      PERMX        25.00     1         M               /
      PERMX        100.0     2         M               /
      PERMX       -50.00     3         M               /
/
```

The example first defines the PORO and PERMX property arrays for the model and then sets the MULTNUM array to 1 for all cells in the model, after which selected areas of model are assigned various MULTNUM integer values. The ADDREG can then be invoked to add or subtract constant values from the PORO and PERMX arrays for the various MULTNUM regions.

6.3.4 ADDZCORN – ADD A CONSTANT TO THE ZCORN DEPTH ARRAY

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The ADDZCORN keyword adds a constant to the ZCORN array or part of the array based on cells defined in the specified input box. The constant can be real or integer and can be negative or positive.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

6.3.5 AMALGAM – DEFINE LGR AMALGAMATIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The AMALGAM keyword defines a Cartesian Local Grid Refinements ("LGR") amalgamations, that is merging several LGRs into one amalgamated LGR.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

6.3.6 AQUANCON – DEFINE ANALYTICAL CONNECTIONS TO THE GRID

Description

The AQUANCON keyword defines how analytical aquifers are connected to the simulation grid, this includes the Carter-Tracy, Fetkovich and Constant Flux analytical aquifers, all of which are implemented in OPM Flow. Carter-Tracy analytical aquifers are characterized by the AQUCT keyword in the GRID section and Fetkovich analytical aquifers are defined by the AQUFETP keyword in the SOLUTION section. Finally, the Constant Flux aquifer is defined by the AQUFLUX keyword in SOLUTION section.

Note that numerical aquifers are connected to the grid using the AQUCON keyword in the GRID section and that both aquifer types dimensions are declared by the AQUDIMS keyword in the RUNSPEC section.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	AQUID	AQUID is a positive integer greater than or equal to one and less than the maximum number of analytical aquifers as defined by the NANAQU variable on the AQUDIMS keyword in the RUNSPEC section, that defines the aquifer to be connected to the grid.			None
2	I1	A positive integer that defines the lower bound of the cells in the I-direction to be connected to the aquifer and must be greater than or equal to one and less than or equal to I2 and NX.			1
3	I2	A positive integer that defines the upper bound of the of the cells in the I-direction to be connected to the aquifer and must be greater than or equal to I1 and less than or equal to NX			NX
4	J1	A positive integer that defines the lower bound of the cells in the J-direction to be connected to the aquifer and must be greater than or equal to one and less than or equal to J2 and NY.			1
5	J2	A positive integer that defines the upper bound of the cells in the J-direction to be connected to the aquifer and must be greater than or equal to J1 and less than or equal to NY.			NY
6	K1	A positive integer that defines the lower bound of the cells in the K-direction to be to be connected to the aquifer and must be greater than or equal to one and less than or equal to K2 and NZ.			1
7	K2	A positive integer that defines the upper bound of the cells in the K-direction to be connected to the aquifer and must be greater than or equal to K1 and less than or equal to NZ.			NZ
8	AQUFACE	AQUFACE is a character string that sets the connection "face" of the cells declared by this record and should be set to one of the following: 1) X+, Y+, or Z+ for the positive direction, or X-, Y- or Z- for the negative direction transmissibilities. 2) I+, J+, or K+ for the positive direction, or I-, J- or K- for the negative direction transmissibilities.			None
9	AQUFLUX	AQUFLUX is a positive real value that sets the fraction of the total influx between the aquifer and the defined cells declared on this keyword. If defaulted the cell face area for each cell is applied and if a value is declared then this value is applied to all cells declared by this record.			1*

No.	Name	Description			Default
		Field	Metric	Laboratory	
		ft^2	m^2	cm^2	
10	AQUCOEF	AQUCOEF is a real positive values that scales the calculated connection between the aquifer and the cells declared on this record.			
		dimensionless	dimensionless	dimensionless	1.0
11	AQUOPT	AQUOPT is a character string that sets the cell face connection and should be set to one of the following: 1) YES: Aquifer connections can adjoin to active cells allowing for connections inside the reservoir grid. It is not recommended to use this option without thoroughly checking the connections in the model. 2) NO: Aquifer connections cannot adjoin to active cells preventing connections inside the reservoir grid. This is the recommended and the default value.			NO

Notes:

1) Where NX, NY and NZ are the dimensions of the model as defined on the DIMENS keyword in the RUNSPEC section.
2) Each record must be terminated by a "/" and the keyword is terminated by a "/".

Table 6.7: AQUANCON Keyword Description

Note

If the AQUANCON keyword has been utilized in the run deck then OPM Flow will write the AQUIFERA array to the *.INIT file in order to visualize the aquifer connections in OPM ResInsight.

This is accomplished by setting the AQUIFERA value to 2(AQUID-1) for cells connected to aquifer AQUID. If a cell is connected to multiple analytical aquifers then AQUIFERA is summed for all aquifers connected to a cell. Note that connecting cells to multiple aquifers is best avoided.

Example

The following example defines aquifer number one connected to the I+ face of various cells in the model.

```
--
--                      ANALYTIC AQUIFER CONNECTION
--
--       ID     ---------- BOX ---------    CONNECT  AQF    AQF      ADJOIN
--       NUMBER I1  I2   J1  J2   K1  K2    FACE     INFLX  MULTI    CELLS
AQUANCON
         1     57  57   28  36   46  58     'I+'    1*     1*       'NO'        /
         1    111 111   38  41   22  31     'I+'    1*     1*       'NO'        /
         1     96  96   44  49   22  31     'I+'    1*     1*       'NO'        /
         1     43  43   28  35   54  58     'I+'    1*     1*       'NO'        /
         1     98  98   38  42   32  40     'I+'    1*     1*       'NO'        /
         1     79  79   41  67    5  11     'I+'    1*     1*       'NO'        /
         1     61  61   48  72   12  17     'I+'    1*     1*       'NO'        /
/
```

See the AQUCT keyword in the GRID section for a complete example on defining and connecting a Carter-Tracy aquifer to a simulation grid.

6.3.7 AQUCON – DEFINE NUMERICAL AQUIFER CONNECTIONS TO THE GRID

Description

AQUCON keyword defines how numerical aquifers are connected to the simulation grid and these type of aquifers are characterized by the AQUNUM keyword in the GRID section. Analytical aquifers are connected to the simulation grid by the AQUANCON keyword in the GRID section, this includes the Carter-Tracy and Fetkovich analytical aquifers, both of which are implemented in OPM Flow. Both aquifer types dimensions are declared by the AQUDIMS keyword in the RUNSPEC section.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	AQUID	AQUID is a positive integer greater than or equal to one and less than or equal to the maximum number of numerical aquifers as defined by the MXNAQN variable on the AQUDIMS keyword in the RUNSPEC section, that defines the aquifer to be connected to the grid.			None
2	I1	A positive integer that defines the lower bound of the cells in the I-direction to be connected to the aquifer and must be greater than or equal to one and less than or equal to I2 and NX.			1
3	I2	A positive integer that defines the upper bound of the of the cells in the I-direction to be connected to the aquifer and must be greater than or equal to I1 and less than or equal to NX			NX
4	J1	A positive integer that defines the lower bound of the cells in the J-direction to be connected to the aquifer and must be greater than or equal to one and less than or equal to J2 and NY.			1
5	J2	A positive integer that defines the upper bound of the cells in the J-direction to be connected to the aquifer and must be greater than or equal to J1 and less than or equal to NY.			NY
6	K1	A positive integer that defines the lower bound of the cells in the K-direction to be to be connected to the aquifer and must be greater than or equal to one and less than or equal to K2 and NZ.			1
7	K2	A positive integer that defines the upper bound of the cells in the K-direction to be connected to the aquifer and must be greater than or equal to K1 and less than or equal to NZ.			NZ
8	AQUFACE	AQUFACE is a character string that sets the connection "face" of the cells declared by this record and should be set to one of the following: 1) X+, Y+, or Z+ for the positive direction, or X-, Y- or Z- for the negative direction transmissibilities. 2) I+, J+, or K+ for the positive direction, or I-, J- or K- for the negative direction transmissibilities.			None

No.	Name	Description			Default
		Field	Metric	Laboratory	
9	AQUMULT	AQUMULT is a positive real number greater than or equal to zero that scales the OPM Flow calculated transmissibility between the AQUID aquifer and the grid cell connections defined by this record. The default value of one sets the transmissibility between the aquifer and grid cells to the OPM Flow calculated value.			
		dimensionless	dimensionless	dimensionless	1.0
10	AQUOPT1	AQUOPT1 is a defined integer value set to either zero or one, that defines the area to be used in calculating the connection transmissibility between the aquifer and the grid cells: 1) A value of zero means the cross-sectional defined on the AQUNUM keyword will be used, whereas, 2) A value of one means the cross-sectional defined by the grid cell connections defined by this record will be used.			
		dimensionless	dimensionless	dimensionless	0
11	AQUOPT2	AQUOPT2 is a character string that sets the cell face connection and should be set to one of the following: 1) YES: Aquifer connections can adjoin to active cells allowing for connections inside the reservoir grid. It is not recommended to use this option without thoroughly checking the connections in the model. 2) NO: Aquifer connections cannot adjoin to active cells preventing connections inside the reservoir grid. This is the recommended and the default value.			NO
12	VEOPT1	Vertical Equilibrium Option Number 1– Not Used			1
13	VEOPT2	Vertical Equilibrium Option Number 2– Not Used			1

Notes:

1) Where NX, NY and NZ are the dimensions of the model as defined on the DIMENS keyword in the RUNSPEC section.
2) Each record must be terminated by a "/" and the keyword is terminated by a "/".

Table 6.8: AQUCON Keyword Description

Note

If the AQUCON keyword has been utilized in the run deck then OPM Flow will write the AQUIFERN array to the *.INIT file in order to visualize the aquifer connections in OPM ResInsight.

This is accomplished by setting the AQUIFERN value to 2(AQUID-1) for cells connected to aquifer AQUID. If a cell is connected to multiple numerical aquifers then AQUIFERN is summed for all aquifers connected to a cell. Note that connecting cells to multiple aquifers is best avoided.

Finally, for cells representing the numerical aquifers themselves, AQUIFERN is set to minus AQUID.

Example

The following example defines numerical aquifer number one connected to the I+ face of various cells in the model.

```
--
--        NUMERICAL AQUIFER CONNECTIONS
--
--       ID     ---------- BOX ---------    CONNECT  TRANS  TRANS  ADJOIN
--       NUMBER I1  I2   J1  J2   K1  K2    FACE     MULT   OPTN   CELLS
AQUCON
         1     57  57   28  36   46  58    'I+'     1*     1*     'NO'        /
         1    111 111   38  41   22  31    'I+'     1*     1*     'NO'        /
         1     96  96   44  49   22  31    'I+'     1*     1*     'NO'        /
         1     43  43   28  35   54  58    'I+'     1*     1*     'NO'        /
         1     98  98   38  42   32  40    'I+'     1*     1*     'NO'        /
         1     79  79   41  67    5  11    'I+'     1*     1*     'NO'        /
         1     61  61   48  72   12  17    'I+'     1*     1*     'NO'        /
/
```

See the AQUNUM keyword in the GRID section for a complete example on defining and connecting a numerical aquifer to a simulation grid.

6.3.8 AQUCT – DEFINE CARTER-TRACY ANALYTICAL AQUIFERS

Description

The AQUCT keyword defines Carter-Tracy[116] and [117] analytical aquifers, the properties of the aquifer, including the Carter-Tracy aquifer influence function associated with the aquifer, defined by the AQUTAB keyword in the PROPS section.

Each row entry in the AQUCT keyword defines one Carter-Tracy aquifer.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	AQUID	A positive integer greater than or equal to one and less than or equal to NANAQU on the AQUDIMS keyword in the RUNSPEC section, that defines the Carter-Tracy aquifer number.			1
2	DATUM	DATUM is a single positive value that defines the Carter-Tracy reference datum depth for PRESS.			
		feet	m	cm	None
3	PRESS	PRESS is a single positive value that defines the aquifer pressure at DATUM. If PRESS is defaulted then the simulator will set the aquifer's initial reservoir pressure to be in equilibrium with the cells the aquifer is contacted to.			
		psia	barsa	atma	1*
4	PERM	PERM is a real positive number that assigns the permeability to the aquifer.			
		mD	mD	mD	None
5	PORO	PORO is a real positive number greater than zero and less than or equal to one that assigns the porosity to the aquifer.			
		dimensionless	dimensionless	dimensionless	None
6	RCOMP	RCOMP is a real number defining the total (rock and water) compressibility (Ct) at the DATUM pressure.			
		1/psia	1/barsa	1/atma	None
7	RE	RE is a real positive number that defines the Carter-Tracy aquifer external radius.			
		feet	m	cm	None
8	DZ	DZ is a real positive number that defines the Carter-Tracy aquifer average net thickness.			
		feet	m	cm	None

[116] *Carter, R. D. and Tracy, G. W. "An Improved Method for Calculating Water Influx," Transactions of AIME (1960) 219, 215-417.*

[117] *Van Everdingen, A. & Hurst, W.,. The Application of the Laplace Transformation to Flow Problems in Reservoirs. Petroleum Transactions, AIME (December, 1949).*

No.	Name	Description			Default
		Field	Metric	Laboratory	
9	ANGLE	ANGLE is a real positive number that defines the angle of influence, that is the angular connection between the aquifer and the hydrocarbon reservoir. A value of 360° degrees, the default value, indicates that the aquifer completely surrounds the hydrocarbon reservoir.			
		degrees	degrees	degrees	360.0
10	PVTNUM	PVTNUM is positive integer greater than zero and less than the NTPVT variable on the TABDIMS keyword in the RUNSPEC section, that defines the PVTW table allocated to the Carter-Tracy aquifer.			1
11	AQUTAB	AQUTAB is positive integer greater than zero and less than the NIFTBL variable as declared on the AQUDIMS keyword in the RUNSPEC section, that defines the AQUTAB table allocated to this Carter-Tracy aquifer. The default value of one sets the internal infinite acting Carter-Tracy aquifer influence table not the first table in the AQUTAB keyword in the PROPS section The first table in the AQUTAB keyword is considered to be table number two. The internal table, AQUTAB equal to one, is based on the Radial Flow, Constant Pressure and Constant Terminal Rate Cases for Infinite Reservoirs (Table I) in Van Everdingen and Hurst's [118] paper.			1
12	SALTCON	SALTCON is a real positive number that defines the initial salt concentration in the aquifer, for when with simulator's Brine Model has been activated via the BRINE keyword in the RUNSPEC section. This variable is ignored by OPM Flow.			
		lb/stb	kg/sm^3	gm/scc	0.0
13	TEMP	TEMP is a real positive number that defines the initial temperature of the aquifer at DATUM for use with OPM Flow's thermal option. The THERMAL keyword in the RUNSPEC section must be activated to use this option.			
		°F	°C	°C	1*

Notes:

1) The keyword is followed by up to NANAQ records as defined on the AQUDIMS keyword in the RUNSPEC section
2) Each record is terminated by a "/" and the keyword should be terminated by a "/".

Table 6.9:AQUCT Keyword Description

Note

OPM Flow includes the infinite acting Carter-Tracy aquifer influence table as a default for table number one; thus data entered on AQUTAB keyword starts from table number two.

In order to full define a Carter-Tracy aquifer one has to define the aquifer properties via the AQUCT keyword, the Carter-Tracy influence function via the AQUTAB keyword in the PROPS section (*AQUTAB – Define Carter-Tracy Aquifer Influence Functions*), if the default infinite acting table is not being employed, and

[118] *Van Everdingen, A. & Hurst, W.,. The Application of the Laplace Transformation to Flow Problems in Reservoirs. Petroleum Transactions, AIME (December, 1949).*

finally, how the aquifer is connected to the reservoir using the AQUANCON keyword in the GRID or SOLUTION sections.

Example

Given the following grid and aquifer dimensions in the RUNSPEC section:

```
--
--       MAX     MAX     MAX
--       NDIVIX  NDIVIY  NDIVIZ
DIMENS
         20      1       5                                                    /

--       AQF     AQF     AQF     AQF     AQF     AQF     AQF     AQF
--       MXAQN   MXNAQC  NIFTBL  NRIFTB  NANAQU  NCAMAX  MXNALI  MXAAQL
AQUDIMS
         1*      1*      5       100     1       1*      1*      1*              /
```

And AQUTAB in the PROPS section

```
--
--       CARTER-TRACY AQUIFER INFLUENCE TABLES
--       (STARTS FROM TABLE NO. 2, AS DEFAULT IS TABLE NO. 1)
--
AQUTAB
--       DIMLESS    DIMLESS
--       TIME       PRESSURE
--       -------    ---------
          0.01         0.112
          0.05         0.229
          0.10         0.315
          0.15         0.376
          0.20         0.424
          0.22         0.443
          0.24         0.459
          0.26         0.476
          0.28         0.492
          0.30         0.507
          0.32         0.522
          0.34         0.536
          0.36         0.551
          0.38         0.565
          0.40         0.579
          0.42         0.593
          0.44         0.607
          0.46         0.621
          0.48         0.634
          0.50         0.648
          0.60         0.715
          0.70         0.782
          0.80         0.849
          0.90         0.915
          1.00         0.982
          2.00         1.649
          3.00         2.316
          5.00         3.649
         10.00         6.982
         20.00        13.649
         30.00        20.316
         50.00        33.649
        100.00        66.982
        200.00       133.649
```

```
        300.00      200.316
        500.00      333.649
       1000.00      666.982 /
```

The Carter-Tracy aquifer is defined in the GRID or SOLUTION sections as:

```
-- ================================================================================
--
-- SOLUTION SECTION
--
-- ================================================================================
SOLUTION
--
--                      CARTER-TRACY AQUIFER DESCRIPTION
--
--      ID   DATUM   AQF    AQF    AQF    AQF     AQF   AQF   INFL   PVT  AQU
--      NUM  DEPTH   PRESS  PERM   PORO   RCOMP   RE    DZ    ANGLE  NUM  TAB
--
AQUCT
         1   2000.0  269    100.0  0.30   3.0e-5  330   10.0  360.0   1    2   /
/
```

And the connection of the aquifer is set in the GRID or SOLUTION sections as:

```
--
--                      ANALYTIC AQUIFER CONNECTION
--
--       ID     ---------- BOX ---------   CONNECT  AQF    AQF      ADJOIN
--       NUMBER I1  I2   J1  J2   K1  K2   FACE     INFLX  MULTI    CELLS
AQUANCON
         1      1   1    1   1    1   1     J-      1.0    1.0      'NO'        /
/
```

Here one Carter-Tracy aquifer is connected to a single cell (1, 1, 1) at the J- face (or X- face) of the cell.

6.3.9 AQUNNC – DEFINE NUMERICAL AQUIFER NON-NEIGHBOR CONNECTIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The AQUNNC keyword defines Numerical Aquifer Non-Neighbor Connections.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

6.3.10 AQUNUM – DEFINE NUMERICAL AQUIFER PROPERTIES

Description

The AQUNUM keyword defines the properties of numerical aquifers, including which grid blocks in the model should be utilized as part of the numerical aquifer. Each row entry in the AQUNUM keyword defines one numerical aquifer. Note that a numerical aquifer may consists of more than one grid cell, in order to better describe the water influx from the aquifer to the grid.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	AQUID	AQUID is a positive integer greater than or equal to one and less than or equal to the maximum number of numerical aquifers as defined by the MXNAQN variable on the AQUDIMS keyword in the RUNSPEC section, that defines the aquifer to be connected to the grid.			None
2	I	A positive integer that defines the cell in the I-direction that represents the AQUID aquifer, and which must be greater than or equal to one and less than or equal to NX.			None
3	J	A positive integer that defines the cell in the J-direction that represents the AQUID aquifer, and which must be greater than or equal to one and less than or equal to NY.			None
4	K	A positive integer that defines the cell in the K-direction that represents the AQUID aquifer, and which must be greater than or equal to one and less than or equal to NZ.			None
5	AREA	AREA is a real positive value that defines the cross-sectional area of the aquifer used in calculating the aquifer connection transmissibility. The actual transmissibility between the numerical aquifer and the connected grid cell is the harmonic average of the aquifer connection transmissibility and the calculated connected cell transmissibility. The value entered for AREA does not effect the visualization of the cell in OPM ResInsight, as it is only used in calculating the transmissibility. Note that the AQUOPT1 variable on the AQUCON keyword in the GRID section allows one to use the value entered for AREA or to use the grid cell cross-sectional area instead for the transmissibility calculation.			
		ft^2	m^2	cm^2	None
6	LENGTH	LENGTH is a real positive value that defines the length of the numerical aquifer. Similar to the AREA variable, LENGTH is not constrained by the original host cell size and the value entered does not effect the visualization of the cell in OPM ResInsight.			
		feet	m	cm	None
7	PORO	PORO is a real positive number greater than zero and less than or equal to one that assigns the porosity to the numerical aquifer.			
		dimensionless	dimensionless	dimensionless	None

No.	Name	Description			Default
		Field	Metric	Laboratory	
8	PERM	PERM is a real positive number that assigns the permeability to the numerical aquifer.			
		mD	mD	mD	None
9	DATUM	DATUM is a real positive number that sets the reference datum depth of the numerical aquifer. Similar to the AREA variable, DEPTH is not constrained by the original host cell depth and the value entered does not effect the visualization of the cell in OPM ResInsight. If defaulted then the depth of cell (I, J, K) will be used			
		feet	m	cm	Cell Depth
10	PRESS	PRESS is a single positive value that defines the numerical aquifer pressure at DATUM. If PRESS is defaulted then the simulator will set the aquifer's initial reservoir pressure to be in equilibrium with the cells the aquifer is contacted to. This is the preferred manner to initialize the numerical aquifer.			
		psia	barsa	atma	1*
11	PVTNUM	PVTNUM is positive integer greater than zero and less than the NTPVT variable on the TABDIMS keyword in the RUNSPEC section, that defines the PVTW table allocated to the numerical aquifer. If defaulted then the PVT tables allocated to cell (I, J, K) will be used.			Cell PVTNUM
12	SATNUM	SATNUM is positive integer greater than zero and less than the NTSFUN variable on the TABDIMS keyword in the RUNSPEC section, that defines the saturation tables allocated to the numerical aquifer. If defaulted then the saturation tables allocated to cell (I, J, K) will be used.			Cell SATNUM

Notes:

1) Where NX, NY and NZ are the dimensions of the model as defined on the DIMENS keyword in the RUNSPEC section.
2) The keyword is followed by up to MXNAQN records as defined on the AQUDIMS keyword in the RUNSPEC section
3) Each record is terminated by a "/" and the keyword should be terminated by a "/".

Table 6.10:AQUNUM Keyword Description

Numerical aquifers ares modeled as one-dimensional, with aquifer flow assumed to be in the direction defined by LENGTH, and flux out of the aquifer is through the cross sectional area defined by AREA. Note that, all the aquifer cells must be isolated from the reservoir cells, with only the AQUCON connections connecting to the actual reservoir cells.

The values entered on the AQUNUM keyword are used to calculate the aquifer's pore volume and the transmissibility between the aquifer and the connected cell faces defined on the AQUCON keyword. Thus:

1) The aquifer's pore volume is always calculated from the data entered on the AQUNUM keyword using $Pore\ \ Volume = AREA \times LENGTH \times PORO$ and any modifications to the host cell values performed in either the GRID or EDIT sections are always ignored for cells declared as numerical aquifers cells.

2) For the transmissibility calculation either the cross-sectional area (AREA) defined on the AQUNUM keyword may be used or the connecting cell cross-sectional area by setting the AQUOPT1 variable on the AQUCON keyword.

In order to full define a numerical aquifer one has to define the aquifer properties via the AQUNUM keyword, and how the aquifer is connected to the reservoir using the AQUCON keyword in the GRID or SOLUTION sections.

Note

If the AQUCON keyword has been utilized in the run deck then OPM Flow will write the AQUIFERN array to the *.INIT file in order to visualize the aquifer connections in OPM ResInsight.

This is accomplished by setting the AQUIFERN value to 2(AQUID-1) for cells connected to aquifer AQUID. If a cell is connected to multiple numerical aquifers then AQUIFERN is summed for all aquifers connected to a cell. Note that connecting cells to multiple aquifers is best avoided.

Finally for cells representing the numerical aquifers themselves, AQUIFERN is set to minus AQUID.

Using one aquifer cell should generally be sufficient, provided the aquifer properties are constant. However, employing multiple cells may be appropriate if the aquifer properties vary with distance or depth, which is not uncommon. Secondly, having multiple cells may help to minimize throughput-related convergence problems by increasing the pore volumes exponentially away from the reservoir cells. Typically, three to five aquifer cells are employed under theses circumstances.

The aquifer pore volume can be used to define the amount of pressure support; whereas, the aquifer transmissibility will influence the responsiveness of the aquifer.

See also the AQUDIMS keyword in the RUNSPEC section that defines the numerical aquifer dimensions.

Example

Given the following grid and aquifer dimensions in the RUNSPEC section:

```
--
--        MAX      MAX      MAX
--        NDIVIX   NDIVIY   NDIVIZ
DIMENS
          96       58       28                                                  /

--        AQF      AQF      AQF      AQF      AQF      AQF      AQF      AQF
--        MXAQN    MXNAQC   NIFTBL   NRIFTB   NANAQU   NCAMAX   MXNALI   MXAAQL
AQUDIMS
          3        3        1*        1*       1*       1*       1*       1*              /
```

The following numerical aquifer definition:

```
--
--        NUMERICAL AQUIFER DESCRIPTION
--
--        ID      - LOCATION - AQF       AQF      AQF    AQF   AQF     AQF   PVT SATNUM
--        NUMBER   I1   J1   K1   AREA      LENGTH   PORO   PERM DEPTH   PRES  TAB TAB
--
AQUNUM
          1         1    3   28   5.20E03   2.0E3    0.05   0.3  1*      1*    1*  1*    /
          1         1    2   28   5.20E06   2.0E6    0.05   0.3  1*      1*    1*  1*    /
          1         1    1   28   5.20E09   2.0E9    0.05   0.3  1*      1*    1*  1*    /
/
```

defines one numerical aquifer consisting of three cells. The connection to the grid cells would take the form of:

```
--
--        NUMERICAL AQUIFER CONNECTIONS
--
--       ID     ---------- BOX ---------   CONNECT  TRANS  TRANS  ADJOIN
--       NUMBER I1  I2   J1  J2   K1  K2   FACE     MULT   OPTN   CELLS
AQUCON
         1       1   1    4  58   28   28  'K+'     1.3    1*     1*          /
         1       2  96    1  58   28   28  'K+'     1.3    1*     1*          /
/
```

that creates a basal aquifer[119].

[119] *Basal Aquifer: An aquifer located at the bottom of a geologic unit.*

6.3.11 AUTOCOAR - DEFINE AUTO REFINEMENT GRID COARSEN AREA

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The AUTOCOAR keyword defines an area in the global grid that should be coarsen for when the AUTOREF keyword has been declared in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

6.3.12 BCCON – DEFINE BOUNDARY CONDITIONS CONNECTIONS

Description

The BCCON keyword defines the grid connections for the boundary conditions.

Together the BCCON and BCPROP keywords define the boundary conditions for the model, and can be used to set boundary conditions for when external influx or efflux volumes are influencing the reservoir pressure and production history. For example, when the average reservoir pressure remains constant throughout the production period due to water influx, or gas migration from an external source.

Note

This is an OPM Flow specific keyword and will therefore cause an error in the commercial simulator.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	INDEX	A positive integer that identifies the boundary condition.			None
2	I1	A positive integer that defines the lower bound of the grid in the I-direction for which the boundary conditions are to be applied, must be greater than or equal 1 and less than or equal to I2 and NX.			1
2	I2	A positive integer that defines the upper bound of the grid in the I-direction for which the boundary conditions are to be applied, must be greater than or equal to I1 and less than or equal to NX			NX
3	J1	A positive integer that defines the lower bound of the grid in the J-direction for which the boundary conditions are to be applied, must be greater than or equal 1 and less than or equal to J2 and NY.			1
4	J2	A positive integer that defines the upper bound of the grid in the J-direction for which the boundary conditions are to be applied, must be greater than or equal to J1 and less than or equal to NY.			NY
5	K1	A positive integer that defines the lower bound of the grid in the K-direction for which the boundary conditions are to be applied, must be greater than or equal to one and less than or equal to K2 and NZ.			1
6	K2	A positive integer that defines the upper bound of the grid in the K-direction for which the boundary conditions are to be applied, must be greater than or equal to K1 and less than or equal to NZ.			NZ
7	DIRECT	A character string that defines the direction to apply the boundary conditions, and should be set to one of the following X, Y, or Z for the positive direction, or X-, Y- or Z- for the negative direction.			None

Notes:

1) Where NX, NY and NZ are the dimensions of the model as defined on the DIMENS keyword in the RUNSPEC section.

2) Each record must be terminated by a "/" and the keyword is terminated by a "/".

Table 6.11: BCCON Keyword Description

See also the AQUFLUX keyword that is supported by OPM Flow in both the SOLUTION and SCHEDULE sections, to define a constant flux analytical aquifer.

If the BCCON and BCPROP keywords are not present in the input deck, then the boundary conditions for the model are set to be no flow, which is the normal behavior in both OPM Flow and the commercial simulator.

The BC keyword has been replaced by the BCCON and BCPROP keywords.

Examples

The first example shows how to set a constant pressure boundary using TYPE equal to FREE:

```
--
--       DEFINE BOUNDARY CONDITIONS CONNECTION (OPM FLOW SOLUTION GRID KEYWORD)
--
--INDEX  ---------- BOX ---------   BC
--       I1  I2   J1  J2   K1  K2   DIRC
BCCON
1        1   1    1   1*   1   1*   X-    /
2        1   1*   1   1    1   1*   Y     /
/

--
--       DEFINE BOUNDARY CONDITIONS PROPERTIES (OPM FLOW SOLUTION GRID KEYWORD)
--
--INDEX  BC         BC     BC     BC     BC
--       TYPE       PHASE  RATE   PRESS  TEMP
BCPROP
1        FREE       1*     1*     1*     1*    /
2        FREE                                 /
/
```

With this option it is only necessary to define the boundary cells and all the other parameters (PHASE, RATE, PRESS, and TEMP) can be defaulted, as they are ignored when TYPE equals FREE.

The next example is based on NX, NY and NZ equal to 20, 1, 10 respectively, on the DIMENS keyword in the RUNSPEC section, and shows how different boundary types can be assigned to different parts of the model.

```
--
--       DEFINE BOUNDARY CONDITIONS CONNECTION (OPM FLOW SOLUTION GRID KEYWORD)
--
--INDEX  ---------- BOX ---------   BC
--       I1  I2   J1  J2   K1  K2   DIRC
BCCON
1        1   1    1   1    1   10   X-    /
2        20  20   1   1    1   10   X     /
/

--
--       DEFINE BOUNDARY CONDITIONS PROPERTIES (OPM FLOW SOLUTION GRID KEYWORD)
--
--INDEX  BC         BC     BC     BC     BC
--       TYPE       PHASE  RATE   PRESS  TEMP
BCPROP
1        RATE       GAS    1*     256.0  100.0 /
2        FREE       4*                         /
/
```

The last example shows how the DIRICHLET boundary condition option may be used:

```
--
--       DEFINE BOUNDARY CONDITIONS CONNECTION (OPM FLOW SOLUTION GRID KEYWORD)
--
--INDEX  ---------- BOX ---------   BC
--       I1  I2   J1  J2   K1  K2   DIRC
BCCON
1        1   1    1   1*   1   1*   X-    /
2        1   1*   1   1    1   1*   Y     /
/

--
--       DEFINE BOUNDARY CONDITIONS PROPERTIES (OPM FLOW SOLUTION GRID KEYWORD)
--
--INDEX  BC         BC     BC    BC     BC
--       TYPE       PHASE  RATE  PRESS  TEMP
BCPROP
1        DIRICHLET  WAT    1*    256.0  100.0 /
2        DIRICHLET  WAT    1*    1*     100.0 /
/
```

Here, the first line sets both the pressure and temperature at the boundary, and the second line defaults the pressure entry, so that the simulator calculated initial boundary pressure will be used.

6.3.13 BOUNDARY – DEFINE A BOUNDARY BOX FOR PRINTING

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The BOUNDARY keyword defines a rectangular grid for printing various arrays to the output print file (*.PRT); thus, avoiding printing all the elements in the selected array.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

6.3.14 BOX - DEFINE A RANGE OF GRID BLOCKS TO ENTER PROPERTY DATA

Description

BOX defines a range of grid blocks for which subsequent data is assigned for all the cells in the defined BOX. Values are set for cells within the defined box grid using natural reading order, initially along the I-direction then J-direction and finally the K-direction. If fewer values are assigned than exist within the defined block space, then subsequent values are set starting from the next block that was not previously assigned for that property. This is the same behavior as applies to setting grid properties for an unboxed grid.

Note that the BOX grid is reset by the keyword ENDBOX by resetting the current defined BOX to be the whole grid. The keyword can be used for any array and for all grid types.

No.	Name	Description	Default
1	I1	A positive integer that defines the lower bound of the array in the I-direction to be modified must be greater than or equal 1 and less than or equal to I2 and NX.	1
2	I2	A positive integer that defines the upper bound of the array in the I-direction to be modified must be greater than or equal to I1 and less than or equal to NX	NX
3	J1	A positive integer that defines the lower bound of the array in the J-direction to be modified must be greater than or equal 1 and less than or equal to J2 and NY.	1
4	J2	A positive integer that defines the upper bound of the array in the J-direction to be modified must be greater than or equal to J1 and less than or equal to NY.	NY
5	K1	A positive integer that defines the lower bound of the array in the K-direction to be modified must be greater than or equal to one and less than or equal to K2 and NZ.	1
6	K2	A positive integer that defines the upper bound of the array in the K-direction to be modified must be greater than or equal to K1 and less than or equal to NZ.	NZ

Notes:

1) Where NX, NY and NZ are the dimensions of the model as defined on the DIMENS keyword in the RUNSPEC section.
2) The keyword is terminated by a "/".

Table 6.12: BOX Keyword Description

See also the ADD, COPY, ENDBOX, EQUALS, and MULTIPLY keywords can also be used to enter data in a subset of the model.

Examples

```
--
--        DEFINE A BOX GRID FOR THE BOTTOM LAYER OF A 100 X 100 X 20 MODEL
--
--        ---------- BOX ---------
--        I1  I2   J1  J2   K1  K2
BOX
          1*  1*   1*  1*   20  20 / SELECT THE BOTTOM LAYER
--
--        DEFINE THE POROSITY AND OTHER PROPERTIES ON THE BOX GRID
--
PORO
          10000*0.300
/
PERMX
          5000*100.0   5000*75.0
/
NTG
          10000*0.500
/
--
--        RESET THE INPUT BOX TO BE THE FULL MODEL
--
ENDBOX
```

The above example set the BOX grid to be the last layer in the model which means that 100 x 100, that is 10,000 values need to entered for each property.

Alternatively, one could use the EQUALS keyword to accomplish the same thing.

```
-- -- ARRAY    CONSTANT --  ---------- BOX ---------
--                         I1  I2   J1  J2   K1  K2
EQUALS
   'PORO'      0.3000       1*  1*   1*  1*   20  20 / PORO  TO 0.30 IN LAYER 20
   'PERMX'     100.00       1*  1*   1   50   20  20 / PERMX TO 100. IN LAYER 20
   'PERMX'      75.00       1*  1*   51  100  20  20 / PERMX TO 75.0 IN LAYER 20
   'NTG'       0.5000       1*  1*   1*  1*   20  20 / NTG   TO 0.50 IN LAYER 20
/
```

Note

It is good practice to always use the ENDBOX keyword to reset the input back to the full grid when all the modifications for a sub-grid have been completed.

6.3.15 BTOBALFA – DUAL POROSITY MATRIX TO FRACTURE MULTIPLIER (ALL CELLS)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The BTOBALFA keyword defines a dual porosity matrix to fracture multiplier that is applied to all cells, for when the Dual Porosity model has been activated by either the DUALPORO or the DUALPERM keywords in the RUNSPEC section.

See also the BTOBALFV keyword in the GRID section that applies a multipliers to individual cells.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

6.3.16 BTOBALFV – DUAL POROSITY MATRIX TO FRACTURE MULTIPLIER (INDIVIDUAL CELLS)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The BTOBALFV keyword defines a dual porosity matrix to fracture multiplier that is applied to individual cells, for when the Dual Porosity model has been invoked by either the DUALPORO or the DUALPERM keywords in the RUNSPEC section.

See also the BTOBALFAV keyword in the GRID section that applies a constant multiplier to all cells.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

6.3.17 CARFIN – DEFINE A CARTESIAN LOCAL GRID REFINEMENT

Description

CARFIN defines a Cartesian Local Grid Refinement ("LGR") in a cell or a group of cells in the host grid, for when LGRs have been activated for the input deck using the LGR keyword in the RUNSPEC section. The keyword marks the start of an LGR description section and all subsequent keywords between the CARFIN and ENDFIN keywords are deemed to be associated with the current LGR and not the host grid.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description	Default
1	LGRNAME	A character string of up to eight characters in length that defines the LGR name for which the LGR is being being defined.	None
2	I1	A positive integer that defines the lower index of the global or host grid in the I-direction to be refined; must be greater than or equal 1 and less than or equal to I2 and NX on the DIMENS keyword in the RUNSPEC section.	None
3	I2	A positive integer that defines the upper index of the global or host grid in the I-direction to be refined; must be greater than or equal I and II, and less than or equal to NX on the DIMENS keyword in the RUNSPEC section.	None
4	J1	A positive integer that defines the lower index of the global or host grid in the J-direction to be refined; must be greater than or equal 1 and less than or equal to J2 and NY on the DIMENS keyword in the RUNSPEC section.	None
5	J2	A positive integer that defines the upper index of the global or host grid in the J-direction to be refined; must be greater than or equal 1 and JI, and less than or equal to NY on the DIMENS keyword in the RUNSPEC section.	None
6	K1	A positive integer that defines the lower index of the global or host grid in the K-direction to be refined; must be greater than or equal 1 and less than or equal to K2 and NZ on the DIMENS keyword in the RUNSPEC section.	None
7	K2	A positive integer that defines the upper index of the global or host grid in the K-direction to be refined; must be greater than or equal 1 and KI, and less than or equal to NZ on the DIMENS keyword in the RUNSPEC section.	None
8	NX	A positive integer value that defines the number of LGR grid blocks in the x direction for Cartesian grids or the number of grid blocks in the r direction for radial grids	None
9	NY	A positive integer value that defines the number of LGR grid blocks in the y direction for Cartesian grids or the number of grid blocks in the theta direction for radial grids.	None
10	NZ	A positive integer value that defines the number of LGR grid blocks in the z direction for both Cartesian and radial grids.	None
11	MXWELS	A positive integer defining the maximum number of wells contained in this LGR.	None

No.	Name	Description	Default
12	HOSTNAME	A character string of up to eight characters in length that defines the host grid name for nested refinements. The default value of "GLOBAL" sets the host name to the global grid, that is for a conventional LGR. A nested refinement is when the HOSTNAME is a previously declared LGR for which the current LGR is specifying a further LGR refinement.	GLOBAL
Notes: 1) The keyword is terminated by a "/".			

Table 6.13: CARFIN Keyword Description

Note that if the Dual Porosity option has be activated by either the DUALPORO or DUALPERM keywords in the RUNSPEC section, then the host grid definition (I1-I2, J1-J2, K1-K2) applies only to the matrix cells; however, the LGR NZ parameter in this case must include the fracture blocks, similar to the NZ parameter on the DIMENS keyword. This means that all property data should be entered for both the matrix and fracture cells in the LGR description.

Example

The example below defines an LGR in the global grid, named LGR-OP01 with a maximum of one well allowed in the LGR.

```
--
--        CARFIN LGR GRID COMMANDS
--
--        LGR         ----- HOST GRID ------    -- CARFIN GRID --   MAX      HOST
--        NAME        I1   I2   J1   J2   K1   K2      NX     NY     NZ    WELLS    NAME
CARFIN
          LGR-OP01    24   24   87   87    1   50       3      3     50      1      GLOBAL /

ENDFIN
```

Here the one global cell in the areal plane (24, 87) is divided into three LGR cells in the x-direction and three cells in the y-direction. Since no other property data is given, then the LGR cells take their properties from the host grid, that is the global grid.

6.3.18 COALNUM – DEFINE THE COAL REGION NUMBERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The COALNUM keyword defines the coal region numbers for each grid block used with the Coal Bed Methane option ("CBM"). OPM Flow does not have a CBM option; however, the keyword is documented here for completeness.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description	Default
1	COALNUM	COALNUM defines an array of positive integers assigning a grid cell to a particular coal region. The maximum number of COALNUM regions is set by the NTCREG variable on REGDIMS keywords in the RUNSPEC section.	1

Notes:

1) A total of NX x NY x NZ integer values, as defined by the DIMENS keyword in the RUNSPEC section, must be specified for the array.
2) If a cell is not assigned a COALNUM region then the default value of 1 will be used.
3) COALNUM value of 0 sets the cell be a non-coal region.
4) The keyword is terminated by a "/".

Table 6.14: COALNUM Keyword Description

Example

The example below sets three COALNUM regions for a 4 x 5 x 2 model.

```
--
--       DEFINE COALNUM REGIONS FOR ALL CELLS
--
COALNUM
         2  2  1  1  2  2  1  1  1  1  1  1  1  1  1  1  1  1  1  1
         3  3  1  1  3  3  1  1  1  1  1  1  1  1  1  1  1  1  1  1
/
```

The above will no effect in an OPM Flow input deck.

6.3.19 COARSEN – DEFINE GRID COARSENING CELLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The COARSEN keyword defines how a set of cells should be coarsened for when the Local Grid Refinement ("LGR") option has been activated by LGR keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

6.3.20 COLLAPSE – DEFINE COMPRESSED VERTICAL EQUILIBRIUM CELLS

Description

The COLLAPSE keyword defines the which cells can be collapsed in a collapsed Vertical Equilibrium ("VE") run when the VE option has been invoked via the VE keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

6.3.21 COLUMNS – DEFINE INPUT FILE COLUMN MARGINS

The COLUMNS keyword defines the input file column margins; characters outside the margins are ignored by the input parser.

See *COLUMNS – Define Input File Column Margins* in the GLOBAL section for a full description.

6.3.22 COORD – DEFINE A SET OF COORDINATES LINES FOR A RESERVOIR GRID

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

COORD defines a set of coordinate lines or pillars for a reservoir grid via an array. A total of 6 x (NX+1) x (NY+1) values must be specified for each coordinate data set (or reservoir). For multiple reservoirs, where NUMRES is greater than one, there must be 6 x (NX+1) x (NY+1) x NUMRES values. In OPM Flow NUMRES can only be set to one.

For Cartesian geometry, each line is defined by the (x, y, z) coordinates of two distinct points on the line. The lines are entered with I cycling fastest then J. For radial geometry, each line is defined by the (r, theta, z) coordinates of two distinct points on the line. The lines are entered with R cycling fastest then THETA.

The keyword can only be used with Irregular Corner-Point Grids.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	X1-Column	Top X coordinate			
2	Y1-Column	Top Y coordinate			
3	Z1-Column	Top Z coordinate			
4	X2-Column	Base X coordinate			
5	Y2-Column	Base Y coordinate			
6	Z2-Column	Base Z coordinate			
		feet	metres	cm	None

Notes:

1) THETA values are in degrees for all units.
2) Regular Cartesian and Radial grid keywords cannot be used with this keyword, that is: DR, DRV, DTHETA, DTHETAV, DX, DXV, DY, DYV, DZ, INRAD, and TOPS.
3) The keyword is terminated by a "/".

Table 6.15: COORD Keyword Description

See also the SPECGRID, COORDSYS and ZCORN keywords to fully define an Irregular Corner-Point Grid.

Example

```
--
--       SPECIFY VERTICAL COORDINATE LINES FOR A REGULAR 3 x 2 GRID
--       (DX = 100 and DY = 200)
--
--       X1     Y1    Z1      X2     Y2    Z2
--       ---    ---   ----    ---    ---   ----
COORD
           0      0   1000      0      0   5000
         100      0   1000    100      0   5000
         200      0   1000    200      0   5000
         300      0   1000    300      0   5000
           0    200   1000      0    200   5000
         100    200   1000    100    200   5000
         200    200   1000    200    200   5000
         300    200   1000    300    200   5000
           0    400   1000      0    400   5000
         100    400   1000    100    400   5000
         200    400   1000    200    400   5000
         300    400   1000    300    400   5000
/
```

The above example defines vertical coordinate lines for a regular 3 by 2 grid with DX equal to 100 and DY equal to 200.

6.3.23 COORDSYS – DEFINE COORDINATE GRID OPTIONS

Description

This keyword sets various options for when multiple grid systems are being used, as declared by the NUMRES keyword in the RUNSPEC section. OPM Flow does not support multiple grid systems. The keyword is also used to stipulate for radial grids if the completion of the circle in the THETA direction should be implemented using non-neighbor connections.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

No.	Name	Description	Default
1	K1	A positive integer that defines the lower bound of the array in the K-direction for the given grid system.	None
2	K2	A positive integer that defines the upper bound of the array in the K-direction for the given grid system.	None
3	COMPLETE	COMPLETE is a defined character string that determines for radial grids if the circle should be completed in THETA direction, and should be set to COMP to complete the circle, or INCOMP for not completing the circle.	INCOMP
4	CONNECT	A defined character string that declares how the reservoir below should be connected to the given reservoir, and should be set to JOIN to connect the two reservoirs by calculating the inter-reservoir transmissibilities, or SEPARATE to isolate the reservoirs.	SEPARATE
5	R1	R1 is a positive integer defining the lower reservoir unit that is is connected to the given reservoir unit.	Current Reservoir Record
6	R2	R2 is a positive integer defining the upper reservoir unit that is is connected to the given reservoir unit.	

Notes:

1) The DIMENS keyword defines the dimensions of the model (NX, NY and NZ) and the NUMRES keyword the number of multiple grid systems (reservoirs), both keywords are in the RUNSPEC section.
2) Each record must be terminated by a "/" and there must be exactly NUMRES records entered for the keyword.
3) There is no "/" terminator for this keyword.

Table 6.16: COORDSYS Keyword Description

Example

```
--
--       DEFINE COORDINATE GRID OPTIONS
--
--       K1     K2     COMP    CONNECT  LOWER  UPPER
--       Layer  Layer  CIRCLE  RES      RES    RES
COORDSYS
         1      1      COMP                            /
/
```

The above example connects the circle in the THETA direction for the RADIAL model, for when the number of grids have been set to one via the NUMRES keyword in the SCHEDULE section.

6.3.24 COPY – COPY ARRAY DATA TO ANOTHER ARRAY

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The COPY keyword copies an array (or part of an array) to another array or part of an array. The arrays can be real or integer depending on the array type; however, the arrays that can be operated on is dependent on which section the COPY keyword is being used.

No.	Name	Description	Default
1	ARRAY-1	The name of the array to be copied from. This is the keyword name identifying the property and is up to eight characters in length and enclosed in quotes.	None
2	ARRAY-2	The name of the array to be copied to. This is the keyword name identifying the property and is up to eight characters in length and enclosed in quotes.	None
3	I1	A positive integer that defines the lower bound of the array in the I-direction to be modified must be greater than or equal 1 and less than or equal to I2 and NX.	1
4	I2	A positive integer that defines the upper bound of the array in the I-direction to be modified must be greater than or equal to I1 and less than or equal to NX	NX
5	J1	A positive integer that defines the lower bound of the array in the J-direction to be modified must be greater than or equal 1 and less than or equal to J2 and NY.	1
6	J2	A positive integer that defines the upper bound of the array in the J-direction to be modified must be greater than or equal to J1 and less than or equal to NY.	NY
7	K1	A positive integer that defines the lower bound of the array in the K-direction to be modified must be greater than or equal to one and less than or equal to K2 and NZ.	1
8	K2	A positive integer that defines the upper bound of the array in the K-direction to be modified must be greater than or equal to K1 and less than or equal to NZ.	NZ

Notes:

1) Where NX, NY and NZ are the dimensions of the model as defined on the DIMENS keyword in the RUNSPEC section.
2) Each record must be terminated by a "/" and the keyword is terminated by a "/".

Table 6.17: COPY Keyword Description

The applicable arrays for each section are defined in Table 6.18 on the following page.

COPY Keyword And Variable Options By Section						
GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
DX	DEPTH	SWL	ENDNUM	PRESSURE		
DY	PORV	SWCR	EQLNUM	SWAT		
DZ	TRANX	SWU	FIPNUM	SGAS		
PERMX	TRANY	SGL	IMBNUM	RV		
PERMY	TRANZ	SGCR	MISCNUM	RS		
PERMZ	DIFFX	SGU	PVTNUM	TBLK		
MULTX	DIFFY	KRW	ROCKNUM	GI		
MULTY	DIFFZ	KRO	SATNUM	OILAPI		
MULTZ	TRANR	KRG	WH2NUM	SALT		
DR	TRANTHT	PCG		GASCONC		
THETA	DIFFR	PCW		SOLVCONC		
PERMR	DIFFTHT			SOLVFRAC		
PERMTHT				SFOAM		
DZNET				SPOLY		
PORO						
NTG						
FLUXNUM						
MULTNUM						
MPANUM						
DIFFX						
DIFFY						
DIFFZ						
DIFFR						
DIFFTHT						

Table 6.18: COPY Keyword Applicable Arrays by Section

Example

```
--
--        SOURCE       DESTIN.       ---------- BOX ---------
--                                    I1  I2   J1  J2   K1  K2
COPY
          PERMX        PERMY         1*  1*   1*  1*   1*  1* / CREATE PERMY
          PERMX        PERMZ         1*  1*   1*  1*   1*  1* / CREATE PERMZ
/

--
--        ARRAY        CONSTANT      ---------- BOX ---------
--                                    I1  I2   J1  J2   K1  K2
MULTIPLY
          PERMZ        0.50000       1*  1*   1*  1*   1*  1* / PERMZ * 0.5
/
```

The above example copies PERMX array to the PERMY and PERMZ arrays in the GRID section for all grid blocks in the model. The PERMZ array is then multiplied by 0.5 for all grid blocks in the model.

6.3.25 COPYBOX – COPY ARRAY DATA DEFINED BY A BOX

Description

The COPYBOX keyword copies an array (or part of an array) to another part of the same array. The array can be real or integer depending on the array type; however, the array that can be operated on is dependent on which section the COPYBOX keyword is being used.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description	Default
1	ARRAY-1	The name of the array to be copied This is the keyword name identifying the property and is up to eight characters in length and enclosed in quotes.	None
2	I1	A positive integer that defines the SOURCE lower bound of the array in the I-direction to be modified must be greater than or equal 1 and less than or equal to I2 and NX.	1
3	I2	A positive integer that defines the SOURCE upper bound of the array in the I-direction to be modified must be greater than or equal to I1 and less than or equal to NX	NX
4	J1	A positive integer that defines the SOURCE lower bound of the array in the J-direction to be modified must be greater than or equal 1 and less than or equal to J2 and NY.	1
5	J2	A positive integer that defines the SOURCE upper bound of the array in the J-direction to be modified must be greater than or equal to J1 and less than or equal to NY.	NY
6	K1	A positive integer that defines the SOURCE lower bound of the array in the K-direction to be modified must be greater than or equal to one and less than or equal to K2 and NZ.	1
7	K2	A positive integer that defines the SOURCE upper bound of the array in the K-direction to be modified must be greater than or equal to K1 and less than or equal to NZ.	NZ
8	I3	A positive integer that defines the DESTINATION lower bound of the array in the I-direction to be modified must be greater than or equal 1 and less than or equal to I2 and NX.	1
9	I4	A positive integer that defines the DESTINATION upper bound of the array in the I-direction to be modified must be greater than or equal to I1 and less than or equal to NX	NX
10	J3	A positive integer that defines the DESTINATION lower bound of the array in the J-direction to be modified must be greater than or equal 1 and less than or equal to J2 and NY.	1
11	J4	A positive integer that defines the DESTINATION upper bound of the array in the J-direction to be modified must be greater than or equal to J1 and less than or equal to NY.	NY

No.	Name	Description	Default
12	K3	A positive integer that defines the DESTINATION lower bound of the array in the K-direction to be modified must be greater than or equal to one and less than or equal to K2 and NZ.	1
13	K4	A positive integer that defines the DESTINATION upper bound of the array in the K-direction to be modified must be greater than or equal to KI and less than or equal to NZ.	NZ

Notes:

1) Where NX, NY and NZ are the dimensions of the model as defined on the DIMENS keyword in the RUNSPEC section.

2) Each record must be terminated by a "/" and the keyword is terminated by a "/".

Table 6.19: COPYBOX Keyword Description

Note that the SOURCE and DESTINATION arrays must be of the same size in all dimensions and the applicable arrays for each section are defined in Table 6.20.

COPYBOX Keyword And Variable Options By Section						
GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
DX		SWL	ENDNUM			
DY		SWCR	EQLNUM			
DZ		SWU	FIPNUM			
PERMX		SGL	IMBNUM			
PERMY		SGCR	MISCNUM			
PERMZ		SGU	PVTNUM			
MULTX		KRW	ROCKNUM			
MULTY		KRO	SATNUM			
MULTZ		KRG	WH2NUM			
DR		PCG				
THETA		PCW				
PERMR						
PERMTHT						
DZNET						
PORO						
NTG						
FLUXNUM						
MULTNUM						
MPANUM						
DIFFX						
DIFFY						
DIFFZ						
DIFFR						
DIFFTHT						

Table 6.20: COPYBOX Keyword Applicable Arrays by Section

Example

```
--
--        SOURCE    ------ SOURCE BOX ------    ---- DESTINATION BOX ----
--        ARRAY     I1  I2   J1  J2   K1  K2    I1  I2   J1  J2   K1  K2
COPYBOX
           PORO     1*  1*   1*  1*   12  14    1*  1*   1*  1*   15  17 / PORO
           PERMX    1*  1*   1*  1*   12  14    1*  1*   1*  1*   15  17 / PERMX
/
```

The above example copies all the PORO and PERMX values in layers 12 to 14 to layers 15 and 17.

6.3.26 COPYREG – COPY AN ARRAY TO ANOTHER ARRAY BASED ON A REGION NUMBER

Description

The COPYREG keyword copies a specified array or part of an array based on cells with a specific region number to another array. The region number array can be FLUXNUM, MULTNUM or OPERNUM and these arrays must be defined and be available before the COPYREG keyword is read by the simulator. The property arrays can be real or integer depending on the property array type; however, the property arrays that can be operated on is dependent on which section the COPYREG keyword is being used.

No.	Name	Description	Default
1	ARRAY-1	The name of the array to be copied from. This is the keyword name identifying the property and is up to eight characters in length and enclosed in quotes.	None
2	ARRAY-2	The name of the array to be copied to. This is the keyword name identifying the property and is up to eight characters in length and enclosed in quotes.	None
3	REGION NUMBER	Integer REGION NUMBER is the region for which the array data in (1) should be copied to array data in (2).	None
4	REGION ARRAY	The REGION ARRAY to use for selecting the REGION NUMBER in (3) for selecting the data to be copied. REGION ARRAY can have the following values: 1) F for the FLUXNUM array 2) M for the MULTNUM array 3) O for the OPERNUM array	M
Notes: 1) Where the REGION NUMBER should be less than or equal to the maximum number of regions as defined on the REGDIMS keyword for the FIPNUM and OPERNUM arrays or the GRIDOPTS keyword for the MULTNUM array in the RUNSPEC section. 2) Each record must be terminated by a "/" and the keyword is terminated by a "/".			

Table 6.21: COPYREG Keyword Description

The applicable arrays for each section are defined in Table 6.22.

COPYREG Keyword And Variable Options By Section						
GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
DX	DEPTH	SWL	ENDNUM	PRESSURE		
DY	PORV	SWCR	EQLNUM	SWAT		
DZ	TRANX	SWU	FIPNUM	SGAS		
PERMX	TRANY	SGL	IMBNUM	RV		
PERMY	TRANZ	SGCR	MISCNUM	RS		
PERMZ	DIFFX	SGU	PVTNUM	TBLK		

COPYREG Keyword And Variable Options By Section						
GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
MULTX	DIFFY	KRW	ROCKNUM	GI		
MULTY	DIFFZ	KRO	SATNUM	OILAPI		
MULTZ	TRANR	KRG	WH2NUM	SALT		
DR	TRANTHT	PCG		GASCONC		
THETA	DIFFR	PCW		SOLVCONC		
PERMR	DIFFTHT			SOLVFRAC		
PERMTHT				SFOAM		
DZNET				SPOLY		
PORO						
NTG						
FLUXNUM						
MULTNUM						
MPANUM						
DIFFX						
DIFFY						
DIFFZ						
DIFFR						
DIFFTHT						

Table 6.22: COPYREG Keyword Applicable Arrays by Section

Example

```
--
--        COPY AN ARRAY TO ANOTHER ARRAY BASED ON A REGION NUMBER
--
--        ARRAY     ARRAY     REGION   REGION ARRAY
--        FROM      TO        NUMBER    M / F / O
COPYREG
          PERMX     PERMY     1         M                  / COPY PERMX TO PERMY
          PERMX     PERMZ     1         M                  / COPY PERMX TO PERMZ
/
--
--        NOW RESET PERMZ BASED ON THE MULTNUM REGION NUMBER
--
--
--        MULTIPLY AN ARRAY BY A CONSTANT BASED ON A REGION NUMBER
--
--        ARRAY     CONSTANT  REGION   REGION ARRAY
--                  VALUE     NUMBER    M / F / O
MULTIREG
          PERMX     0.95      1         M                  /
/
```

The above example first copies the PERMX property array for region number one to the PERMY and PERMZ property arrays for region one using the MULTNUM array to define the region numbers. After which PERMZ property array for region one is multiplied by 0.5 using the MULTIREG keyword.

6.3.27 CRITPERM – DEFINE MINIMUM PERMEABILITY FOR VERTICAL EQUILIBRIUM GRID CELL COMPRESSION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The CRITPERM keyword is used to define the minimum permeability for Vertical Equilibrium("VE") grid cell compression, for when the Vertical Equilibrium formulation has been activated by the VE keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

6.3.28 DEBUG – DEFINE THE DEBUG DATA TO BE PRINTED TO FILE

This keyword defines the debug data to be written to the debug file (*.DBG). This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

See *DEBUG – Define the Debug Data to be Printed to File* in the GLOBAL section for a full description.

6.3.29 DIFFMR – DEFINE GRID BLOCK RADIAL DIRECTION DIFFUSIVITY MULTIPLIERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The DIFFMR keyword defines the radial direction diffusivity multipliers for cells in the current input box for when the Diffusivity option has been activated by the DIFFUSE keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

6.3.30 DIFFMR- – DEFINE GRID BLOCK NEGATIVE RADIAL DIRECTION DIFFUSIVITY MULTIPLIERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The DIFFMR- keyword defines the negative radial direction diffusivity multipliers for cells in the current input box for when the Diffusivity option has been activated by the DIFFUSE keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

6.3.31 DIFFMTH- – DEFINE GRID BLOCK NEGATIVE THETA DIRECTION DIFFUSIVITY MULTIPLIERS

Description

The DIFFMR- keyword defines the negative theta direction diffusivity multipliers for cells in the current input box for when the Diffusivity option has been activated by the DIFFUSE keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

6.3.32 DIFFMTHT – DEFINE GRID BLOCK THETA DIRECTION DIFFUSIVITY MULTIPLIERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The DIFFMTHT keyword defines the theta direction diffusivity multipliers for cells in the current input box for when the Diffusivity option has been activated by the DIFFUSE keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

6.3.33 DIFFMX – DEFINE GRID BLOCK X-DIRECTION DIFFUSIVITY MULTIPLIERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The DIFFMX keyword defines the x-direction diffusivity multipliers for cells in the current input box for when the Diffusivity option has been activated by the DIFFUSE keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

6.3.34 DIFFMX- – DEFINE GRID BLOCK NEGATIVE X-DIRECTION DIFFUSIVITY MULTIPLIERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The DIFFMX- keyword defines the negative x-direction diffusivity multipliers for cells in the current input box for when the Diffusivity option has been activated by the DIFFUSE keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

6.3.35 DIFFMY – DEFINE GRID BLOCK Y-DIRECTION DIFFUSIVITY MULTIPLIERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The DIFFMY keyword defines the y-direction diffusivity multipliers for cells in the current input box for when the Diffusivity option has been activated by the DIFFUSE keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

6.3.36 DIFFMY- – DEFINE GRID BLOCK NEGATIVE Y-DIRECTION DIFFUSIVITY MULTIPLIERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The DIFFMY- keyword defines the negative y-direction diffusivity multipliers for cells in the current input box for when the Diffusivity option has been activated by the DIFFUSE keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

6.3.37 DIFFMZ – DEFINE GRID BLOCK Z-DIRECTION DIFFUSIVITY MULTIPLIERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The DIFFMZ keyword defines the z-direction diffusivity multipliers for cells in the current input box for when the Diffusivity option has been activated by the DIFFUSE keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

6.3.38 DIFFMZ- – DEFINE GRID BLOCK NEGATIVE Z-DIRECTION DIFFUSIVITY MULTIPLIERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The DIFFMZ- keyword defines the negative z-direction diffusivity multipliers for cells in the current input box for when the Diffusivity option has been activated by the DIFFUSE keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

6.3.39 DOMAINS – DEFINE THE PARALLEL DOMAIN PROPERTIES

Description

The DOMAINS keyword defines the parallel domain properties for when parallel processing has been invoked by the PARALLEL keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

See section *2.2 Running OPM Flow 2023-10 From The Command Line* on how to run OPM Flow in parallel mode.

6.3.40 DPGRID – ACTIVATE THE MATRIX CELL TO FRACTURE CELL OPTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The DPGRID keyword activates the matrix cell to fracture cell option for dual porosity runs for when a Dual Porosity model has been activated by either the DUALPORO or DUALPERM keywords in the RUNSPEC section. The keyword allows for only the matrix grid data to be entered and the missing fracture cells are set to the inputted matrix cells.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

6.3.41 DPNUM – DEFINE DUAL AND SINGLE POROSITY GRID BLOCK ARRAY

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

In dual porosity runs only, that is not dual permeability runs, the DPNUM keyword defines which wells should be treated as single porosity cells and which cells should be treated as dual porosity cells, for when the Dual Porosity model has been activated by the DUALPORO keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

6.3.42 DR - DEFINE THE SIZE OF GRID BLOCKS IN THE R DIRECTION FOR ALL CELLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

DR defines the size of all grid blocks in the R direction via an array for each cell in the model. The RADIAL or SPIDER keyword in the RUNSPEC section should be activated to indicate that radial or spider geometry is being used.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	DR	DR is an array of real numbers describing the cell size in the R direction for each cell in the model in a radial grid. Repeat counts may be used, for example 10*100.0.			
		feet	m	cm	None

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) The first layer (K=1) must always be fully defined; however, subsequent layers may be default and if defaulted they will take the values from the layer above.
3) The keyword is terminated by a "/".

Table 6.23: DR Keyword Description

Note that the SPIDER keyword activates OPM Flow's radial grid geometry option for the model. This keyword will create a spiderweb-shaped grid based on a corner-point grid using the standard radial grid keywords: INRAD, DRV, DTHETAV, DZ/DZV etc. in the GRID the section. A spider grid can be viewed in 3D in OPM ResInsight unlike radial grids that cannot be viewed in the software. To overcome this, the simulator now converts radial grids to Irregular Corner-Point Grids and adjusts the model's pore volume to reflect radial coordinates; thus, overcoming the display limitation.

See also the DRV, DTHETAV, DZ and TOPS keywords in the GRID section to fully define a radial or spider grid model.

Example

Given the dimensions of the grid in the RUNSPEC section to be 10,1, 8 for NX, NY and NZ respectively, we first define the inner radius of the radial model,

```
--
--       INNER RADIUS OF FIRST GRID BLOCK IN THE RADIAL DIRECTION
--
INRAD
         0.25
/
```

and then DR should be defined as:

```
--
--       DEFINE GRID BLOCK R DIRECTION CELL SIZE
--
DR
          1.75  2.32  5.01  10.84  23.39  50.55  109.21  235.92  509.68  1101.0
         1.75  2.32  5.01  10.84  23.39  50.55  109.21  235.92  509.68  1101.0
          1.75  2.32  5.01  10.84  23.39  50.55  109.21  235.92  509.68  1101.0
          1.75  2.32  5.01  10.84  23.39  50.55  109.21  235.92  509.68  1101.0
          1.75  2.32  5.01  10.84  23.39  50.55  109.21  235.92  509.68  1101.0
          1.75  2.32  5.01  10.84  23.39  50.55  109.21  235.92  509.68  1101.0
          1.75  2.32  5.01  10.84  23.39  50.55  109.21  235.92  509.68  1101.0
    /
```

The above example defines the size of the cells in the R direction based on 80 cells in the model as defined by the DIMENS keyword in the RUNSPEC section.

Note that since the first layer (K=1) must be defined and subsequent layers default to the layer above then:

```
--
--       INNER RADIUS OF FIRST GRID BLOCK IN THE RADIAL DIRECTION
--
INRAD
         0.25
/
--
--       DEFINE GRID BLOCK R DIRECTION CELL SIZE
--
DR
          1.75  2.32  5.01  10.84  23.39  50.55  109.21  235.92  509.68  1101.0
    /
```

is equivalent to previous example.

6.3.43 DRV - DEFINE THE SIZE OF GRID BLOCKS IN THE R DIRECTION VIA A VECTOR

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

DRV defines the size of grid blocks in the R direction via a vector as opposed to defining the property for each cell for a Radial or Spider Grid. The RADIAL or SPIDER keyword in the RUNSPEC section should be activated to indicate that radial or spider geometry is being used.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	DRV	DRV is a vector of real numbers describing the cell size for the grid blocks in the R direction in a radial grid. Repeat counts may be used, for example 10*100.0.			
		feet	m	cm	None
Notes: 1) The number of entries should correspond to the NX parameter on the DIMENS keyword in the RUNSPEC section 2) The keyword is terminated by a "/".					

Table 6.24: DRV Keyword Description

Note that the SPIDER keyword activates OPM Flow's radial grid geometry option for the model. This keyword will create a spiderweb-shaped grid based on a corner-point grid using the standard radial grid keywords: INRAD, DRV, DTHETAV, DZ/DZV etc. in the GRID the section. A spider grid can be viewed in 3D in OPM ResInsight unlike radial grids that cannot be viewed in the software. To overcome this, the simulator now converts radial grids to Irregular Corner-Point Grids and adjusts the model's pore volume to reflect radial coordinates; thus, overcoming the display limitation.

See also the DR, DTHETAV, DZ and TOPS keywords in the GRID section to fully define a radial or spider grid model.

Example

```
--
--        INNER RADIUS OF FIRST GRID BLOCK IN THE RADIAL DIRECTION
--
INRAD
          0.25
/
--
--        DEFINE GRID BLOCK SIZES IN THE R DIRECTION
--
DRV
          1.75  2.32  5.01  10.84  23.39  50.55  109.21  235.92  509.68  1101.08 /
```

The above example defines the size of the cells in the R direction based on NX equals 10 on the DIMENS keyword in the RUNSPEC section. Note the INRAD keyword to define the inner radius of the radial grid.

6.3.44 DTHETA - DEFINE THE SIZE OF GRID BLOCKS IN THE THETA DIRECTION FOR ALL CELLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

DTHETA defines the size of all grid blocks in the Theta direction via an array for each cell in model. The RADIAL or SPIDER keyword in the RUNSPEC section should be activated to indicate that radial or spider geometry is being used.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	DTHETA	DTHETA is an array of real numbers describing the cell size in the THETA direction in radial grids for each cell in the model. Repeat counts may be used, for example 10*25.0			
		degrees	degrees	degrees	None

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.

2) The keyword is terminated by a "/".

Table 6.25: DTHETA Keyword Description

Note that the SPIDER keyword activates OPM Flow's radial grid geometry option for the model. This keyword will create a spiderweb-shaped grid based on a corner-point grid using the standard radial grid keywords: INRAD, DRV, DTHETAV, DZ/DZV etc. in the GRID the section. A spider grid can be viewed in 3D in OPM ResInsight unlike radial grids that cannot be viewed in the software. To overcome this, the simulator now converts radial grids to Irregular Corner-Point Grids and adjusts the model's pore volume to reflect radial coordinates; thus, overcoming the display limitation. See also the DRV, DTHETAV, DZ and TOPS keywords in the GRID section to fully define a radial or spider grid model.

Example

Given the dimensions of the grid in the RUNSPEC section to be 10, 6, 1 for NX, NY and NZ respectively, then DTHETA should be defined as:

```
--
--       DEFINE GRID BLOCK SIZES IN THE THETA DIRECTION
--
DTHETA
         10*60.0
         10*60.0
         10*60.0
         10*60.0
         10*60.0
         10*60.0
/
```

The above example defines the size of the cells in the R direction based on 60 cells in the model as defined by the DIMENS keyword in the RUNSPEC section.

6.3.45 DTHETAV - SETS THE SIZE OF GRID BLOCKS IN THETA DIRECTION VIA A VECTOR

Description

DTHETAV defines the size of grid blocks in the THETA direction via a vector as opposed to defining the property for each cell for a Radial Grid. The RADIAL or SPIDER keyword in the RUNSPEC should be activated to indicate that radial geometry is being used.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	DTHETAV	DTHETAV is a vector of real numbers describing the cell size for the grid blocks in the THETA direction in a radial grid. Repeat counts may be used, for example 10*100.0.			
		degrees	degrees	degrees	None
Notes: 1) The number of entries should correspond to the NY parameter of the DIMENS keyword in the RUNSPEC section. 2) The keyword is terminated by a "/".					

Table 6.26: DTHETA Keyword Description

Note that the SPIDER keyword activates OPM Flow's radial grid geometry option for the model. This keyword will create a spiderweb-shaped grid based on a corner-point grid using the standard radial grid keywords: INRAD, DRV, DTHETAV, DZ/DZV etc. in the GRID the section. A spider grid can be viewed in 3D in OPM ResInsight unlike radial grids that cannot be viewed in the software. To overcome this, the simulator now converts radial grids to Irregular Corner-Point Grids and adjusts the model's pore volume to reflect radial coordinates; thus, overcoming the display limitation.

See also the DRV, DZV and TOPS keywords to fully define a radial or spider grid model.

Example

```
--
--       DEFINE GRID BLOCK SIZES IN THE THETA DIRECTION (BASED ON NY = 6)
--
DTHETAV
         60.0  60.0  60.0  60.0  60.0  60.0                                  /
```

The above example defines the size of the cells in the THETA direction based on NY equals six in the DIMENS keyword in the RUNSPEC section.

6.3.46 DUMPFLUX – ACTIVATE WRITING OUT OF A FLUX FILE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword activates the writing out of a full field (the full grid) FLUX file for later processing in a Flux Boundary run. The Flux Boundary feature allows for the segmentation of the full grid into flux boundary areas which allow for a sub-area of the grid to be run and at the same time model the flux across the boundary derived from the main grid. The object of this feature is to be able to investigate the performance of various areas of the model without having to run the full field, thus improving computational efficiency and run times, but still obtain "reasonable" results due to the incorporation of the fluxes across the boundary.

This feature is not available in OPM Flow; however it is documented here for completeness.

There is no data required for this keyword and there is no terminating "/" for this keyword.

Example

```
--
--       ACTIVATE WRITING OUT OF A FLUX FILE
--
DUMPFLUX                                                                    /
```

The above example switches on the writing of the FLUX output file; the keyword has no effect and is ignored by the simulator.

6.3.47 DX - DEFINE THE SIZE OF GRID BLOCKS IN THE X DIRECTION FOR ALL CELLS

Description

DX defines the size of all grid blocks in the X direction via an array for each cell in a Cartesian Regular Grid model.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	DX	DX is an array of real numbers describing the cell size in the X direction for each cell in the model. Repeat counts may be used, for example 10*100.0.			
		feet	m	cm	None

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.

2) The keyword is terminated by a "/".

Table 6.27: DX Keyword Description

See also the DY, DZ and TOPS keywords to fully define a Cartesian Regular Grid.

Example

```
--
--       DEFINE GRID BLOCK X DIRECTION CELL SIZE (BASED ON NX x NY x NZ = 300)
--
DX
         300*1000                                                              /
```

The above example defines the size of the cells in the X direction based on 300 cells in the model as defined by the DIMENS keyword in the RUNSPEC section.

6.3.48 DXV - DEFINE THE SIZE OF GRID BLOCKS IN THE X DIRECTION VIA A VECTOR

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

DXV defines the size of grid blocks in the X direction via a vector as opposed to defining the X direction cell size for each cell for a Cartesian Regular Grid.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	DXV	DXV is a vector of real numbers describing the cell size for the grid blocks in the X direction. Repeat counts may be used, for example 10*100.0.			
		feet	m	cm	None
Notes: 1) The number of entries should correspond to the NX parameter on the DIMENS keyword in the RUNSPEC section. 2) The keyword is terminated by a "/".					

Table 6.28: DXV Keyword Description

See also the DYV, DZV and TOPS keywords to fully define a Cartesian Regular Grid.

Example

```
--
--       DEFINE GRID BLOCK X DIRECTION CELL SIZE (BASED ON NX = 5)
--
DXV
         5*100                                                    /
```

The above example defines the size of the cells in the X direction based on NX equals 5 on the DIMENS keyword in the RUNSPEC section.

6.3.49 DY - DEFINE THE SIZE OF GRID BLOCKS IN THE Y DIRECTION FOR ALL CELLS

Description

DY defines the size of all grid blocks in the Y direction via an array for each cell in a Cartesian Regular Grid model.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	DY	DY is an array of real numbers describing the cell size in the Y direction for each cell in the model. Repeat counts may be used, for example 10*100.0.			
		feet	m	cm	None

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.

2) The keyword is terminated by a "/".

Table 6.29: DY Keyword Description

See also the DX, DZ and TOPS keywords to fully define a Cartesian Regular Grid.

Example

```
--
--        DEFINE GRID BLOCK Y DIRECTION CELL SIZE (BASED ON NX x NY x NZ = 300)
--
DY
          300*1000                                                              /
```

The above example defines the size of the cells in the Y direction based on 300 cells in the model as defined by the DIMENS keyword in the RUNSPEC section.

6.3.50 DYV - DEFINE THE SIZE OF GRID BLOCKS IN THE Y DIRECTION VIA A VECTOR

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

DYV defines the size of grid blocks in the Y direction via a vector as opposed to defining the Y direction cell size for each cell for a Cartesian Regular Grid.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	DYV	DYV is a vector of real numbers describing the cell size for the grid blocks in the Y direction. Repeat counts may be used, for example 10*100.0.			
		feet	m	cm	None
Notes: 1) The number of entries should correspond to the NY parameter on the DIMENS keyword in the RUNSPEC section. 2) The keyword is terminated by a "/".					

Table 6.30: DYV Keyword Description

See also the DXV, DZV and TOPS keywords to fully define a Cartesian Regular Grid.

Example

```
--
--        DEFINE GRID BLOCK Y DIRECTION CELL SIZE (BASED ON NY = 5)
--
DYV
          5*100                                                              /
```

The above example defines the size of the cells in the Y direction based on NY equals 5 on the DIMENS keyword in the RUNSPEC section.

6.3.51 DZ - DEFINE THE SIZE OF GRID BLOCKS IN THE Z DIRECTION FOR ALL CELLS

Description

DZ defines the size of all grid blocks in the Z direction via an array for each cell in a Cartesian Regular Grid model.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	DZ	DZ is an array of real numbers describing the cell size in the Z direction for each cell in the model. Repeat counts may be used, for example 10*100.0.			
		feet	m	cm	None
Notes: 1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement. 2) The keyword is terminated by a "/".					

Table 6.31: DZ Keyword Description

See also the DX, DY and TOPS keywords to fully define a Cartesian Regular Grid.

Example

```
--
--        DEFINE GRID BLOCK Z DIRECTION CELL SIZE (BASED ON NX x NY x NZ = 300)
--
DZ
          100*20.0   100*30.0   100*50.0                                        /
```

The above example defines the size of the cells in the Z direction based on 300 cells in the model as defined by the DIMENS keyword in the RUNSPEC section.

6.3.52 DZMATRIX - MATRIX BLOCK HEIGHT FOR GRAVITY DRAINAGE MODEL FOR ALL CELLS

Description

The DZMATRIX keyword defines the matrix block height for the gravity drainage model by grid block for when the Dual Permeability or Dual Porosity models are activated by the DUALPERM and DUALPORO keywords and the Gravity Drainage option is invoked via the GRAVDR and GRAVDRM keywords. All keywords are in the RUNSPEC section.

The keyword is identical to the DZMTRXV keyword in the GRID section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

6.3.53 DZMTRX - MATRIX BLOCK HEIGHT FOR GRAVITY DRAINAGE MODEL FOR THE GRID

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The DZMTRX keyword defines a constant matrix block height for the gravity drainage model for the entire grid for when the Dual Permeability or Dual Porosity models are activated by the DUALPERM and DUALPORO keywords and the Gravity Drainage option is invoked via the GRAVDR and GRAVDRM keywords. All keywords are in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

6.3.54 DZMTRXV - MATRIX BLOCK HEIGHT FOR GRAVITY DRAINAGE MODEL FOR ALL CELLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The DZMATRIX keyword defines the matrix block height for the gravity drainage model by grid block for when the Dual Permeability or Dual Porosity models are activated by the DUALPERM and DUALPORO keywords and the Gravity Drainage option is invoked via the GRAVDR and GRAVDRM keywords. All keywords are in the RUNSPEC section.

The keyword is identical to the DZMATRIX keyword in the GRID section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

6.3.55 DZNET – DEFINE GRID BLOCK NET THICKNESS FOR ALL CELLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

DZNET defines the net thickness of all grid blocks in the Z direction via an array for each cell in a Cartesian Regular Grid model.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	DZNET	DZNET is an array of real numbers describing the net thickness in the Z direction for each cell in the model. Repeat counts may be used, for example 10*100.0. If the value for a grid block is not defined then the grid block size (DZ) is assigned to the missing values.			
		feet	m	cm	DZ

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.

2) The keyword is terminated by a "/".

Table 6.32: DZNET Keyword Description

See also the DX, DY, DZ, NTG and TOPS keywords to fully define a Cartesian Regular Grid.

Example

```
--
--       DEFINE GRID BLOCK Z DIRECTION NET THICKNESS(BASED ON NX x NY x NZ = 300)
--
DZNET
         100*15.0   100*25.0   00*45.0                                          /
```

The above example defines the net thickness of the cells in the Z direction based on 300 cells in the model as defined by the DIMENS keyword in the RUNSPEC section.

6.3.56 DZV - DEFINE THE SIZE OF GRID BLOCKS IN THE Z DIRECTION VIA A VECTOR

Description

DZV defines the size of grid blocks in the Z direction via a vector as opposed to defining the thickness property for each cell. The keyword is used for both Cartesian Regular Grids and Radial Grids.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	DZV	DZV is a vector of real numbers describing the cell size for the grid blocks in the Z direction. Repeat counts may be used, for example 10*20.0.			
		feet	m	cm	None
Notes: 1) The number of entries should correspond to the NZ parameter on the DIMENS keyword in the RUNSPEC section.. 2) The keyword is terminated by a "/".					

Table 6.33: DZV Keyword Description

See also the DXV, DXY and TOPS keywords for a Cartesian Regular Grid and DRV, DTHETAV and TOPS keywords to fully define a Radial Grid model.

Example

```
--
--        DEFINE GRID BLOCK SIZES IN THE Z DIRECTION (BASED ON NZ = 20)
--
DZV
          3.0   5.0   3.0   2.0   5.0  15*3.0                                  /
```

The above example defines the size of the cells in the Z direction based on NZ equals 20 on the DIMENS keyword in the RUNSPEC section.

6.3.57 ECHO – ACTIVATE ECHOING OF USER INPUT FILES TO THE PRINT FILE

Turns on echoing of all the input files to the print file; note that this keyword is activated by default and can subsequently be switched off by the NOECHO activation keyword.

See *ECHO – Activate Echoing of User Input Files to the Print File* in the GLOBAL section for a full description.

6.3.58 END – DEFINE THE END OF THE INPUT FILE

This keyword marks the end of the input file and can occur in any section. Any keywords and data after the END keyword are ignored.

See *END – Define the End of the Input File* in the GLOBAL section for a full description.

6.3.59 ENDBOX – DEFINE THE END OF THE BOX DEFINED GRID

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword marks the end of a previously defined BOX sub-grid as defined by a previously entered BOX keyword. The keyword resets the input grid to be the full grid as defined by the NX, NY, and NZ variables on the DIMENS keyword in the RUNSPEC section.

There is no data required for this keyword.

Example

```
--
--       DEFINE INPUT BOX FOR EDITING INPUT ARRAYS
--
--       ---------- BOX ---------
--       I1  I2   J1  J2   K1  K2
BOX
         10  10   1   6    1   1                          / DEFINE BOX AREA
--
--
--       DEFINE GRID BLOCK PERMZ DATA FOR THE INPUT BOX
--
PERMZ
         6*0.01                                          /
--
--       DEFINE END OF INPUT BOX EDITING OF INPUT ARRAYS
--
ENDBOX
```

The above example defines a subset of the grid and sets the cells PERMZ values to 0.01 for that area. After which the ENDBOX keyword resets the input to be the full grid.

Note

It is good practice to always use the ENDBOX keyword to reset the input back to the full grid when all the modifications for a sub-grid have been completed.

6.3.60 ENDFIN – END THE DEFINITION OF A LOCAL GRID REFINEMENT

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The ENDFIN keyword defines the end of a Cartesian or radial local grid refinement ("LGR") definition and a LGR property definition data set. In the GRID section the CARFIN, RADFIN, and RADFIN4 keywords defines the start of an LGR description section, whereas the REFINE keyword in the EDIT, PROPS, REGIONS, SOLUTION and SCHEDULE section defines the start. The REFINE keyword can also be used in the GRID section provided the LGR has been previously specified by the CARFIN, RADFIN, or RADFIN4 keywords.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

There is no data required for this keyword and there is no terminating "/" for this keyword.

Example

The example below is based on using the CARFIN keyword in the GRID section to define an LGR in the global grid, named LGR-OP01 with a maximum of one well allowed in the LGR.

```
--
--        CARFIN LGR GRID COMMANDS
--
--        LGR         ----- FINE GRID ------    -- CARFIN GRID --   MAX      HOST
--        NAME        I1   I2   J1   J2   K1   K2     NX     NY     NZ    WELLS    NAME
CARFIN
          LGR-OP01    24   24   87   87    1   50      3      3     50      1      GLOBAL /


ENDFIN
```

Here the one global cell in the areal plane (24, 87) is divided into three LGR cells in the x-direction and three cells in the y-direction. Since no other property data is given, then the LGR cells take their properties from the host grid, that is the global grid.

6.3.61 ENDINC – DEFINE THE END OF AN INCLUDE FILE

This keyword marks the end of an include file specified on the INCLUDE keyword. When the ENDINC keyword is encountered in the INCLUDE file, input data is read from the next keyword in the current file. Any keywords and data after the ENDINC keyword in the INCLUDE file are ignored.

See *ENDINC – Define the End of an Include File* in the GLOBAL section for a full description.

6.3.62 ENDSKIP – DEACTIVATE SKIPPING OF KEYWORDS AND INPUT DATA

Turns off skipping of keywords that was activated by the SKIP, SKIP100, or SKIP300 keywords. Note that for each SKIP keyword activated there must be a corresponding ENDSKIP keyword.

See *ENDSKIP – DeActivate Skipping of Keywords and Input Data* in the GLOBAL section for a full description.

6.3.63 EQLZCORN - MODIFY THE DEPTH OF THE CORNER-POINT DEPTH ARRAY

Description

The EQLZCORN keyword modifies the depth of a corner point of a grid block on the pillars defining the reservoir grid. The keyword can be only used be used with Irregular Corner-Point Grids.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

6.3.64 EQUALREG – SETS AN ARRAY TO A CONSTANT BY REGION NUMBER

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The EQUALREG keyword sets a specified array to a constant for cells with a specific region number. The region number array can be FLUXNUM, MULTNUM or OPERNUM and these arrays must be defined and be available before the EQUALREG keyword is read by the simulator. The constant can be real or integer depending on the property array type; however, the property arrays that can be operated on is dependent on which section the EQUALREG keyword is being used.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description	Default
1	ARRAY	The name of the array to be modified. This is the keyword name identifying the property and is up to eight characters in length and enclosed in quotes.	None
2	CONSTANT	An integer or real value to be assigned to the ARRAY in the same units as the ARRAY property for a given REGION	0
3	REGION NUMBER	REGION NUMBER is a positive integer representing the region for which the CONSTANT in (2) should be applied	None
4	REGION ARRAY	The REGION ARRAY to use for applying the CONSTANT in (2) based on the REGION NUMBER in (3). REGION ARRAY can have the following values: 1) F for the FLUXNUM array 2) M for the MULTNUM array 3) O for the OPERNUM array	M

Notes:

1) Where the REGION NUMBER should be less than or equal to the maximum number of regions as defined on the REGDIMS keyword for the FIPNUM and OPERNUM arrays or the GRIDOPTS keyword for the MULTNUM array in the RUNSPEC section.
2) Each record must be terminated by a "/" and the keyword is terminated by a "/".

Table 6.34: EQUALREG Keyword Description

The applicable arrays for each section are defined in Table 6.35.

EQUALREG Keyword And Variable Options By Section						
GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
DX	DEPTH	SWL	ENDNUM	PRESSURE		
DY	PORV	SWCR	EQLNUM	SWAT		
DZ	TRANX	SWU	FIPNUM	SGAS		
PERMX	TRANY	SGL	IMBNUM	RV		
PERMY	TRANZ	SGCR	MISCNUM	RS		
PERMZ	DIFFX	SGU	PVTNUM	TBLK		
MULTX	DIFFY	KRW	ROCKNUM	GI		

EQUALREG Keyword And Variable Options By Section						
GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
MULTY	DIFFZ	KRO	SATNUM	OILAPI		
MULTZ	TRANR	KRG	WH2NUM	SALT		
DR	TRANTHT	PCG		GASCONC		
THETA	DIFFR	PCW		SOLVCONC		
PERMR	DIFFTHT			SOLVFRAC		
PERMTHT				SFOAM		
DZNET				SPOLY		
PORO						
NTG						
FLUXNUM						
MULTNUM						
MPANUM						
DIFFX						
DIFFY						
DIFFZ						
DIFFR						
DIFFTHT						

Table 6.35: EQUALREG Keyword Applicable Arrays by Section

Example

```
-- FIRST DEFINE MULTNUM ARRAYS FOR 10 X 10 X 20 MODEL
--
--       ARRAY     CONSTANT     ---------- BOX ---------
--                              I1  I2   J1  J2   K1  K2
EQUALS
        MULTNUM     1          1*  1*   1*  1*   1*  1*      / MULTNUM IN MODEL
        MULTNUM     2          1*  1*   1*  1*   6   6       / MULTNUM IN MODEL
        MULTNUM     3          1*  1*   1*  1*   10  10      / MULTNUM IN MODEL
/
--     NOW SET PORO AND PERMX BASED ON THE MULTNUM REGION NUMBER
--
--        SETS A CONSTANT TO AN ARRAY BASED ON A REGION NUMBER
--
--        ARRAY      CONSTANT  REGION   REGION ARRAY
--                   VALUE     NUMBER    M / F / O
EQUALREG
          PORO       0.200     1         M                  /
          PORO       0.150     2         M                  /
          PORO       0.120     3         M                  /
          PERMX    100.00      1         M                  /
          PERMX     75.00      2         M                  /
          PERMX     50.00      3         M                  /
/
```

The example first defines the MULTNUM array to 1 for all cells in the model, after which selected areas of model are assigned various MULTNUM integer values. The EQUALREG can then be invoked to set a constant values for the PORO and PERMX arrays for the various MULTNUM regions.

6.3.65 EQUALS – SETS A SPECIFIED ARRAY TO A CONSTANT

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The EQUALS keyword sets a specified array or part of an array to a constant. The constant can be real or integer depending on the array type; however, the arrays that can be operated on is dependent on which section the EQUALS keyword is being used.

No.	Name	Description	Default
1	ARRAY	The name of the array to be modified. This is the keyword name identifying the property and is up to eight characters in length and enclosed in quotes.	None
2	CONSTANT	An integer or real value that the ARRAY will be set to in the same units as the ARRAY property.	None
3	I1	A positive integer that defines the lower bound of the array in the I-direction to be modified must be greater than or equal to one and less than or equal to I2 and NX.	1
4	I2	A positive integer that defines the upper bound of the array in the I-direction to be modified must be greater than or equal to I1 and less than or equal to NX	NX
5	J1	A positive integer that defines the lower bound of the array in the J-direction to be modified must be greater than or equal to one and less than or equal to J2 and NY.	1
6	J2	A positive integer that defines the upper bound of the array in the J-direction to be modified must be greater than or equal to J1 and less than or equal to NY.	NY
7	K1	A positive integer that defines the lower bound of the array in the K-direction to be modified must be greater than or equal to one and less than or equal to K2 and NZ.	1
8	K2	A positive integer that defines the upper bound of the array in the K-direction to be modified must be greater than or equal to K1 and less than or equal to NZ.	NZ

Notes:

1) Where NX, NY and NZ are the dimensions of the model as defined on the DIMENS keyword in the RUNSPEC section.

2) Each record must be terminated by a "/" and the keyword is terminated by a "/".

Table 6.36: EQUALS Keyword Description

The applicable arrays for each section are defined in Table 6.37.

EQUALS Keyword And Variable Options By Section						
GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
DX	DEPTH	SWL	ENDNUM	PRESSURE		
DY	PORV	SWCR	EQLNUM	SWAT		
DZ	TRANX	SWU	FIPNUM	SGAS		

EQUALS Keyword And Variable Options By Section						
GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
PERMX	TRANY	SGL	IMBNUM	RV		
PERMY	TRANZ	SGCR	MISCNUM	RS		
PERMZ	DIFFX	SGU	PVTNUM	TBLK		
MULTX	DIFFY	KRW	ROCKNUM	GI		
MULTY	DIFFZ	KRO	SATNUM	OILAPI		
MULTZ	TRANR	KRG	WH2NUM	SALT		
DR	TRANTHT	PCG		GASCONC		
THETA	DIFFR	PCW		SOLVCONC		
PERMR	DIFFTHT			SOLVFRAC		
PERMTHT				SFOAM		
DZNET				SPOLY		
PORO						
NTG						
FLUXNUM						
MULTNUM						
MPANUM						
DIFFX						
DIFFY						
DIFFZ						
DIFFR						
DIFFTHT						

Table 6.37: EQUALS Keyword Applicable Arrays by Section

Note

Unlike the commercial simulator, using the EQUALS keyword to setup the structure of the grid using the DX, DY, DZ and TOPS keywords should be avoided as it may cause OPM Flow to prematurely fail during the initialization.

See the second example on the correct way to setup this type of grid.

Examples

The first example resets the PERMX, PERMY and PERMZ, arrays to 0.10, 0.10, and 0.01 for all cells in layer five, respectively.

```
--
--   ARRAY      CONSTANT       ---------- BOX ---------
-                              I1  I2   J1  J2   K1  K2
EQUALS
    PERMX      0.1000         1*  1*   1*  1*   5   5  / PERMX TO 0.10 IN LAYER 5
    PERMY      0.1000         1*  1*   1*  1*   5   5  / PERMY TO 0.10 IN LAYER 5
    PERMZ      0.0100         1*  1*   1*  1*   5   5  / PERMZ TO 0.01 IN LAYER 5
/
```

The second example illustrates how to correctly setup a Cartesian Regular Grid in OPM Flow, given the DIMENS keyword in the RUNSPEC section is set to:

```
--
--       MAX     MAX     MAX
--       NDIVIX  NDIVIY  NDIVIZ
DIMENS
         10       10      3                                                  /
```

and the following keywords in the GRID section:

```
--
--       ACTIVATE IRREGULAR CORNER-POINT GRID TRANSMISSIBILITIES
--
OLDTRAN
--
--       DEFINE GRID BLOCK X DIRECTION CELL SIZE (BASED ON NX = 5)
--
DX
         300*1000                                                           /
--
--       DEFINE GRID BLOCK Y DIRECTION CELL SIZE (BASED ON NY = 5)
--
DY
         300*1000                                                           /
--
--       DEFINE GRID BLOCK SIZES IN THE Z DIRECTION
--
DZ
         100*20  100*30  100*50                                             /
--
--       DEFINE GRID BLOCK TOPS FOR THE TOP LAYER
--
TOPS
         100*8325                                                           /
--
--       ARRAY    CONSTANT        ---------- BOX ---------
--                                I1  I2   J1  J2   K1  K2
EQUALS
         PERMX     500.000        1*  1*   1*  1*    1   1 / Layer #01 Properties
         PERMY     500.000                                  /
         PERMZ      20.000                                  /
         PORO        0.300                                  /
         NTG         1.000                                  /

         PERMX      50.000        1*  1*   1*  1*    2   2 / Layer #02 Properties
         PERMY      50.000                                  /
         PERMZ      50.000                                  /
         PORO        0.300                                  /
         NTG         1.000                                  /

         PERMX     200.000        1*  1*   1*  1*    3   3 / Layer #03 Properties
         PERMY     200.000                                  /
         PERMZ     200.000                                  /
         PORO        0.300                                  /
         NTG         1.000                                  /
/
```

Notice that the DX, DY, DZ and TOPS keywords are defined separately, that is they are not included in the EQUALS keyword.

6.3.66 EXTFIN - DEFINE AN EXTERNAL UNSTRUCTURED LOCAL GRID REFINEMENT

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The EXTFIN keyword defines an external Unstructured Local Grid Refinement ("LGR") in a cell or a group of cells in the global grid, and for when LGRs have been activated for the model using the LGR keyword in the RUNSPEC section. Note the global grid can be either structured, see the EXTREPL keyword in the GRID section for global structure grids, or unstructured, see the EXTHOST keyword in the GRID section for unstructured global grids.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

6.3.67 EXTHOST - DEFINE HOST CELLS FOR EXTERNAL LGRS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The EXTHOST keyword defines the host global grid blocks for an external Local Grid Refinement ("LGR") for when LGRs have been activated for the model using the LGR keyword in the RUNSPEC section, and the global grid is an unstructured grid.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

6.3.68 EXTRAPMS – ACTIVATE EXTRAPOLATION WARNING MESSAGES

The EXTRAPMS keyword activates extrapolation warning messages for when OPM Flow extrapolates the PVT or VFP tables. Frequent extrapolation warning messages should be investigated and resolved as this would indicate possible incorrect data and may result in the simulator extrapolating to unrealistic values.

See *EXTRAPMS – Activate Extrapolation Warning Messages* in the GLOBAL section for a full description.

6.3.69 EXTREPGL - DEFINE HOST CELLS FOR EXTERNAL UNSTRUCTURED LGRS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The EXTREPGL keyword defines the host global grid blocks for an external Unstructured Local Grid Refinement ("LGR") for when LGRs have been activated for the model using the LGR keyword in the RUNSPEC section, and the global grid is a structured grid.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

6.3.70 FAULTS – DEFINE FAULTS IN THE GRID GEOMETRY

Description

The FAULTS keyword defines the faults in the grid geometry and the keyword is normally exported with the grid geometry COORD and ZCORN data sets from static earth modeling software. Note that the FAULTS keyword is not required to describe the structural geometry as this is already accounted for in the COORD and ZCORN data sets, but instead lists the fault traces with respect to the grid. Once the fault traces have been defined with the FAULTS keyword then the fault transmissibilities can be modified by the MULTFLT keyword. Note that without the FAULTS keyword one would still get proper cross-fault transmissibilities but they would not be modifiable using MULTFLT keyword.

No.	Name	Description	Default
1	FLTNAME	FLTNAME is a character string enclosed in quotes with a maximum length of eight characters, that defines the name of the fault.	None
2	I1	The lower bound of the fault's I-direction range must be greater than or equal to one and less than or equal to I2 and NX.	None
3	I2	The upper bound of the fault's I-direction range must be greater than or equal to I1 and less than or equal to NX	None
4	J1	The lower bound of the fault's J-direction range must be greater than or equal to one and less than or equal to J2 and NY.	None
5	J2	The upper bound of the fault's J-direction range must be greater than or equal to J1 and less than or equal to NY.	None
6	K1	The lower bound of the fault's K-direction range must be greater than or equal to one and less than or equal to K2 and NZ.	None
7	K2	The upper bound of the fault's K-direction range must be greater than or equal to K1 and less than or equal to NZ.	None
8	FLTFACE	FLTFACE is a character string enclosed in quotes with a maximum length of two characters, that classifies the fault face. 1) If TRANMULT on the GRIDOPTS keyword in the RUNSPEC section is set to NO then FLTFACE can have values of X, Y, or Z. 2) Alternatively, if TRANMULT on the GRIDOPTS keyword in the RUNSPEC section is set to YES then FLTFACE can have values of X, Y, or Z for the positive direction, or X-, Y- or Z- for the negative direction transmissibilities.	None

Notes:

1) Where NX, NY and NZ are the dimensions of the model as defined on the DIMENS keyword in the RUNSPEC section.
2) The FAULTDIM keyword in the RUNSPEC defines the maximum number of records (or segments) that can be entered with the FAULTS keyword.
3) Each record must be terminated by a "/" and the keyword is terminated by a "/".

Table 6.38: FAULTS Keyword Description

Example

The example below defines two fault traces, the first being the 'M_WEST' fault and the second the 'BC' fault trace.

```
--
--        DEFINE FAULTS IN THE GRID GEOMETRY
--
--        FAULT        ------------ FAULT TRACE -------------
--        NAME          I1   I2    J1   J2    K1   K2    FACE
FAULTS
          'M_WEST'       5    5     3    3     1    22    'X' /
          'M_WEST'       5    5     4    4     1    22    'X' /
          'M_WEST'       5    5     5    5     1    22    'X' /
          'M_WEST'       5    5     6    6     1    22    'X' /
          'M_WEST'       5    5     7    7     1    22    'X' /
          'M_WEST'       5    5     8    8     1    22    'X' /
          'M_WEST'       5    5     9    9     1    22    'X' /
          'M_WEST'       5    5    10   10     1    22    'X' /
          'M_WEST'       5    5    11   11     1    22    'X' /
……….……..

          'BC'          43   43     8    8     1    22   'Y'  /
          'BC'          42   42     9    9     1    22   'X'  /
          'BC'          44   44     8    8     1    22   'Y'  /
          'BC'          45   45     8    8     1    22   'Y'  /
          'BC'          46   46     8    8     1    22   'Y'  /
          'BC'          31   31     9    9     1    22   'Y'  /
          'BC'          30   30    10   10     1    22   'X'  /
          'BC'          32   32     9    9     1    22   'Y'  /
          'BC'          33   33     9    9     1    22   'Y'  /
          'BC'          34   34     9    9     1    22   'Y'  /
          'BC'          35   35     9    9     1    22   'Y'  /
          'BC'          36   36     9    9     1    22   'Y'  /
          'BC'          37   37     9    9     1    22   'Y'  /
          'BC'          38   38     9    9     1    22   'Y'  /
          'BC'          39   39     9    9     1    22   'Y'  /
          'BC'          40   40     9    9     1    22   'Y'  /
……….……..
/
```

6.3.71 FILEUNIT – ACTIVATE UNIT CONSISTENCY CHECKING

Description

The FILEUNIT keyword defines the units of the data set, and is used to verify that the units in the input deck and any associated include files are consistent. The keyword does not provide for the conversion between different sets of units.

No.	Name	Description	Default
1	FILEUNIT	A character string that defines the units of the data set, and should be set to: 1) FIELD for field units, 2) METRIC for metric units, or 3) LAB for laboratory units	None
Notes: 1) No unit conversion is performed. 2) The keyword is terminated by a "/".			

Table 6.39: FILEUNIT Keyword Description

OPM Flow's behavior is controllable through the "UNIT_SYSTEM_MISMATCH" environment variable. The default behavior if the check fails (i.e., if one of the INCLUDE files has a unit system different from the main run specification) is to terminate the simulation with an error.

Example

```
--
--       ACTIVATE UNIT CONSISTENCY CHECKING
--
FILEUNIT
         FIELD                                                          /
```

The above example defines the data set units to be FIELD units.

6.3.72 FLUXNUM – DEFINE THE FLUX REGIONS

Description

The FLUXNUM keyword defines the flux region numbers for each grid block, as such there must be one entry for each cell in the model. The array is used with the Flux Boundary option to define the various flux regions; however, the Flux Boundary option has not been implemented in OPM Flow. In addition, the array can be used with the EQUALREG, ADDREG, COPYREG, MULTIREG, MULTREGP and MULTREGT keywords in calculating various grid properties in the GRID section. This facility has been implemented in OPM Flow.

No.	Name	Description	Default
1	FLUXNUM	FLUXNUM defines an array of positive integers assigning a grid cell to a particular flux region. The maximum number of flux regions is set by the MXNFLX variable on the REGDIMS keyword in the RUNSPEC section.	1

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.

2) The keyword is terminated by a "/".

Table 6.40: FLUXNUM Keyword Description

Examples

The example below sets three FLUXNUM regions for a 4 x 5 x 2 model.

```
--
--        DEFINE FLUXNUM REGIONS FOR ALL CELLS
--
FLUXNUM
          2  2  1  1  2  2  1  1  1  1  1  1  1  1  1  1  1  1  1  1
          3  3  1  1  3  3  1  1  1  1  1  1  1  1  1  1  1  1  1  1
/
```

Alternatively the EQUALS keyword could be employed to accomplish the same task, that is:

```
--
--        ARRAY     CONSTANT       ---------- BOX ---------
--                                 I1  I2   J1  J2   K1  K2
EQUALS
          FLUXNUM    1            1*  1*   1*  1*   1*  1* / SET REGION 1
          FLUXNUM    2            1   2    1   2    1   1  / SET REGION 2
          FLUXNUM    3            1   2    1   2    2   2  / SET REGION 3
/
```

6.3.73 FLUXREG – DEFINE ACTIVE FLUX REGIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The FLUXREG is used in conjunction with the USEFLUX keyword in runs with have multiple flux regions, to reduce the number of flux regions, that is the keyword specifies which flux regions are active and which are not in the current run.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

6.3.74 FLUXTYPE – DEFINES THE FLUX BOUNDARY TYPE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The FLUXTYPE keyword defines the type of flux boundary to be used in the run.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

6.3.75 FORMFEED – DEFINED THE PRINT FILE FORM-FEED CHARACTER

The FORMFEED keyword defines the form-feed character, or carriage control character, for the output print (*.PRT) run summary (*.RSM) files. The keyword should be place at the very top of the input file.

See *FORMFEED – Defined the Print File Form-Feed Character* in the GLOBAL section for a dull description.

6.3.76 GDFILE – LOAD A GRID FILE

Description

The GDFILE keyword loads a GRID file that contains the structural data for the grid as a set of topological cuboidal cells, and EGRID files that contain structural and property data. Note OPM Flow only supports reading in EGRID files at this time.

No.	Name	Description	Default
1	GRIDFILE	A character string enclosed in quotes that defines the GRID or EGRID file to be read in and be processed by OPM Flow. Again, OPM Flow only supports reading in EGRID files.	None
2	FMTOPT	A defined character string that defines the format of the GRID or EGRID file to be read and should be set to one of the following: 1) FORMATTED: If the file is formatted as ASCII i.e. a text file, as oppose to a binary file. The option can be abbreviated to just the letter F. 2) UNFORMATTED: If the file is in binary format, note this option can be abbreviated to just the letter U. If the variable FMTOPT is omitted then the default is for binary file input for the commercial simulator; whereas, OPM Flow derives FMTOPT from the file extension (*.EGRID or *.FEGRID), making FMTOPT superfluous.	U
Notes: 1) The keyword is terminated by a "/".			

Table 6.41: GDFILE Keyword Description

See also the GRIDFILE keyword in the GRID section for exporting the GRID and EGRID files from the current simulation run.

Examples

The first example shown below loads the NOR-OPM-A00-GRID.EGRID file in binary format from the same directory as the data file.

```
--
--        LOAD A GRID FILE
--
GDFILE
          'NOR-OPM-A00-GRID.EGRID'       /
```

The next example loads the same EGRID file one directory above from where the data file is located.

```
--
--        LOAD a GRID FILE
--
GDFILE
          '../NOR-OPM-A00-GRID.EGRID'   /
```

6.3.77 GDORIENT - DEFINE GRID ORIENTATION PARAMETERS

Description

This keyword defines the grid orientation parameters for post-processing applications.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

6.3.78 GETDATA – LOAD AND ASSIGN DATA ARRAY FROM INIT OR RESTART FILES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The GETDATA keyword loads a data array from a previously generated INIT or RESTART file and assigns the loaded array to either same array in the run or another property array.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

6.3.79 GRID - DEFINE THE START OF THE GRID SECTION OF KEYWORDS

Description

The GRID activation keyword marks the end of the RUNSPEC section and the start of the GRID section that defines the key grid property data for the simulator including the grid structure, porosity, permeability and other relevant grid property data.

There is no data required for this keyword.

Example

```
-- ==========================================================================
--
-- GRID SECTION
--
-- ==========================================================================
GRID
```

The above example marks the end of the RUNSPEC section and the start of the GRID section in the OPM Flow data input file.

6.3.80 GRIDFILE – SET THE GRID FILE OUTPUT OPTIONS

Description

This keyword controls the output of a standard GRID or extended GRID file, as well as the extensible EGRID file for post-processing applications. The extended and extensible GRID formats are comparable; however, the extensible GRID format is more compact and is the only format supported by OPM Flow.

No.	Name	Description	Default
1	NGRID	A positive integer that controls the output of the GRID geometry file: 0) - for no GRID file to be written out. 1) - for the standard GRID file to be written out. 2) - for the extended GRID file to be written out. Only the default value of zero is supported.	0
2	NEGRID	A positive integer that controls the output of the EGRID geometry file: 0) - for no extensible GRID file to be written out. 1) - for the extensible GRID file to be written out. Only the default value of one is supported.	1
Notes: 1) The keyword is terminated by a "/".			

Table 6.42: GRIDFILE Keyword Description

Example

```
--
--       GRID FILE OUTPUT OPTIONS
--       GRID    EGRID
--       OPTN    OPTN
GRIDFILE
         0       1                                                    /
```

The above example defines that no GRID file will be written out and that the extensible GRID (that is the EGRID geometry format) file will be produced. This is the only configuration that OPM Flow supports

6.3.81 GRIDUNIT – DEFINE THE GRID UNITS

Description

The GRIDUNIT keyword defines the units of the grid data. It is usually output by pre-processing software when exporting the grid geometry. The data is not used by OPM Flow intrinsically, but is merely written to the output EGRID file, as specified by the GRIDFILE keyword, for the use of post-processing software like OPM ResInsight.

No.	Name	Description	Default
1	GRIDUNIT	A character string that defines the units of the coordinates stated on the MAPAXES keyword, and should be set to: 1) FEET for field units, 2) METRES for metric units, or 3) CM for laboratory units.	METRES
2	MAPOPT	A character string that defines if the grid data are measured relative to the map, or relative to the origin as stated on the MAPAXES keyword. MAPOPT should either be left blank (the default) indicating the origin is relative to the origin on the MAPAXES keyword, or set equal to MAP measured relative to the map.	1*
Notes: 1) Note the alternative spelling METRES, that is METERS is not recognized. 2) The keyword is terminated by a "/".			

Table 6.43: GRIDUNIT Keyword Description

Example

```
--
--       SET THE GRID UNITS FOR THE GRID
--
GRIDUNIT
         METRES                                                    /
```

The above example defines that the GRID units to be metric.

6.3.82 HALFTRAN – ACTIVATE HALF BLOCK TRANSMISSIBILITY CALCULATIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The HALFTRAN keyword activates the half block transmissibility calculation option.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

6.3.83 HEATCR – DEFINE RESERVOIR ROCK HEAT CAPACITY FOR ALL CELLS

Description

The HEATCR keyword defines the reservoir rock volumetric heat capacity for all cells for when OPM Flow's thermal calculation is activated by the THERMAL keywords in the RUNSPEC section.

This keyword can only be used if OPM Flow's thermal option has been activated by the THERMAL keyword in the RUNSPEC section. Note this is different to the commercial simulator that uses the TEMP keyword in the RUNSPEC section to activate the black-oil thermal model.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	HEATCR	HEATCR is an array of real positive numbers that define reservoir rock volumetric heat capacity of a grid block. Repeat counts may be used, for example 3000*25.0			
		Btu/ft3/°R	kJ/m3/K	J/cm3/K	None

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) The keyword is terminated by a "/".

Table 6.44: HEATCR Keyword Description

Note this keyword is incompatible with SPECROCK keyword in the PROPS section.

Example

```
--
--       DEFINE GRID BLOCK RESERVOIR ROCK HEAT CAPACITY
--       FOR ALL CELLS (BASED ON NX x NY x NZ = 300)
--       KEYWORD IS INCOMPATIBLE WITH THE SPECROCK KEYWORD
--       (OPM FLOW THERMAL OPTION ONLY)
--
HEATCR
         300*32.0                                                    /
```

The above example defines the reservoir rock volumetric heat capacity of 32.0 for each cell in the 300 grid block model.

6.3.84 HEATCRT – DEFINE RESERVOIR ROCK HEAT CAPACITY TEMPERATURE DEPENDENCE FOR ALL CELLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The HEATCRT keyword defines the reservoir rock volumetric heat capacity temperature dependence for all cells for when OPM Flow's thermal calculation is activated by the THERMAL keywords in the RUNSPEC section.

This keyword can only be used if OPM Flow's thermal option has been activated by the THERMAL keyword in the RUNSPEC section. Note this is different to the commercial simulator that uses the TEMP keyword in the RUNSPEC section to activate the black-oil thermal model.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	HEATCRT	HEATCRT is an array of real positive numbers that define reservoir rock volumetric heat capacity temperature dependence of a grid block. Repeat counts may be used, for example 3000*0.05			
		Btu/ft3/°R2	kJ/m3/K2	J/cm3/K2	None

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.

2) The keyword is terminated by a "/".

Table 6.45: HEATCRT Keyword Description

Note this keyword is incompatible with SPECROCK keyword in the PROPS section.

The data for this keyword and the HEATCR keyword are used to calculate the reservoir rock volumetric heat capacity temperature dependence using the following relationship:

$$\text{Heat Capacity of Rock} = \text{HEATCR}\left(Temp - Temp_{ref}\right) + \frac{\text{HEATCRT}\left(Temp - Temp_{ref}\right)^2}{2} \quad (6.3)$$

Example

```
--
--       DEFINE RESERVOIR ROCK HEAT CAPACITY TEMPERATURE DEPENDENCE
--       FOR ALL CELLS (BASED ON NX x NY x NZ = 300)
--       KEYWORD IS INCOMPATIBLE WITH THE SPECROCK KEYWORD
--       (OPM FLOW THERMAL OPTION ONLY)
--
HEATCRT
         300*0.05                                                    /
```

The above example defines the reservoir rock volumetric heat capacity temperature dependence of 0.05 for each cell in the 300 grid block model.

6.3.85 HMAQUNUM - HISTORY MATCH NUMERICAL AQUIFER GRADIENT PARAMETERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The HMAQUNUM keyword defines the history match numerical aquifer gradient parameters for when the History Match Gradient option has been activated by the HMDIMS keyword in the RUNSPEC section, and numerical aquifers have been specified in the model via the AQUNUM keyword and connected to the grid using AQUCON keyword. All keywords are in the GRID section.

See also the HMDIMS keyword in the RUNSPEC section that specifies the dimensions for the gradient option, including the maximum number of aquifers that can be used with the History Match Gradient option.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

6.3.86 HMFAULTS – HISTORY MATCH FAULT GRADIENT PARAMETERS

Description

The HMFAULTS keyword defines the history match faults gradient parameters for when the History Match Gradient option has been activated by the HMDIMS keyword in the RUNSPEC section, and for when the FAULTS keyword in the GRID section has been used to define faults in the model.

See also the HMDIMS keyword in the RUNSPEC section that specifies the dimensions for the gradient option, including the maximum number of faults that can be used with the History Match Gradient option.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

6.3.87 HMMLAQUN – HISTORY MATCH NUMERICAL AQUIFER GRADIENT MULTIPLIERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The HMMLAQUN keyword defines the history match numerical aquifer gradient multipliers for when the History Match Gradient option has been activated by the HMDIMS keyword in the RUNSPEC section, and numerical aquifers have been specified in the model via the AQUNUM keyword and connected to the grid using the AQUCON keyword. All keywords are in the GRID section.

Multipliers can be declared for numerical aquifers' pore volume, permeability, and aquifer to grid connection factors.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

6.3.88 HMMLT – HISTORY MATCH GRID PERMEABILITY GRADIENT CUMULATIVE MULTIPLIERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The HMMLT series of keywords defines the history match gradient cumulative permeability multipliers, for when the History Match Gradient option has been activated by the HMDIMS keyword in the RUNSPEC section. The keyword consists of the first five characters of "HMMLT" followed by a two or three character string shown in Table 6.46, that determines the permeability direction, for example, HMMLTPX.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

Mnemonic	Cartesian Grid		Radial Grid	
	Grid Keyword	HMMULT Keyword	Grid Keyword	HMMULT Keyword
PX/PR	PERMX	HMMLTPX	PERMR	HMMLTPR
PXY	PERMXY	HMMLTPXY		
PY/THT	PERMY	HMMLTPY	PERMTHT	HMMLTTH
PZ	PERMZ	HMMLTPZ	PERMZ	HMMLTPZ

Table 6.46: HMMLT Keyword List

See also the HMMULT keyword in the EDIT section

6.3.89 HMMMREGT - HISTORY MATCH REGION TRANSMISSIBILITY GRADIENT CUMULATIVE MULTIPLIERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The HMMMREGT keyword multiplies the transmissibility between two regions by a constant, for when the History Match Gradient option has been activated by the HMDIMS keyword in the RUNSPEC section. The constant should be a real number. Unlike the MULTREGT keyword in the GRID section, the HMMMREGT keyword modifications are cumulative.

Note that the HMMMREGT keyword only declares the two regions and the multiplier between those regions, the transmissibility direction (DIR on the MULTREGT keyword), type of transmissibility multiplier (TYPE on the MULTREGT keyword), and the region number array to use (ARRAY on the MULTREGT keyword), are all taken from the MULTREGY keyword. For example, the region number array can be FLUXNUM, MULTNUM or OPERNUM and these arrays must be defined and be available before the MULTREGT keyword is read by the simulator, and before the HMMMREGT keyword is used.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

6.3.90 HMMULRGT – HISTORY MATCH REGION TRANSMISSIBILITY PARAMETERS

Description

HMMULRGT defines the transmissibility between two regions gradient parameters, for when the History Match Gradient option has been activated by the HMDIMS keyword in the RUNSPEC section.

See also the HMDIMS keyword in the RUNSPEC section that specifies the dimensions for the gradient option, including the maximum number of gradient regions that can be used with the History Match Gradient option.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

6.3.91 HMMULTFT – HISTORY MATCH FAULT TRANSMISSIBILITY GRADIENT CUMULATIVE MULTIPLIERS

Description

HMMULTFT defines the history match fault transmissibility gradient cumulative multipliers to be applied to the fault transmissibilities for faults declared by the FAULT keyword in the GRID section, for when the History Match Gradient option has been activated by the HMDIMS keyword in the RUNSPEC section. The keyword allows for the re-scaling of the existing fault transmissibilities calculated by OPM Flow, or if the MULTFLT keyword has been entered, then HMMULTFT is applied to the existing MULTFLT multipliers.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

6.3.92 HMMULTSG – HISTORY MATCH DUAL POROSITY SIGMA GRADIENT CUMULATIVE MULTIPLIERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

HMMULTSG defines the history match dual porosity sigma parameter gradient cumulative multipliers applied to the dual porosity sigma value declared by the SIGMAV and SIGMAGDV keywords in the PROPS section, for when the History Match Gradient option has been activated by the HMDIMS keyword in the RUNSPEC section. In addition to the HMDIMS keyword, either the DUALPERM keyword that activates the Dual Permeability option, or the DUALPORO keyword that activates the Dual Porosity option for the run, must be declared in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

6.3.93 HRFIN - DEFINE THE RATIO OF LGR GRID BLOCKS IN THE R-DIRECTION

Description

HRFIN[120] defines the ratio of grid blocks for the DRV keyword in the r-direction via a vector within a Local Grid Refinement ("LGR") as opposed to defining the size for each cell for a Radial LGR Grid. The LGR keyword in the RUNSPEC section should be activated to indicate an LGR is being used, and the keyword HRFIN should be placed in between the RADIN (or RAFDIN4) and ENDFIN keywords in the GRID section. The DRV keyword in the GRID section defines the radial grid size in terms of the length, that is feet for field units, this keyword defines the length as the ratio of the previous cell size, staring with the inner radius (INRAD).

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	HRFIN	HRFIN is a vector of real numbers describing the ratio of cell size for the grid blocks in the r-direction in a radial LGR for the DRV keyword. Repeat counts may be used, for example 2*1.5.			
		dimensionless	dimensionless	dimensionless	None

Notes:

1) The number of entries should correspond to the NR parameter minus one on the RADFIN or RADFIN4 keywords in the GRID section

2) The keyword is terminated by a "/".

Table 6.47: HRFIN Keyword Description

See also the DR, DRV, DTHETAV, and DZ keywords in the GRID section to fully define a radial LGR model.

Example

```
--
--       INNER RADIUS OF FIRST GRID BLOCK IN THE RADIAL DIRECTION
--
INRAD    0.25                                                          /
--
--       DEFINE GRID BLOCK DRV RATIOS IN THE R DIRECTION
--
HRFIN
         1.50  2.00  3.00  5.00  7.00  10.00                           /
```

The above example defines the size of the cells in the R direction based on NR equals 7, resulting in NR-1 values on the RADFIN keyword in the GRID section. Note the INRAD keyword to define the inner radius of the radial grid.

[120] *Radial grids are not currently implemented in this version of OPM Flow, but is expected to be incorporated in a future release.*

6.3.94 HXFIN - DEFINE THE RATIO OF LGR GRID BLOCKS IN THE X-DIRECTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

HXFIN defines the split ratio of grid blocks for the DXV keyword in the x-direction via a vector within a Local Grid Refinement ("LGR") as opposed to defining the size for each cell for a Cartesian LGR Grid. The LGR keyword in the RUNSPEC section should be activated to indicate an LGR is being used, and the keyword HXFIN should be placed in between the CARFIN and ENDFIN keywords in the GRID section. The DXV keyword in the GRID section defines the grid size in terms of the length, that is feet for field units, this keyword defines the length as the ratio of the coarse cells.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	HXFIN	HXFIN is a vector of real numbers describing the ratio of cell size for the grid blocks in the x-direction in a Cartesian LGR grid. Repeat counts may be used, for example 2*2.0.			
		dimensionless	dimensionless	dimensionless	None

Notes:

1) The number of entries should correspond to the NX parameter on the CARFIN keyword in the GRID section.
2) The keyword is terminated by a "/".

Table 6.48: HXFIN Keyword Description

See also the CARFIN, ENDFIN, HYFIN, and HZFIN keywords in the GRID section to fully define a Cartesian LGR grid model.

Example

```
--
--       CARFIN LGR GRID COMMANDS
--
--       LGR        ----- HOST GRID ------    -- CARFIN GRID --   MAX     HOST
--       NAME       I1  I2  J1  J2  K1  K2     NX    NY    NZ   WELLS   NAME
CARFIN
         LGR-OP01   24  25  86  87   1  50      5     3    50     1     GLOBAL /
--
--       DEFINE LGR GRID BLOCK IN THE X-DIRECTION
NXFIN
         3  2                                                              /
--
--       DEFINE GRID BLOCK LGR RATIOS IN THE X-DIRECTION
--
HXFIN
         1.00  2.00  3.00  2.00  1.00                                      /
ENDFIN
```

The above example defines the size of the cells in the x-direction based on NX equals five on the CARFIN keyword in the GRID section.

6.3.95 HYFIN - DEFINE THE RATIO OF LGR GRID BLOCKS IN THE Y-DIRECTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

HYFIN defines the split ratio of grid blocks for the DYV keyword in the y-direction via a vector within a Local Grid Refinement ("LGR") as opposed to defining the size for each cell for a Cartesian LGR Grid. The LGR keyword in the RUNSPEC section should be activated to indicate an LGR is being used, and the keyword HYFIN should be placed in between the CARFIN and ENDFIN keywords in the GRID section. The DYV keyword in the GRID section defines the grid size in terms of the length, that is feet for field units, this keyword defines the length as the ratio of the coarse cells.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	HYFIN	HYFIN is a vector of real numbers describing the ratio of cell size for the grid blocks in the y-direction in a Cartesian LGR grid. Repeat counts may be used, for example 2*2.0.			
		dimensionless	dimensionless	dimensionless	None

Notes:

1) The number of entries should correspond to the NY parameter on the CARFIN keyword in the GRID section.
2) The keyword is terminated by a "/".

Table 6.49: HYFIN Keyword Description

See also the CARFIN, ENDFIN, HXFIN, and HZFIN keywords in the GRID section to fully define a Cartesian LGR grid model.

Example

```
--
--       CARFIN LGR GRID COMMANDS
--
--       LGR        ----- HOST GRID ------    -- CARFIN GRID --   MAX     HOST
--       NAME       I1  I2  J1  J2  K1  K2     NX    NY    NZ   WELLS   NAME
CARFIN
         LGR-OP01   24  25  86  87   1  50      3     5    50     1     GLOBAL /
--
--       DEFINE LGR GRID BLOCK IN THE Y-DIRECTION
NYFIN
         3  2                                                              /
--
--       DEFINE GRID BLOCK LGR RATIOS IN THE Y-DIRECTION
--
HYFIN
         1.00  2.00  3.00  2.00  1.00                                      /
ENDFIN
```

The above example defines the size of the cells in the y-direction based on NY equals five on the CARFIN keyword in the GRID section.

6.3.96 HZFIN - DEFINE THE RATIO OF LGR GRID BLOCKS IN THE Z-DIRECTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

HZFIN defines the split ratio of grid blocks for the DZV keyword in the z-direction via a vector within a Local Grid Refinement ("LGR") as opposed to defining the size for each cell for a Cartesian LGR Grid. The LGR keyword in the RUNSPEC section should be activated to indicate an LGR is being used, and the keyword HYFIN should be placed in between the CARFIN and ENDFIN keywords in the GRID section. The DZV keyword in the GRID section defines the grid size in terms of the length, that is feet for field units, this keyword defines the length as the ratio of the coarse cells.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	HYZIN	HZFIN is a vector of real numbers describing the ratio of cell size for the grid blocks in the z-direction in a Cartesian LGR grid. Repeat counts may be used, for example 2*2.0.			
		dimensionless	dimensionless	dimensionless	None

Notes:

1) The number of entries should correspond to the NZ parameter on the CARFIN keyword in the GRID section.
2) The keyword is terminated by a "/".

Table 6.50: HZFIN Keyword Description

See also the CARFIN, ENDFIN, HXFIN, and HYFIN keywords in the GRID section to fully define a Cartesian LGR grid model.

Example

```
--
--       CARFIN LGR GRID COMMANDS
--
--       LGR        ----- HOST GRID ------    -- CARFIN GRID --   MAX     HOST
--       NAME       I1  I2  J1  J2  K1  K2     NX    NY    NZ   WELLS   NAME
CARFIN
         LGR-OP01   24  25  86  87   1  50      5     3    100    1     GLOBAL /
--
--       DEFINE LGR GRID BLOCK IN THE Z-DIRECTION
NZFIN
         50*2                                                              /
--
--       DEFINE GRID BLOCK LGR RATIOS IN THE Z-DIRECTION
--
HZFIN
         50*2.0                                                            /
ENDFIN
```

The above example defines the size of the cells in the z-direction based on NZ equals 100 on the CARFIN keyword in the GRID section.

6.3.97 IHOST – ASSIGN LGRS TO PARALLEL PROCESS NUMBER

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The IHIST keyword assigns Local Grid Refinements ("LGR") to a parallel process number, for when the PARALLEL keyword has been invoked in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

6.3.98 IMPORT – IMPORT GRID FILE DATA AT THE CURRENT POSITION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The IMPORT keyword informs the simulator to import data from the specified IMPORT file. When the end of the IMPORT file is reached, input data is read from the next keyword in the current file. Normally IMPORT files are generated by grid pre-processing software and the keyword allows for both formatted and unformatted (binary) files to be loaded. The keyword can be used to import any valid grid arrays within a section, for example the EQLNUM array in the REGIONS section

No.	Name	Description	Default
1	FILENAME	A character string enclosed in quotes that defines a file to be imported and to be processed by OPM Flow.	None
2	FMTOPT	A defined character string that defines the format of the file to be imported and should be set to one of the following: 1) FORMATTED: If the file is formatted as ASCII i.e. a text file, as oppose to a binary file. The option can be abbreviated to just the letter F. 2) UNFORMATTED: If the file is in binary format, note this option can be abbreviated to just the letter U. If the variable FMTOPT is omitted then the default is for binary file input.	U
Notes: 1) The keyword is terminated by a "/".			

Table 6.51: IMPORT Keyword Description

Examples

The first example shown below loads the grid file from the same directory as the data file.

```
--
--       LOAD A IMPORT FILE
--
IMPORT
         'NOR-OPM-A00-GRID.EGRID'  /
```

The next example loads the same file one directory above from where the data file is located.

```
--
--       LOAD A IMPORT FILE
--
IMPORT
         '../NOR-OPM-A00-GRID.EGRID'  /
```

The final example loads the same file from a separate include subdirectory in the parent directory of the data file.

```
--
--       LOAD A IMPORT FILE
--
IMPORT
         '../INCLUDE/NOR-OPM-A00-GRID.EGRID'  /
```

6.3.99 INCLUDE – LOAD ANOTHER DATA FILE AT THE CURRENT POSITION

The INCLUDE keyword informs OPM Flow to continue reading input data from the specified INCLUDE file. When the end of the INCLUDE file is reached, or the ENDINC keyword is encountered in the included file, input data is read from the next keyword in the current file.

See *INCLUDE – Load Another Data File at the Current Position* in the GLOBAL section for a full description.

6.3.100 INIT – ACTIVATE THE INIT FILE OPTION

Description

This keyword switches on the writing of the INIT file that contains the static data specified in the GRID, PROPS and REGIONS sections. For example, the PORO, PERM and NTG arrays from the GRID section. The data is used in post-processing software, for example OPM ResInsight, to visualize the static grid properties.

The INIT file can either be written out in formatted form as ASCII i.e. text files, if the FMTOUT keyword has been activated, or binary format if the FMTOUT keyword has not been activated. Normally, this option is always activated by the user and when activated the binary form of the file is used.

There is no data required for this keyword and there is no terminating "/" for this keyword.

Example

```
--
--       ACTIVATE WRITING THE INIT FILE FOR POST-PROCESSING
INIT
```

The above example switches on the writing of the INIT file for post-processing in ResInsight.

6.3.101 INRAD – DEFINE THE INNER RADIUS OF A RADIAL GRID

Description

INRAD defines the inner radius of the reservoir model for a radial grid geometry. The RADIAL or SPIDER keyword in the RUNSPEC should be activated to indicate that radial geometry is being used.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	INRAD	A single real positive number defining the inner radius of a radial grid.			
		feet	m	cm	None
Notes: 1) The keyword is terminated by a "/".					

Table 6.52: INRAD Keyword Description

Note that the SPIDER keyword activates OPM Flow's radial grid geometry option for the model. This keyword will create a spiderweb-shaped grid based on a corner-point grid using the standard radial grid keywords: INRAD, DRV, DTHETAV, DZ/DZV etc. in the GRID the section. A spider grid can be viewed in 3D in OPM ResInsight unlike radial grids that cannot be viewed in the software. To overcome this, the simulator now converts radial grids to Irregular Corner-Point Grids and adjusts the model's pore volume to reflect radial coordinates; thus, overcoming the display limitation.

See also the DR, DRV, DTHETA, DTHETAV and TOPS keywords to fully define a radial or spider grid in the model.

Example

```
--
--       INNER RADIUS OF FIRST GRID BLOCK IN THE RADIAL DIRECTION
--
INRAD
         0.25                                                                  /
```

The above example defines the inner radius of a radial grid to be 0.25 feet.

6.3.102 IONROCK – DEFINE THE ION EXCHANGE CAPACITY FOR ALL THE CELLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The IONROCK keyword defines the ion exchange capacity for all the cells in the model, for when the brine phase has been activated by the BRINE keyword and the Multi-Component Brine model, that allows for the water phase to have multiple water salinities, has been activated by the ECLMC keyword. Both keywords are in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

6.3.103 ISOLNUM – DEFINE THE INDEPENDENT RESERVOIR REGIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The ISOLNUM keyword defines areas of the grid that consists of isolated reservoirs where the only form of communication between the reservoirs is via wellbore connections This enables the reservoir flow equations to be solved independently for greater computational efficiency.

No.	Name	Description	Default
1	ISOLNUM	ISOLNUM defines an array of positive integers assigning a grid cell to a particular isolated reservoir region. The maximum number of ISOLNUM regions is set by the NRFREG variable on the REGDIMS keyword in the RUNSPEC section.	1

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) Setting ISOLNUM for a cell to zero makes the cell inactive.
3) The keyword is terminated by a "/".

Table 6.53: ISOLNUM Keyword Description

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

Example

The example below defines three separate independent reservoirs; the first reservoir covers the whole grid and layers 1 to 50, reservoir two cover the whole grid and layers 52 to 150, and finally the third reservoir again covers the whole grid but with layers 152 to 300. The layers 51 and 151 are shale layers made inactive by setting ISOLNUM to zero.

```
--
--        ARRAY     CONSTANT       ---------- BOX ---------
--                                 I1  I2   J1  J2   K1  K2
EQUALS
          ISOLNUM    1             1*  1*   1*  1*   1   50 / DEFINED RESERVOIR 1
          ISOLNUM    0             1*  1*   1*  1*   51  51 / DEFINED A SHALE
          ISOLNUM    2             1*  1*   1*  1*   52 150 / DEFINED RESERVOIR 2
          ISOLNUM    0             1*  1*   1*  1*  151 151 / DEFINED A SHALE
          ISOLNUM    3             1*  1*   1*  1*  152 300 / DEFINED RESERVOIR 3
/
```

Note the above example has no effect as the keyword is ignored by the simulator.

6.3.104 JFUNC - ACTIVATE THE LEVERETT J-FUNCTION OPTION

Description

The JFUNC keyword activates the Leverett-J-Function[121] option which is a commonly used technique to normalize capillary pressure based on laboratory measured core plugs porosity and permeability values and the resulting capillary pressure data. The keyword performs the calculation based on the parameters on the this keyword combined with a cells porosity and permeability to perform the scaling globally.

The keyword should only be used if end-point scaling is switched on using the ENDSCALE keyword in the RUNSPEC section.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	JFOPT	A character string that defines which capillary data sets the J-Function option should be applied to, based on the following options: 1) WATER: apply the J-Function option to the water-oil capillary pressure data only. 2) GAS: apply the J-Function option to the gas-oil capillary pressure data only. 3) BOTH: apply the J-Function option to the water-oil and the gas-oil capillary pressure data.			BOTH
2	OWSTEN	A positive real number that defines oil-water surface tension used to de-normalized J-Function data entered in the PROPS section.			
		dynes/cm	dynes/cm	dynes/cm	None
3	OGSTEN	A positive real number that defines oil-gas surface tension used to de-normalized J-Function data entered in the PROPS section.			
		dynes/cm	dynes/cm	dynes/cm	None
4	ALPHA	A positive real value that defines an alternative power value for the porosity term in the J-Function equation, that is instead of $\sqrt{\frac{k}{\varphi}}$ use $\frac{k^{0.5}}{\varphi^{\alpha}}$ instead in the transformation.			0.5
5	BETA	A positive real number that defines an alternative power value for the permeability term in the J-Function equation, that is instead of $\sqrt{\frac{k}{\varphi}}$ use $\frac{k^{\beta}}{\varphi^{0.5}}$ instead in the transformation.			0.5

[121] *Leverett, M. C.; "Capillary Behaviour in Porous Solids", Trans. AIME (1941) 142, 152-168.*

No.	Name	Description			Default
		Field	Metric	Laboratory	
6	PERM	PERM is a character string that sets the permeability array to be used in the transform, based on the following options: 1) X: use the PERMX array. 2) XY: use the average of the PERMX and PERMY arrays. 3) Y: use the PERMY array. 4) Z: use the PERMZ array. 5) U: use the PERMJFUN array			XY
Notes: 1) The keyword is terminated by a "/".					

Table 6.54: JFUNC Keyword Description

Just like the relative permeability data capillary pressure data are measured on core plugs with varying quality and perhaps from different reservoirs. It is therefore necessary to determine averaged data, before employing the data in engineering calculations. This is commonly done by using the Leverett J-function[122], which is defined as:

$$J\left(S_w\right) = \frac{P_{c,res}\left(S_w\right)\sqrt{\frac{k}{\varphi}}}{\sigma} \tag{6.4}$$

Where:

$J(S_w)$	=	dimensionless function of water saturation
$P_c(S_w)$	=	capillary pressure (kPa)
k	=	permeability, (m^2)
φ	=	porosity (fraction)
σ	=	interfacial tension (mN/m)
Θ	=	contact angle

Sometimes the equation is stated with the cos θ term included, that is:

$$J\left(S_w\right) = \frac{P_{c,res}\left(S_w\right)\sqrt{\frac{k}{\varphi}}}{\sigma\ \cos\ \Theta} \tag{6.5}$$

Since the above function is just a normalizing function, then units are not important, as long as when we de-normalize the *average* curve we use the same unit set. Secondly, if all the capillary pressure data has been converted to reservoir conditions, we actually ignore the denominator as it is a constant, and we can therefore just use:

$$J\left(S_w\right) = P_{c,res}\left(S_w\right)\sqrt{\frac{k}{\varphi}} \tag{6.6}$$

[122] *Leverett, M. C.; "Capillary Behaviour in Porous Solids", Trans. AIME (1941) 142, 152-168.*

However, in the simulator it is necessary to use the formal definition as outlined in equation (6.4). In addition to the standard the equation the keyword allows for de-normalizing the curve to use alternative power functions instead of the standard 0.5 used in equation (6.4), that is:

$$J\left(S_w\right) = \frac{P_{c,res}\left(S_w\right)\left(\frac{k^{\beta}}{\varphi^{\alpha}}\right)}{\sigma} \tag{6.7}$$

Where:

$J(S_w)$	=	dimensionless function of water saturation
$P_c(S_w)$	=	capillary pressure (kPa)
k	=	permeability, (m^2)
φ	=	porosity (fraction)
σ	=	interfacial tension (mN/m)
Θ	=	contact angle
α	=	porosity power value
β	=	permeability value

The JFUNC keyword allows the data entered as capillary pressure in the saturation tables, for example, by using the SGFN and SWFN keywords in the PROPS section to be treated as J-functions instead, and to de-normalize these curves for each active cell in the model using the options and values defined with the JFUNC keyword combined with a cells porosity and permeability values.

Note

If either the JFUNC or JFUNCR keywords are used to activate J-Function scaling then the ENDSCALE keyword in the RUNSPEC section must also be present in the input deck, in order for the dimensionless J-function values entered on the SWFN, SGFN or the SWOF, SGOF, SLGOF keywords to be re-scaled to capillary pressure data.

Note if the ENDSCALE keyword is absent, then like the commercial simulator, J-Function scaling is not performed, and the values entered on the SWFN, SGFN or the SWOF, SGOF, SLGOF keywords are used as entered.

See also the JFUNCR keyword in the GRID section that performs similar calculations based on the J-Function parameters being declared by saturation table number.

Example

```
--
--       DEFINE LEVERETT J-FUNCTION PARAMETERS
--       JFUN    OILWAT  GASOIL  PORO    PERM    PERM
--       OPTN    SDENS   SDEN    ALPHA   BETA    OPTN
JFUNC
         WATER   22.5    1*      0.5     0.5     XY                    /
```

The above example results in the oil-water capillary pressure data entered on the SWFN keyword in the PROPS section being treated as J-Functions, and that the J-Function should be de-normalized using an oil-water surface density of 22.5 dynes/cm, using the default power values and the average of the PERMX and PERMY values for each grid block.

6.3.105 JFUNCR - ACTIVATE THE LEVERETT J-FUNCTION SATURATION TABLE OPTION

Description

JFUNCR keyword activates Leverett-J-Function[123] Saturation Table option which is a commonly used technique to normalize capillary pressure base on laboratory measured core plugs porosity and permeability values and the resulting capillary pressure data. This keyword is an extension of the JFUNC keyword in the GRID section that uses the parameters on the JFUNC keyword combined with a cell's porosity and permeability to perform the scaling globally. In comparison, the JFUNCR allows for the J-Function parameters to be declared per saturation table number, resulting in greater flexibility.

The keyword should only be used if end-point scaling is switched on using the ENDSCALE keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	JFOPT	A character string that defines which capillary data sets the J-Function option should be applied to, based on the following options: 1) WATER: apply the J-Function option to the water-oil capillary pressure data only. 2) GAS: apply the J-Function option to the gas-oil capillary pressure data only. 3) BOTH: apply the J-Function option to the water-oil and the gas-oil capillary pressure data.			BOTH
2	OWSTEN	A positive real number that defines oil-water surface tension used to de-normalized J-Function data entered in the PROPS section.			
		dynes/cm	dynes/cm	dynes/cm	None
3	OGSTEN	A positive real number that defines oil-gas surface tension used to de-normalized J-Function data entered in the PROPS section.			
		dynes/cm	dynes/cm	dynes/cm	None
4	ALPHA	A positive real value that defines an alternative power value for the porosity term in the J-Function equation, that is instead of $\sqrt{\frac{k}{\varphi}}$ use $\frac{k^{0.5}}{\varphi^{\alpha}}$ instead in the transformation.			0.5
5	BETA	A positive real number that defines an alternative power value for the permeability term in the J-Function equation, that is instead of $\sqrt{\frac{k}{\varphi}}$ use $\frac{k^{\beta}}{\varphi^{0.5}}$ instead in the transformation.			0.5

[123] *Leverett, M. C.; "Capillary Behaviour in Porous Solids", Trans. AIME (1941) 142, 152-168.*

No.	Name	Description			Default
		Field	Metric	Laboratory	
6	PERM	PERM is a character string that sets the permeability array to be used in the transform, based on the following options: 1) X: use the PERMX array. 2) XY: use the average of the PERMX and PERMY arrays. 3) Y: use the PERMY array. 4) Z: use the PERMZ array. 5) U: use the PERMJFUN array			XY
Notes: 1) The keyword is followed by exactly NTSFUN rows of data, as declared on the TABDIMS keyword in the RUNSPEC section. 2) Each row is terminated by a "/" and there is no "/" terminator for the keyword.					

Table 6.55: JFUNCR Keyword Description

Just like the relative permeability data capillary pressure data are measured on core plugs with varying quality and perhaps from different reservoirs. It is therefore necessary to determine averaged data, before employing the data in engineering calculations. This is commonly done by using the Leverett J-function[124], which is defined as:

$$J\left(S_w\right) = \frac{P_{c,res}\left(S_w\right)\sqrt{\frac{k}{\varphi}}}{\sigma} \tag{6.8}$$

Where:

$J(S_w)$	=	dimensionless function of water saturation
$P_c(S_w)$	=	capillary pressure (kPa)
k	=	permeability, (m^2)
φ	=	porosity (fraction)
σ	=	interfacial tension (mN/m)
Θ	=	contact angle

Sometimes the equation is stated with the cos θ term included, that is:

$$J\left(S_w\right) = \frac{P_{c,res}\left(S_w\right)\sqrt{\frac{k}{\varphi}}}{\sigma\ \cos\ \Theta} \tag{6.9}$$

Since the above function is just a normalizing function, then units are not important, as long as when we de-normalize the *average* curve we use the same unit set. Secondly, if all the capillary pressure data has been converted to reservoir conditions, we can actually ignore the denominator as it is a constant, and we can therefore just use:

$$J\left(S_w\right) = P_{c,res}\left(S_w\right)\sqrt{\frac{k}{\varphi}} \tag{6.10}$$

[124] *Leverett, M. C.; "Capillary Behaviour in Porous Solids", Trans. AIME (1941) 142, 152-168.*

However, in the simulator it is necessary to use the formal definition as outlined in equation (6.8). In addition to the standard the equation the keyword allows for de-normalizing the curve to use alternative power functions instead of the standard 0.5 used in equation (6.8), that is:

$$J\left(S_w\right) = \frac{P_{c,res}\left(S_w\right)\left(\frac{k^{\beta}}{\varphi^{\alpha}}\right)}{\sigma} \tag{6.11}$$

Where:

$J(S_w)$	=	dimensionless function of water saturation
$P_c(S_w)$	=	capillary pressure (kPa)
k	=	permeability, (m^2)
φ	=	porosity (fraction)
σ	=	interfacial tension (mN/m)
Θ	=	contact angle
α	=	porosity power value
β	=	permeability value

The JFUNC keyword allows the data entered as capillary pressure in the saturation tables, for example, by using the SGFN and SWFN keywords in the PROPS section to be treated as J-functions instead, and to de-normalize these curves for each active cell in the model using the options and values defined with the JFUNC keyword combined with a cells porosity and permeability values.

Note

If either the JFUNC or JFUNCR keywords are used to activate J-Function scaling then the ENDSCALE keyword in the RUNSPEC section must also be present in the input deck, in order for the dimensionless J-function values entered on the SWFN, SGFN or the SWOF, SGOF, SLGOF keywords to be re-scaled to capillary pressure data.

Note if the ENDSCALE keyword is absent, then like the commercial simulator, J-Function scaling is not performed, and the values entered on the SWFN, SGFN or the SWOF, SGOF, SLGOF keywords are used as entered.

See also the JFUNC keyword in the GRID section that uses the parameters on the JFUNC keyword combined with a cell's porosity and permeability to perform the scaling globally.

Example

The example below assumes NTSFUN is equal to five on the TABDIMS keyword in the RUNSPEC section.

```
--
--       DEFINE LEVERETT J-FUNCTION PARAMETERS BY SATURATION TABLES
--       JFUN    OILWAT  GASOIL  PORO    PERM    PERM
--       OPTN    SDENS   SDEN    ALPHA   BETA    OPTN
JFUNCR
         WATER   22.5    1*      0.5     0.5     XY                    /
         WATER   22.5    1*      0.5     0.5     XY                    /
         WATER   22.5    1*      0.5     0.5     XY                    /
         WATER   22.5    1*      0.5     0.5     XY                    /
         WATER   22.5    1*      0.5     0.5     XY                    /
```

Here the oil-water capillary pressure data entered on the SWFN keyword in the PROPS section are treated as J-Functions, and that the J-Function should be de-normalized using an oil-water surface density of 22.5 dynes/cm, using the default power values and the average of the PERMX and PERMY values for each grid block, for all five tables. Note that since all the JFUNCR parameters are the same for all saturation tables then the JFUNC keyword could be used instead in this instance.

6.3.106 LGRCOPY – ACTIVATE LOCAL GRID REFINEMENT INHERITANCE

The LGRCOPY keyword actives Local Grid Refinement ("LGR") Inheritance option that allows the LGR to inherit the properties of the global or host cell containing a LGR grid block at the start of the GRID section, as oppose to the normal process of applying this transform at the end of the GRID section. LGRCOPY can be used in the RUNSPEC, GRID and EDIT sections. If used in the RUNSPEC section then the option is applied to all LGRs defined in the input file, whereas if used in the GRID and EDIT sections the keyword must be placed inside a LGR definition section, that is between a CARFIN (Cartesian LGR grid) or RADIN/RADIN4 (radial LGR grid) and the ENDFIN keyword. In the latter case inheritance is applied on an individual LGR basis.

See *LGRCOPY – Activate Local Grid Refinement Inheritance* in the RUNSPEC section for a full description.

6.3.107 LINKPERM - ASSIGN CELL PERMEABILITIES TO CELL FACES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The LINKPERM keyword assigns the grid cell permeabilities entered via the PERMX, PERMY and PERMZ keywords to a cell face (I±, J±, or K±) and results in the simulator using these values directly in the calculating the transmissibility between grid blocks. This is different to the conventional way of entering permeability data that consists of entering the cell centered permeability and the simulator calculating a weighted average transmissibility based on the cell centered permeability of the up-stream and down-stream grid blocks.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

6.3.108 LTOSIGMA - DUAL POROSITY VISCOUS DISPLACEMENT SIGMA PARAMETERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The LTOSIGMA keyword defines parameters to calculate the sigma factor in conjunction with the data entered via the LX, LY and LZ keywords in the GRID section, for when the VISCD keyword has been used in the RUNSPEC section to activate the Dual Porosity Viscous Displacement option. In addition, either the DUALPORO or DUALPERM keyword should be entered in the RUNSPEC section to activate the dual porosity or dual permeability models.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

6.3.109 LX - DUAL POROSITY VISCOUS DISPLACEMENT X DIRECTION MATRIX SIZE FOR ALL CELLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The LX keyword defines the size of "representative" matrix grid blocks in the X direction via an array in dual porosity and dual permeability runs, for when the VISCD keyword has been used in the RUNSPEC section to activate the dual porosity viscous displacement option. In addition, either the DUALPORO or DUALPERM keyword should be entered in the RUNSPEC section to activate the dual porosity or dual permeability models. The VISCD option is used to model the viscous displacement of fluids from the matrix by the fracture pressure gradient, for when the fracture system has a more moderate permeability, and flow to and from the matrix caused by the fracture pressure gradient acts as an additional production mechanism.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	LX	LX is an array of real numbers describing the "representative" cell size in the X direction for each cell in the model. Repeat counts may be used, for example 10*100.0.			
		feet	m	cm	0

Notes:

1) The number of entries should correspond to the NX x NY x (NZ/2) parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.

2) The keyword is terminated by a "/".

Table 6.56: LX Keyword Description

If the VISCD keyword has been used to activate the Dual Porosity Viscous Displacement option and LX has not been specified then LX is set to zero in the calculation of the viscous displacement term. See also the LY, LZ and LTOSIGMA keywords in the GRID section.

Example

```
--
--       DEFINE INPUT BOX FOR EDITING INPUT ARRAYS
--
--       ---------- BOX ---------
--       I1  I2   J1  J2   K1  K2
BOX
         10  10   1   6    1   1                          / DEFINE BOX AREA
--
--       DEFINE DUAL POROSITY VISCOUS DISPLACEMENT X DIRECTION MATRIX SIZE
--
LX
         6*10.0                                           /
--
--       DEFINE END OF INPUT BOX EDITING OF INPUT ARRAYS
--
ENDBOX
```

The above example defines a subset of the grid and the size of the "representative" matrix cells in the X direction to 10.0 ft.; after which the ENDBOX keyword resets the input to be the full grid.

6.3.110 LXFIN – DEFINE LOGARITHMIC LGR GRID BLOCK SPACING IN THE X-DIRECTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The LXFIN keyword defines the parameters for automatically generating a Local Grid Refinement ("LGR") grid in the X direction based on logarithmic block spacing, for when the LGR option has been activated by the LGR keyword in the RUNSPEC section. LXFIN should be placed in between the CARFIN and ENDFIN keywords in the GRID section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

6.3.111 LY - DUAL POROSITY VISCOUS DISPLACEMENT Y DIRECTION MATRIX SIZE FOR ALL CELLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The LY keyword defines the size of "representative" matrix grid blocks in the Y direction via an array in dual porosity and dual permeability runs, for when the VISCD keyword has been used in the RUNSPEC section to activate the dual porosity viscous displacement option. In addition, either the DUALPORO or DUALPERM keyword should be entered in the RUNSPEC section to activate the dual porosity or dual permeability models. The VISCD option is used to model the viscous displacement of fluids from the matrix by the fracture pressure gradient, for when the fracture system has a more moderate permeability, and flow to and from the matrix caused by the fracture pressure gradient acts as an additional production mechanism.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	LY	LY is an array of real numbers describing the "representative" cell size in the Y direction for each cell in the model. Repeat counts may be used, for example 10*100.0.			
		feet	m	cm	0

Notes:

1) The number of entries should correspond to the NX x NY x (NZ/2) parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.

2) The keyword is terminated by a "/".

Table 6.57: LY Keyword Description

If the VISCD keyword has been used to activate the Dual Porosity Viscous Displacement option and LY has not been specified then LY is set to zero in the calculation of the viscous displacement term.

See also the LX, LZ and LTOSIGMA keywords in the GRID section.

Example

```
--
--       DEFINE INPUT BOX FOR EDITING INPUT ARRAYS
--
--       ---------- BOX ---------
--       I1  I2   J1  J2   K1  K2
BOX
         10  10   1   6    1   1                          / DEFINE BOX AREA
--
--       DEFINE DUAL POROSITY VISCOUS DISPLACEMENT Y DIRECTION MATRIX SIZE
--
LY
         6*15.0                                           /
--
--       DEFINE END OF INPUT BOX EDITING OF INPUT ARRAYS
--
ENDBOX
```

The above example defines a subset of the grid and the size of the "representative" matrix cells in the Y direction to 15.0 ft.; after which the ENDBOX keyword resets the input to be the full grid.

6.3.112 LYFIN – DEFINE LOGARITHMIC LGR GRID BLOCK SPACING IN THE Y-DIRECTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The LYFIN keyword defines the parameters for automatically generating a Local Grid Refinement ("LGR") grid in the Y direction based on logarithmic block spacing, for when the LGR option has been activated by the LGR keyword in the RUNSPEC section. LYFIN should be placed in between the CARFIN and ENDFIN keywords in the GRID section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

6.3.113 LZ - DUAL POROSITY VISCOUS DISPLACEMENT Z DIRECTION MATRIX SIZE FOR ALL CELLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The LZ keyword defines the size of "representative" matrix grid blocks in the Z direction via an array in dual porosity and dual permeability runs, for when the VISCD keyword has been used in the RUNSPEC section to activate the dual porosity viscous displacement option. In addition, either the DUALPORO or DUALPERM keyword should be entered in the RUNSPEC section to activate the dual porosity or dual permeability models. The VISCD option is used to model the viscous displacement of fluids from the matrix by the fracture pressure gradient, for when the fracture system has a more moderate permeability, and flow to and from the matrix caused by the fracture pressure gradient acts as an additional production mechanism.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	LZ	LZ is an array of real numbers describing the "representative" cell size in the Z direction for each cell in the model. Repeat counts may be used, for example 10*100.0.			
		feet	m	cm	0

Notes:

1) The number of entries should correspond to the NX x NY x (NZ/2) parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.

2) The keyword is terminated by a "/".

Table 6.58: LZ Keyword Description

If the VISCD keyword has been used to activate the Dual Porosity Viscous Displacement option and LZ has not been specified then LZ is set to zero in the calculation of the viscous displacement term.

See also the LX, LY and LTOSIGMA keywords in the GRID section.

Example

```
--
--       DEFINE INPUT BOX FOR EDITING INPUT ARRAYS
--
--       ---------- BOX ---------
--       I1  I2   J1  J2   K1  K2
BOX
         10  10   1   6    1   1                          / DEFINE BOX AREA
--
--       DEFINE DUAL POROSITY VISCOUS DISPLACEMENT Z DIRECTION MATRIX SIZE
--
LZ
         6*3.0                                            /
--
--       DEFINE END OF INPUT BOX EDITING OF INPUT ARRAYS
--
ENDBOX
```

The example defines a subset of the grid and the size of the "representative" matrix cells in the Y direction to 15.0 ft.; after which the ENDBOX keyword resets the input to be the full grid.

6.3.114 LZFIN – DEFINE LOGARITHMIC LGR GRID BLOCK SPACING IN THE Z-DIRECTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The LZFIN keyword defines the parameters for automatically generating a Local Grid Refinement ("LGR") grid in the Z direction based on logarithmic block spacing, for when the LGR option has been activated by the LGR keyword in the RUNSPEC section. LZFIN should be placed in between the CARFIN and ENDFIN keywords in the GRID section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

6.3.115 MAPAXES - DEFINE THE MAP ORIGIN INPUT DATA

Description

MAPAXES specifies the origin of the map used to create the grid. It is usually output by pre-processing software when exporting the grid geometry. The data is not used by OPM Flow intrinsically, but is merely written to the output EGRID file, as specified by the GRIDFILE keyword, for the use of post-processing software like OPM ResInsight.

No.	Name	Description			Default
		Field (feet)	Metric (metres)	Laboratory (metres)	
1	X1	X1 is a real number that defines the x co-ordinate of a point on the y-axis.			None
2	Y1	Y1 is a real number that defines the y co-ordinate of a point on the y-axis.			None
3	X2	X2 is a real number that defines the x co-ordinate of the origin.			None
4	Y2	Y2 is a real number that defines the y co-ordinate of the origin.			None
5	X3	X3 is a real number that defines the x co-ordinate of a point on the x-axis.			None
6	Y3	Y3 is a real number that defines the y co-ordinate of a point on the x-axis.			None
Notes: 1) The keyword is terminated by a "/".					

Table 6.59: MAPAXES Keyword Description

Example

```
--
--          ---------------- MAPAXES -----------------
--          X1       Y1       X2       Y2       X3       Y3
MAPAXES
            0.0      100.0    0.0      0.0      100.0    0.0    /
```

The above example defines the map axes to be exported to the grid file for use by post-processing software.

6.3.116 MAPUNITS – DEFINE THE MAP AXES UNITS

Description

The MAPUNITS keyword defines the units of the coordinates stated on the MAPAXES keyword. It is usually output by pre-processing software when exporting the grid geometry. The data is not used by OPM Flow intrinsically, but is merely written to the output EGRID file, as specified by the GRIDFILE keyword, for the use of post-processing software like OPM ResInsight.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

No.	Name	Description	Default
1	MAPUNITS	A character string that defines the units of the coordinates stated on the MAPAXES keyword, and should be set to: 1) FEET for field units 2) METRES for metric units, or 3) CM for laboratory units	METRES
Notes: 1) Note the alternative spelling of METRES, that is METERS is not recognized. 2) The keyword is terminated by a "/".			

Table 6.60: MAPUNITS Keyword Description

Example

```
--
--       SET THE MAP UNITS FOR THE MAPAXES KEYWORD
MAPUNITS
         METRES                                                    /
```

The above example specifies the units on the MAPAXES to be the default METRES.

6.3.117 MAXVALUE – SETS A MAXIMUM VALUE FOR AN ARRAY ELEMENT

Description

The MAXVALUE keyword sets a maximum value for the specified array or part of an array. The constant can be real or integer depending on the array type; however, the arrays that can be operated on is dependent on which section the MAXVALUE keyword is being used.

No.	Name	Description	Default
1	ARRAY	The name of the array to be modified. This is the keyword name identifying the property and is up to eight characters in length and enclosed in quotes.	None
2	CONSTANT	CONSTANT is a positive integer or positive real value that an ARRAY element will be reset to if an element in the defined input BOX, as defined by items (3) to (8), is greater than CONSTANT. CONSTANT has in the same units as the ARRAY property.	None
3	I1	The lower bound of the array in the I-direction to be modified must be greater than or equal to one and less than or equal to I2 and NX.	1
4	I2	The upper bound of the array in the I-direction to be modified must be greater than or equal to I1 and less than or equal to NX	NX
5	J1	The lower bound of the array in the J-direction to be modified must be greater than or equal to one and less than or equal to J2 and NY.	1
6	J2	The upper bound of the array in the J-direction to be modified must be greater than or equal to J1 and less than or equal to NY.	NY
7	K1	The lower bound of the array in the K-direction to be modified must be greater than or equal to one and less than or equal to K2 and NZ.	1
8	K2	The upper bound of the array in the K-direction to be modified must be greater than or equal to K1 and less than or equal to NZ.	NZ

Notes:

1) Where NX, NY and NZ are the dimensions of the model as defined on the DIMENS keyword in the RUNSPEC section.
2) Each record must be terminated by a "/" and the keyword is terminated by a "/".

Table 6.61: MAXVALUE Keyword Description

The applicable arrays for each section are defined in Table 6.62 on the following page.

MAXVALUE Keyword And Variable Options By Section						
GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
DX	DEPTH	SWL				
DY	PORV	SWCR				
DZ	TRANX	SWU				
PERMX	TRANY	SGL				
PERMY	TRANZ	SGCR				
PERMZ	DIFFX	SGU				
MULTX	DIFFY	KRW				
MULTY	DIFFZ	KRO				
MULTZ	TRANR	KRG				
DR	TRANTHT	PCG				
THETA	DIFFR	PCW				
PERMR	DIFFTHT					
PERMTHT						
DZNET						
PORO						
NTG						
FLUXNUM						
MULTNUM						
MPANUM						
DIFFX						
DIFFY						
DIFFZ						
DIFFR						
DIFFTHT						

Table 6.62: MAXVALUE Keyword Applicable Arrays by Section

Example

```
--
--        ARRAY      CONSTANT        ---------- BOX ---------
--                                   I1  I2    J1  J2    K1  K2
MAXVALUE
          PERMX      1.0E2           1*  1*    1*  1*    1*  1* / MAX VALUE FOR PERMX
          PERMY      1.0E2           1*  1*    1*  1*    1*  1* / MAX VALUE FOR PERMY
          PERMZ      1.0E1           1*  1*    1*  1*    1*  1* / MAX VALUE FOR PERMZ
/
```

The above example resets the maximum values for the PERMX, PERMY and PERMZ, arrays to 100.0, 100.0 and 10.0, respectively, for all cells.

6.3.118 MESSAGE – OUTPUT USER MESSAGE

The MESSAGE keyword outputs a user message to the terminal, as well as to the print (*.PRT) and debug (*.DBG) files. Note this is different to the MESSAGES keyword, that defines OPM Flows message print limits and stop limits generated by the simulator.

See *MESSAGE – Output User Message* in the GLOBAL section for a full description.

6.3.119 MESSAGES – DEFINE MESSAGE PRINT LIMITS AND STOP LIMITS

The MESSAGES keyword defines the print and stops levels for various messages. The "print limits" set the maximum number of messages that will be printed, after which no more messages will be printed and the "stop limits" terminate the run when these limits are exceeded. There are six levels of message that increase in severity from informative all the way to programming errors, as outlined in Error: Reference source not found.

See *MESSAGES – Define Message Print Limits and Stop Limits* in the GLOBAL section for a full description.

6.3.120 MINNNCT – SET A MINIMUM NON-NEIGHBOR CONNECTION TRANSMISSIBILITY

Description

The MINNNCT keyword defines a minimum non-neighbor connection transmissibility below which the non-neighbor connection is deleted. The keyword allows for three minimum values, one for the transmissibility, one for the diffusivity and one for the thermal transmissibility. If the keyword is absent from the input deck then no minimum cut-off is applied.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

6.3.121 MINPORV – SET A MINIMUM GRID BLOCK PORE VOLUME THRESHOLD FOR ALL CELLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

MINPORV defines a minimum threshold pore volume that makes all grid blocks whose pore volume is below this value inactive in the model (inactive cells are not used in OPM Flow calculations). Note this keyword is different to the MINPVV keyword in the GRID section, that sets a minimum threshold pore volume for individual cells in the model.

This keyword is an alias for the MINPV keyword in the GRID section, and thus provides the same functionality.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	MINPORV	MINPORV is a real positive number that defines the minimum pore volume for a cell to be active in the model.			
		rb 1.0e-6	rm^3 1.0e-6	rcc 1.0e-6	Defined
Notes: 1) The keyword is terminated by a "/".					

Table 6.63: MINPORV Keyword Description

The MINPORV, MINPV, and MINPVV keywords only apply their minimum threshold pore volume values to active cells. Thus, cells that have been made inactive via setting their ACTNUM values to zero, remain inactive, even if their pore volume exceeds the values set by the MINPORV, MINPV, and MINPVV keywords.

Secondly, although the MINPORV keyword allows one to set a minimum threshold pore volume below the default value, this is not recommended, as cells with small pore volumes can cause significant numerical convergence errors. Thus, in practice, values greater than the default values are normally applied to eliminate cells that have relatively small pore volumes. In addition, the simulator reports the total pore volume and the number of cells made inactive, as well as the pore volume reduction, when the keyword is invoked; allowing one to run some sensitivities to the minimum pore volume value.

See also the PINCH keyword for the treatment of inactive grid cells and pinch-outs.

Example

```
--
--       MINIMUM PORE VOLUME FOR ACTIVE CELLS
--
MINPORV
         500.0                                                    /
```

The above example defines 500 rb (or m3) as the minimum pore volume for a cell to be active in the model.

6.3.122 MINPV – SET A MINIMUM GRID BLOCK PORE VOLUME THRESHOLD FOR ALL CELLS

Description

MINPV defines a minimum threshold pore volume that makes all grid blocks whose pore volume is below this value inactive in the model (inactive cells are not used in OPM Flow calculations). Note this keyword is different to the MINPVV keyword in the GRID section, that sets a minimum threshold pore volume for individual cells in the model.

Note that the MINPORV keyword is an alias for the MINPV keyword in the GRID section, and thus provides the same functionality.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	MINPV	MINPV is a real positive number that defines the minimum pore volume for a cell to be active in the model.			
		rb 1.0e-6	rm^3 1.0e-6	rcc 1.0e-6	Defined
Notes: 1) The keyword is terminated by a "/".					

Table 6.64: MINPV Keyword Description

The MINPV, MINPORV, and MINPVV keywords only apply their minimum threshold pore volume values to active cells. Thus, cells that have been made inactive via setting their ACTNUM values to zero, remain inactive, even if their pore volume exceeds the values set by the MINPV, MINPORV, and MINPVV keywords.

Secondly, although the MINPV keyword allows one to set a minimum threshold pore volume below the default value, this is not recommended, as cells with small pore volumes can cause significant numerical convergence errors. Thus, in practice, values greater than the default values are normally applied to eliminate cells that have relatively small pore volumes. In addition, the simulator reports the total pore volume and the number of cells made inactive, as well as the pore volume reduction, when the keyword is invoked; allowing one to run some sensitivities to the minimum pore volume value.

See also the PINCH keyword for the treatment of inactive grid cells and pinch-outs.

Example

```
--
--       MINIMUM PORE VOLUME FOR ACTIVE CELLS
--
MINPV
         500.0                                                    /
```

The above example defines 500 rb (or m3) as the minimum pore volume for a cell to be active in the model.

6.3.123 MINPVV – SET A MINIMUM GRID BLOCK PORE VOLUME THRESHOLD FOR INDIVIDUAL CELLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

MINPVV is an array that defines the minimum threshold pore volume for each cell, that makes grid blocks whose pore volume is below this value inactive in the model (inactive cells are not used in OPM Flow calculations).

Note this keyword is different to the MINPV and MINPORV keywords in the GRID section, that set a constant minimum threshold pore volume for all cells in the model.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	MINPVV	MINPVV is an array of real positive numbers that defines the minimum pore volumes for each cell in the model in order for the cells to be active.			
		rb 1.0e-6	rm^3 1.0e-6	rcc 1.0e-6	Defined

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.

2) The keyword is terminated by a "/".

Table 6.65: MINPVV Keyword Description

The MINPVV, MINPV, and MINPORV keywords only apply their minimum threshold pore volume values to active cells. Thus, cells that have been made inactive via setting their ACTNUM values to zero, remain inactive, even if their pore volume exceeds the values set by the MINPVV, MINPV, and MINPORV keywords.

Secondly, although the MINPVV keyword allows one to set a minimum threshold pore volume below the default value, this is not recommended, as cells with small pore volumes can cause significant numerical convergence errors. Thus, in practice, values greater than the default values are normally applied to eliminate cells that have relatively small pore volumes. In addition, the simulator reports the total pore volume and the number of cells made inactive, as well as the pore volume reduction, when the keyword is invoked; allowing one to run some sensitivities to the minimum pore volume value.

See also the PINCH keyword for the treatment of inactive grid cells and pinch-outs.

Example

The example below shows how to define 500 rb (or m3) as the minimum pore volume for all cells in layer 19 to be active in the model, and 750 rb (or m3) as the minimum pore volume for all cells in layer 20, by using the BOX keyword to set the portion of the grid of interest.

```
--
--       DEFINE A BOX GRID FOR THE BOTTOM TWO LAYERS OF A 100 X 100 X 20 MODEL
--
--       ---------- BOX ---------
--       I1  I2   J1  J2   K1  K2
BOX
         1*  1*   1*  1*   19  20 / SELECT THE BOTTOM LAYERS
--
--       MINIMUM PORE VOLUME FOR INDIVIDUAL CELLS TO BE ACTIVE
--
MINPVV
         10000*500.0   10000*750.0
/
--
--       RESET THE INPUT BOX TO BE THE FULL MODEL
--
ENDBOX
```

Although this will work in the commercial simulators, it does not currently work in OPM Flow, that is one cannot use the MINPVV keyword in conjunction with the BOX keyword, as shown in the aforementioned example.

Instead one can use:

```
--
--       MINIMUM PORE VOLUME FOR INDIVIDUAL CELLS TO BE ACTIVE
--
MINPVV
         10000*        10000*     10000*   10000*   10000*
         10000*        10000*     10000*   10000*   10000*
         10000*        10000*     10000*   10000*   10000*
         10000*        10000*     10000*
         10000*500.0   10000*750.0                                    /
```

To accomplish the same thing, where the 10000* instructs the simulator to use the default value of 1.0 x 10^{-6} for 10,000 cells, which in this case is one layer.

6.3.124 MINVALUE – SET A MINIMUM VALUE FOR AN ARRAY ELEMENT

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The MINVALUE keyword sets a minimum value for the specified array or part of an array. The constant can be real or integer depending on the array type; however, the arrays that can be operated on is dependent on which section the MINVALUE keyword is being used.

No.	Name	Description	Default
1	ARRAY	The name of the array to be modified. This is the keyword name identifying the property and is up to eight characters in length and enclosed in quotes.	None
2	CONSTANT	CONSTANT is a positive integer or positive real value that an ARRAY element will be reset to if an element in the defined input BOX, as defined by items (3) to (8), is less than CONSTANT. CONSTANT has in the same units as the ARRAY property.	None
3	I1	A positive integer that defines the lower bound of the array in the I-direction to be modified must be greater than or equal to one and less than or equal to I2 and NX.	1
4	I2	A positive integer that defines the upper bound of the array in the I-direction to be modified must be greater than or equal to I1 and less than or equal to NX	NX
5	J1	A positive integer that defines the lower bound of the array in the J-direction to be modified must be greater than or equal to one and less than or equal to J2 and NY.	1
6	J2	A positive integer that defines the upper bound of the array in the J-direction to be modified must be greater than or equal to J1 and less than or equal to NY.	NY
7	K1	A positive integer that defines the lower bound of the array in the K-direction to be modified must be greater than or equal to one and less than or equal to K2 and NZ.	1
8	K2	A positive integer that defines the upper bound of the array in the K-direction to be modified must be greater than or equal to K1 and less than or equal to NZ.	NZ

Notes:

1) Where NX, NY and NZ are the dimensions of the model as defined on the DIMENS keyword in the RUNSPEC section.
2) Each record must be terminated by a "/" and the keyword is terminated by a "/".

Table 6.66: MINVALUE Keyword Description

The applicable arrays for each section are defined in Table 6.67 on the following page.

EQUALS Keyword And Variable Options By Section						
GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
DX	DEPTH	SWL				
DY	PORV	SWCR				
DZ	TRANX	SWU				
PERMX	TRANY	SGL				
PERMY	TRANZ	SGCR				
PERMZ	DIFFX	SGU				
MULTX	DIFFY	KRW				
MULTY	DIFFZ	KRO				
MULTZ	TRANR	KRG				
DR	TRANTHT	PCG				
THETA	DIFFR	PCW				
PERMR	DIFFTHT					
PERMTHT						
DZNET						
PORO						
NTG						
FLUXNUM						
MULTNUM						
MPANUM						
DIFFX						
DIFFY						
DIFFZ						
DIFFR						
DIFFTHT						

Table 6.67: MINVALUE Keyword Applicable Arrays by Section

Example

```
--
--        ARRAY     CONSTANT         ---------- BOX ---------
--                                   I1  I2    J1  J2    K1  K2
MINVALUE
          PERMX     1.0E0            1*  1*    1*  1*    1*  1* / MINIMUM PERMX
          PERMY     1.0E0            1*  1*    1*  1*    1*  1* / MINIMUM PERMY
          PERMZ     1.0E-1           1*  1*    1*  1*    1*  1* / MINIMUM PERMZ
/
```

The above example resets the minimum values for the PERMX, PERMY and PERMZ, arrays to 1.0, 1.0 and 0.1, respectively, for all cells.

6.3.125 MPFANUM – DEFINE MULTI-POINT FLUX DISCRETIZATION REGIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The MPFANUM keyword defines regions in the model where the multi-point flux discretization should be applied.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

6.3.126 MPFNNC – DEFINE MULTI-POINT FLUX NON-NEIGHBOR CONNECTIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The MPFNNC keyword defines multi-point flux non-neighbor connections explicitly.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

6.3.127 MULTFLT – MULTIPLY THE TRANSMISSIBILITY OF A DEFINED FAULT BY A CONSTANT

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

MULTFLT enables the transmissibilities across defined faults, as declared by the FAULTS keyword, to be modified. They keyword allows for the re-scaling of the existing fault transmissibilities calculated by OPM Flow, for example setting a fault to be completely sealing by setting the multiplier to zero.

No.	Name	Description	Default
1	FLTNAME	FLTNAME is a character string enclosed in quotes with a maximum length of eight characters, that defines the name of the fault that FLTMULT will be applied to. FLTNAME must have previously been defined using the FAULTS keyword in GRID section	None
2	FLT-TRS	A positive real number that sets the transmissible multiplier to be applied to the FLTNAME transmissibilities positive real number that sets the transmissible multiplier to be applied to the FLTNAME transmissibilities.	1.0
3	FLT-DIF	A positive real number that sets the diffusivity multiplier to be applied to the FLTNAME diffusivities. This option should only be used if the Diffusion option has been made activate by the DIFFUSE keyword in the RUNSPEC section. OPM Flow does not support this Diffusion option.	1.0
Notes: 1) Repeated entries of the same FLTNAME will result in all but the last entry being overwritten. 2) Each record must be terminated by a "/" and the keyword is terminated by a "/".			

Table 6.68: MULTFLT Keyword Description

Example

```
--
--        MODIFY THE TRANSMISSIBILITES ACROSS DEFINED FAULTS
--
--        FAULT             TRANS           DIFUSS
--        NAME              MULTIPLIER      MULTIPLIER
MULTFLT
          'FAULT01'         0.0                                 / FAULT MULTIPLIERS
          'FAULT02'         0.0                                 / FAULT MULTIPLIERS
          'FAULT03'         0.0                                 / FAULT MULTIPLIERS
/
```

The above example sets the fault transmissibility multiplier for defined faults named FAULT01, FAULT02, and FAULT03 to zero making the faults sealing in the model.

6.3.128 MULTIPLY – MULTIPLY A SPECIFIED ARRAY BY A CONSTANT

Description

The MULTIPLY keyword multiplies a specified array or part of an array by a constant. The constant can be real or integer depending on the array type; however, the arrays that can be operated on is dependent on which section the keyword is being used.

No.	Name	Description	Default
1	ARRAY	The name of the array to be modified. This is the keyword name identifying the property and is up to eight characters in length and enclosed in quotes.	None
2	CONSTANT	An integer or real value to multiply the ARRAY by in the same units as the ARRAY property.	None
3	I1	A positive integer that defines the lower bound of the array in the I-direction to be modified must be greater than or equal to one and less than or equal to I2 and NX.	1
4	I2	A positive integer that defines the upper bound of the array in the I-direction to be modified must be greater than or equal to I1 and less than or equal to NX	NX
5	J1	A positive integer that defines the lower bound of the array in the J-direction to be modified must be greater than or equal to one and less than or equal to J2 and NY.	1
6	J2	A positive integer that defines the upper bound of the array in the J-direction to be modified must be greater than or equal to J1 and less than or equal to NY.	NY
7	K1	A positive integer that defines the lower bound of the array in the K-direction to be modified must be greater than or equal to one and less than or equal to K2 and NZ.	1
8	K2	A positive integer that defines the upper bound of the array in the K-direction to be modified must be greater than or equal to K1 and less than or equal to NZ.	NZ

Notes:

1) Where NX, NY and NZ are the dimensions of the model as defined on the DIMENS keyword in the RUNSPEC section.

2) Each record must be terminated by a "/" and the keyword is terminated by a "/".

Table 6.69: MULTIPLY Keyword Description

The applicable arrays for each section are defined in Table 6.70 as shown on the next page.

MULTIPLY Keyword And Variable Options By Section						
GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
DX	DEPTH	SWL	ENDNUM	PRESSURE		
DY	PORV	SWCR	EQLNUM	SWAT		
DZ	TRANX	SWU	FIPNUM	SGAS		
PERMX	TRANY	SGL	IMBNUM	RV		
PERMY	TRANZ	SGCR	MISCNUM	RS		
PERMZ	DIFFX	SGU	PVTNUM	TBLK		
MULTX	DIFFY	KRW	ROCKNUM	GI		
MULTY	DIFFZ	KRO	SATNUM	OILAPI		
MULTZ	TRANR	KRG	WH2NUM	SALT		
DR	TRANTHT	PCG		GASCONC		
THETA	DIFFR	PCW		SOLVCONC		
PERMR	DIFFTHT			SOLVFRAC		
PERMTHT				SFOAM		
DZNET				SPOLY		
PORO						
NTG						
FLUXNUM						
MULTNUM						
MPANUM						
DIFFX						
DIFFY						
DIFFZ						
DIFFR						
DIFFTHT						

Table 6.70: MULTIPLY Keyword Applicable Arrays by Section

Example

```
--
--        ARRAY       CONSTANT      ---------- BOX ---------
--                                  I1  I2   J1  J2   K1  K2
MULTIPLY
          PERMZ       0.50000       1*  1*   1*  1*   1*  1* / PERMZ * 0.5
/
```

The above example multiples the PERMZ property array by 0.5 throughout the model.

6.3.129 MULTIREG – MULTIPLY AN ARRAY BY A CONSTANT BASED ON A REGION NUMBER

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The MULTIREG keyword multiplies an array or part of an array by a constant for cells with a specific region number. The region number array can be FLUXNUM, MULTNUM or OPERNUM and these arrays must be defined and be available before the MULTIREG keyword is read by the simulator. The constant can be real or integer depending on the property array type; however, the property arrays that can be operated on is dependent on which section the MULTIREG keyword is being used.

No.	Name	Description	Default
1	ARRAY	The name of the array to be modified. This is the keyword name identifying the property and is up to eight characters in length and enclosed in quotes.	None
2	CONSTANT	An integer or real value to multiply the ARRAY by in the same units as the ARRAY property for a given REGION.	0
3	REGION	REGION is a positive integer representing the region for which the CONSTANT in (2) should be applied	None
4	ARRAY	The ARRAY to use for applying the CONSTANT in (2) based on the REGION in (3). ARRAY can have the following values: 1) F for the FLUXNUM array 2) M for the MULTNUM array 3) O for the OPERNUM array	M
Notes: 1) Where the REGION should be less than or equal to the maximum number of regions as defined on the REGDIMS keyword for the FIPNUM and OPERNUM arrays or the GRIDOPTS keyword for the MULTNUM array in the RUNSPEC section. 2) Each record must be terminated by a "/" and the keyword is terminated by a "/".			

Table 6.71: MULTIREG Keyword Description

The applicable arrays for each section are defined in Table 6.72.

MULTREG Keyword And Variable Options By Section						
GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
DX	DEPTH	SWL	ENDNUM	PRESSURE		
DY	PORV	SWCR	EQLNUM	SWAT		
DZ	TRANX	SWU	FIPNUM	SGAS		
PERMX	TRANY	SGL	IMBNUM	RV		
PERMY	TRANZ	SGCR	MISCNUM	RS		
PERMZ	DIFFX	SGU	PVTNUM	TBLK		
MULTX	DIFFY	KRW	ROCKNUM	GI		
MULTY	DIFFZ	KRO	SATNUM	OILAPI		
MULTZ	TRANR	KRG	WH2NUM	SALT		

MULTREG Keyword And Variable Options By Section						
GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
DR	TRANTHT	PCG		GASCONC		
THETA	DIFFR	PCW		SOLVCONC		
PERMR	DIFFTHT			SOLVFRAC		
PERMTHT				SFOAM		
DZNET				SPOLY		
PORO						
NTG						
FLUXNUM						
MULTNUM						
MPANUM						
DIFFX						
DIFFY						
DIFFZ						
DIFFR						
DIFFTHT						

Table 6.72: MULTIREG Keyword Applicable Arrays by Section

Example

```
--
-- FIRST DEFINE THE PROPERTY ARRAYS AND MULTNUM ARRAYS FOR 10 X 10 X 20 MODEL
--
--  ARRAY     CONSTANT      ---------- BOX ---------
--                           I1  I2   J1  J2   K1  K2
EQUALS
    PORO       0.2000       1*  1*   1*  1*   1*  1*  / PORO  TO 0.20 IN MODEL
    PERMX      100.00       1*  1*   1*  1*   1*  1*  / PERMX TO 0.10 IN MODEL
    MULTNUM    1            1*  1*   1*  1*   1*  1*  / MULTNUM IN MODEL
    MULTNUM    2            1*  5    1   5    6   6   / MULTNUM IN MODEL
    MULTNUM    3            1*  1*   1*  1*   10  10  / MULTNUM IN MODEL
/
--
-- NOW RESET PORO AND PERMX BASED ON THE MULTNUM REGION NUMBER
--
-- MULTIPLY AN ARRAY BY A CONSTANT BASED ON A REGION NUMBER
--
--    ARRAY     CONSTANT  REGION   REGION ARRAY
--              VALUE     NUMBER    M / F / O
MULTIREG
      PORO      1.050     1         M               /
      PORO      1.100     2         M               /
      PORO      0.950     3         M               /
      PERMX      1.25     1         M               /
      PERMX      1.30     2         M               /
      PERMX      0.90     3         M               /
/
```

The example first defines the PORO and PERMX property arrays for the model and then sets the MULTNUM array to 1 for all cells in the model, after which selected areas of model are assigned various MULTNUM integer values. The MULTIREG can then be invoked to multiple the PORO and PERMX arrays by a constant for the various MULTNUM regions.

6.3.130 MULTNUM – DEFINE THE MULTIPLE TRANSMISSIBILITY REGIONS

Description

The MULTNUM keyword defines the inter-region transmissibility region numbers for each grid block, as such there must be one entry for each cell in the model. The array can be used with the EQUALREG, ADDREG, COPYREG, MULTIREG, MULTREGP and MULTREGT keywords in calculating various grid properties in the GRID section.

No.	Name	Description	Default
1	MULTNUM	MULTNUM defines an array of positive integers assigning a grid cell to a particular inter-region transmissibility region. The maximum number of MULTNUM regions is set by the NRMULT variable on the GRIDOPTS keyword in the RUNSPEC section.	1

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) The keyword is terminated by a "/".

Table 6.73: MULTNUM Keyword Description

Examples

The example below sets three MULTNUM regions for a 4 x 5 x 2 model.

```
--
--        DEFINE MULTNUM REGIONS FOR ALL CELLS
--
MULTNUM
          2  2  1  1  2  2  1  1  1  1  1  1  1  1  1  1  1  1  1  1
          3  3  1  1  3  3  1  1  1  1  1  1  1  1  1  1  1  1  1  1
/
```

Alternatively the EQUALS keyword could be employed to accomplish the same task, that is:

```
--
--        ARRAY        CONSTANT      ---------- BOX ---------
--                                    I1  I2    J1  J2    K1  K2
EQUALS
          MULTNUM      1             1*  1*    1*  1*    1*  1* / SET REGION 1
          MULTNUM      2             1   2     1   2     1   1  / SET REGION 2
          MULTNUM      3             1   2     1   2     2   2  / SET REGION 3
/
```

One can then increase PERMX by 25% in region three only.

```
--
--        MULTIPLY AN ARRAY BY A CONSTANT BASED ON A REGION NUMBER
--
--        ARRAY      CONSTANT  REGION   REGION ARRAY
--                   VALUE     NUMBER    M / F / O
MULTIREG
          PERMX      1.25      3         M                    /
/
```

6.3.131 MULTPV – MULTIPLY CELL PORE VOLUMES BY A CONSTANT

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

MULTPV multiples the pore volumes of a cell by a real positive constant for all the cells in the model via an array. An alternative to defining the complete array is to use the BOX keyword to define an area of the grid and then use the MULTPV keyword to set the multipliers just for the area defined by the BOX keyword (see the example).

The keyword can be used for all grid types, except for the Radial Grid geometry.

No.	Name	Description	Default
1	MULTPV	MULTPV is an array of real positive numbers assigning the pore volume multipliers for each cell in the model. Repeat counts may be used, for example 20*100.0.	1.0

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.

2) The keyword is terminated by a "/".

Table 6.74: MULTPV Keyword Description

See also the MULTREGP keyword for scaling the cell pore volumes by region numbers.

Example

```
--
--        DEFINE INPUT BOX FOR EDITING INPUT ARRAYS
--
--        ---------- BOX ---------
--        I1  I2   J1  J2   K1  K2
BOX
          10  10   1   6    1   3                          / DEFINE BOX AREA
--
--        SET MULTX+ TRANSMISSIBILITY MULTIPLIERS
--
MULTPV
          18*0.0500                                       /
--
--        DEFINE END OF INPUT BOX EDITING OF INPUT ARRAYS
--
ENDBOX
```

The above example defines a 0.05 scaling multiplier for the 18 cells defined by the preceding BOX statement. The ENDBOX keyword resets the input box to the full grid.

6.3.132 MULTR - MULTIPLY CELL TRANSMISSIBILITY IN THE +R DIRECTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

MULTR multiples the transmissibility between two cell faces in the +R direction for all the cells in the model via an array, that is the keyword sets the transmissibility multiplier of block (I, J, K) between the cells (I, J, K) and (I+1, J, K). An alternative to defining the complete array is to use the BOX keyword to define an area of the grid and then use the MULTR keyword to set the multipliers just for the area defined by the BOX keyword (see the example).

The keyword should only be used with radial and spider grids, as declared by the RADIAL or SPIDER keywords in the RUNSPEC section.

No.	Name	Description	Default
1	MULTR+	MULTR+ is an array of real positive numbers assigning the transmissibility multipliers in the +R direction to each cell face in the model. Repeat counts may be used, for example 20*100.0.	1.0

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) The keyword is terminated by a "/".

Table 6.75: MULTR Keyword Description

See also the MULTR-, MULTTHT, MULTTHT-, MULTZ and MULTZ- keywords for scaling transmissible between between grid cells in the R direction.

Example

```
--
--       DEFINE INPUT BOX FOR EDITING INPUT ARRAYS
--
--       ---------- BOX ---------
--       I1  I2   J1  J2   K1  K2
BOX
         10  10   1   6    1   3                          / DEFINE BOX AREA
--
--       SET MULTR+ TRANSMISSIBILITY MULTIPLIERS
--
MULTR
         18*0.300                                        /
--
--       DEFINE END OF INPUT BOX EDITING OF INPUT ARRAYS
--
ENDBOX
```

The above example defines a 0.3 scaling multiplier for the 18 cells defined by the preceding BOX statement. The ENDBOX keyword resets the input box to the full grid.

6.3.133 MULTR- - MULTIPLY CELL TRANSMISSIBILITY IN THE -R DIRECTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

MULTR- multiples the transmissibility between two cell faces in the -R direction for all the cells in the model via an array, that is the keyword sets the transmissibility multiplier of block (I, J, K) between the cells (I-1, J, K) and (I, J, K). An alternative to defining the complete array is to use the BOX keyword to define an area of the grid and then use the MULTR- keyword to set the multipliers just for the area defined by the BOX keyword (see the example).

The keyword should only be used with radial and spider grids, as declared by the RADIAL or SPIDER keywords in the RUNSPEC section.

No.	Name	Description	Default
1	MULTR-	MULTR- is an array of real positive numbers assigning the transmissibility multipliers in the -R direction to each cell face in the model. Repeat counts may be used, for example 20*100.0.	1.0

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.

2) The keyword is terminated by a "/".

Table 6.76: MULTR- Keyword Description

Note that OPM Flow does not require the GRIDOPTS(TRANMULT) parameter in the RUNSPEC section to be set to YES, in order to use this and other negative directional dependent multiplier keywords in the input deck. Whereas, the commercial simulator will terminate with an error if the keyword is present, and the GRIDOPTS(TRANMULT) parameter has not been set to YES.

See also the MULTR, MULTTHT, MULTTHT-, MULTZ and MULTZ- keywords for scaling transmissible between cells in the R direction.

Example

```
--
--       DEFINE INPUT BOX FOR EDITING INPUT ARRAYS
--
--       ---------- BOX ---------
--       I1  I2   J1  J2   K1  K2
BOX
         10  10   1   6    1   1                          / DEFINE BOX AREA
--
--       SET MULTX TRANSMISSIBILITY MULTIPLIERS CELLS
--
MULTR-
         6*0.500                                         /
--
--       DEFINE END OF INPUT BOX EDITING OF INPUT ARRAYS
--
ENDBOX
```

The above example defines a 0.5 scaling multiplier for the six cells defined by the preceding BOX statement. The ENDBOX keyword resets the input box to the full grid.

6.3.134 MULTREGD – MULTIPLY DIFFUSIVITIES BETWEEN REGIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The MULTREGT keyword multiplies the diffusivity between two regions by a constant. The region number array can be FLUXNUM, MULTNUM or OPERNUM and these arrays must be defined and be available before the MULTREGT keyword is read by the simulator. The constant should be a real number.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description	Default
1	REGION1	A positive integer value that defines the from REGION number for which the CONSTANT in (2) should be applied.	None
2	REGION2	A positive integer value that defines the to REGION number for which the CONSTANT in (2) should be applied.	None
3	CONSTANT	A real value to multiply the diffusivity between REGION1 and REGION2.	1
4	DIR	A character string that defines the direction to apply the diffusivity multiplier between the two regions, should be set to one of the following X, Y, Z, XY, YX, XZ, or XYZ.	XYZ
5	TYPE	A character string that defines the type of connections the diffusivity multiplier should be applied to, should be one of the following: 1) NNC – Only apply the diffusivity multiplier between REGION1 and REGION2 to non-neighbor connections. 2) NONNC – Do not apply the diffusivity multiplier between REGION1 and REGION2 to non-neighbor connections. 3) ALL - Apply the diffusivity multiplier between REGION1 and REGION2 to all connections.	ALL
6	ARRAY	The ARRAY to use for applying the CONSTANT in (2) based on the ARRAY in (1). ARRAY can have the following values: 1) F for the FLUXNUM array 2) M for the MULTNUM array 3) O for the OPERNUM array	M

Notes:

1) Where REGION1 and REGION2 should be less than or equal to the maximum number of regions as defined on the REGDIMS keyword for the FIPNUM and OPERNUM arrays or the GRIDOPTS keyword for the MULTNUM array in the RUNSPEC section.
2) Each record must be terminated by a "/" and the keyword is terminated by a "/".

Table 6.77: MULTREGD Keyword Description

Example

```
--
--    MULTIPLY DIFFUSIVITIES BETWEEN RESERVOIRS
--
--    REGION   REGION   DIFFS   DIREC   NNC    REGION ARRAY
--    FROM     TO       MULT    OPT     OPTS   M / F / O
MULTREGD
      1*       1*       1.05    1*     'ALL'   M           / ALL REGIONS
/
```

The above example multiplies the thermal conductivities between all the MULTNUM regions by 1.05 in all directions and for all connections types.

6.3.135 MULTREGH – MULTIPLY THERMAL CONDUCTIVITIES BETWEEN REGIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The MULTREGH keyword multiplies the thermal conductivity between two regions by a constant. The region number array can be FLUXNUM, MULTNUM or OPERNUM and these arrays must be defined and be available before the MULTREGT keyword is read by the simulator. The constant should be a real number.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description	Default
1	REGION1	A positive integer value that defines the from REGION number for which the CONSTANT in (2) should be applied.	None
2	REGION2	A positive integer value that defines the to REGION number for which the CONSTANT in (2) should be applied.	None
3	CONSTANT	A real value to multiply the thermal conductivity between REGION1 and REGION2.	1
4	DIR	A character string that defines the direction to apply the thermal conductivity multiplier between the two regions, should be set to one of the following X, Y, Z, XY, YX, XZ, or XYZ.	XYZ
5	TYPE	A character string that defines the type of connections the thermal conductivity multiplier should be applied to, should be one of the following: 1) NNC – Only apply the thermal conductivity multiplier between REGION1 and REGION2 to non-neighbor connections. 2) NONNC – Do not apply the thermal conductivity multiplier between REGION1 and REGION2 to non-neighbor connections. 3) ALL - Apply the thermal conductivity multiplier between REGION1 and REGION2 to all connections.	ALL
6	ARRAY	The ARRAY to use for applying the CONSTANT in (2) based on the ARRAY in (1). ARRAY can have the following values: 1) F for the FLUXNUM array. 2) M for the MULTNUM array. 3) O for the OPERNUM array.	M

Notes:

1) Where REGION1 and REGION2 should be less than or equal to the maximum number of regions as defined on the REGDIMS keyword for the FIPNUM and OPERNUM arrays or the GRIDOPTS keyword for the MULTNUM array in the RUNSPEC section.
2) Each record must be terminated by a "/" and the keyword is terminated by a "/".

Table 6.78: MULTREGH Keyword Description

Example

```
--
--    MULTIPLY THERMAL CONDUCTIVITIES BETWEEN RESERVOIRS
--
--    REGION   REGION   CONDS   DIREC   NNC    REGION ARRAY
--    FROM     TO       MULT    OPT     OPTS   M / F / O
MULTREGH
      1*       1*       1.05    1*     'ALL'   M           / ALL REGIONS
/
```

The above example multiplies the diffusivities between all the MULTNUM regions by 1.05 in all directions and for all connections types.

6.3.136 MULTREGP – MULTIPLY PORE VOLUMES BASED ON REGION NUMBER

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The MULTREGP keyword multiplies the pore volume of a cell by a constant for all cells with a specific region number. The region number array can be FLUXNUM, MULTNUM or OPERNUM and these arrays must be defined and be available before the MULTREGP keyword is read by the simulator. The constant should be a real number.

No.	Name	Description	Default
1	REGION	REGION is a positive integer representing the region for which the CONSTANT in (2) should be applied.	None
2	CONSTANT	A real value to multiply the pore volume by for a given REGION.	1
3	ARRAY	The ARRAY to use for applying the CONSTANT in (2) based on the REGION in (1). ARRAY can have the following values: 1) F for the FLUXNUM array. 2) M for the MULTNUM array. 3) O for the OPERNUM array.	M

Notes:

1) Where the REGION should be less than or equal to the maximum number of regions as defined on the REGDIMS keyword for the FIPNUM and OPERNUM arrays or the GRIDOPTS keyword for the MULTNUM array in the RUNSPEC section.
2) Each record must be terminated by a "/" and the keyword is terminated by a "/".

Table 6.79: MULTREGP Keyword Description

Example

```
--
-- RESET PORE VOLUME FOR DIFFERENT REGIONS
--
--    REGION    PORV          REGION ARRAY
--    NUMBER    MULT          M / F / O
MULTREGP
         1       1.0456573    M             /  Fault Block 1
         2       0            M             /  Fault Block 2
         3       0.9756715    M             /  Fault Block 3
         4       0            M             /  Inactive Blocks
/
```

The above example re-scales the pore volumes for MULTNUM regions one and three and makes regions two and four inactive by setting their pore volumes to zero.

6.3.137 MULTREGT – MULTIPLY TRANSMISSIBILITIES BETWEEN REGIONS

Description

The MULTREGT keyword multiplies the transmissibility between two regions by a constant. The region number array can be FLUXNUM, MULTNUM or OPERNUM and these arrays must be defined and be available before the MULTREGT keyword is read by the simulator. The constant should be a real number.

No.	Name	Description	Default
1	REGION1	A positive integer value that defines the from REGION number for which the CONSTANT in (2) should be applied.	None
2	REGION2	A positive integer value that defines the to REGION number for which the CONSTANT in (2) should be applied.	None
3	CONSTANT	A real positive value to multiply the transmissibility between REGION1 and REGION2.	1
4	DIR	A character string that defines the direction to apply the transmissibility multiplier between the two regions, should be set to one of the following X, Y, Z, XY, YX, XZ, or XYZ.	XYZ
5	TYPE	A character string that defines the type of connections the transmissibility multiplier should be applied to, should be one of the following: 1) NNC – Only apply the transmissibility multiplier between REGION1 and REGION2 to non-neighbor connections. 2) NONNC – Do not apply the transmissibility multiplier between REGION1 and REGION2 to non-neighbor connections. 3) ALL - Apply the transmissibility multiplier between REGION1 and REGION2 to all connections.	ALL
6	ARRAY	The ARRAY to use for applying the CONSTANT in (2) based on the ARRAY in (1). ARRAY can have the following values: 1) F for the FLUXNUM array 2) M for the MULTNUM array 3) O for the OPERNUM array	M

Notes:

1) Where REGION1 and REGION2 should be less than or equal to the maximum number of regions as defined on the REGDIMS keyword for the FIPNUM and OPERNUM arrays or the GRIDOPTS keyword for the MULTNUM array in the RUNSPEC section.
2) Each record must be terminated by a "/" and the keyword is terminated by a "/".

Table 6.80: MULTREGT Keyword Description

Note

Note if the MULTREGT keyword is used in the EDIT section, OPM Flow will always apply the changes irrespective, of if the TRANX, TRANY and TRANZ transmissibility arrays have been entered or not in the EDIT section.

This behavior is different to the commercial simulator that only applies the keyword if the transmissibility arrays have been entered in the EDIT section.

Example

```
--
--    SET TRANSMISSIBILITES ACROSS DIFFERENT RESERVOIRS TO ZERO
--
--    REGION   REGION   TRANS   DIREC   NNC    REGION ARRAY
--    FROM     TO       MULT    OPT     OPTS   M / F / O
MULTREGT
      1*       1*       0.0     1*     'ALL'   M           / ALL REGIONS SEALED
/
```

The above example isolates all regions from one another by setting the transmissibility for the MULTNUM regions to zero in all directions and for all connections types.

6.3.138 MULTTHT - MULTIPLY CELL TRANSMISSIBILITY IN THE +THETA DIRECTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

MULTTHT multiples the transmissibility between two cell faces in the +Theta direction for all the cells in the model via an array, that is the keyword sets the transmissibility multiplier of block (I, J, K) between the cells (I, J, K) and (I, J+1, K). An alternative to defining the complete array is to use the BOX keyword to define an area of the grid and then use the MULTTHT keyword to set the multipliers just for the area defined by the BOX keyword (see the example).

The keyword should only be used with radial and spider grids, as declared by the RADIAL or SPIDER keywords in the RUNSPEC section.

No.	Name	Description	Default
1	MULTTHT+	MULTTHT+ is an array of real positive numbers assigning the transmissibility multipliers in the +Theta direction to each cell face in the model. Repeat counts may be used, for example 20*100.0.	1.0

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.

2) The keyword is terminated by a "/".

Table 6.81: MULTTHT Keyword Description

See also the MULTTHT-, MULTR, MULTR-, MULTZ and MULTZ- keywords for scaling transmissible between I grid cells in the Theta direction.

Example

```
--
--       DEFINE INPUT BOX FOR EDITING INPUT ARRAYS
--
--       ---------- BOX ---------
--       I1  I2   J1  J2   K1  K2
BOX
         10  10   1   6    1   3                          / DEFINE BOX AREA
--
--       SET MULTTHT+ TRANSMISSIBILITY MULTIPLIERS
--
MULTTHT
         18*0.300                                        /
--
--       DEFINE END OF INPUT BOX EDITING OF INPUT ARRAYS
--
ENDBOX
```

The above example defines a 0.3 scaling multiplier for the 18 cells defined by the preceding BOX statement. The ENDBOX keyword resets the input box to the full grid.

6.3.139 MULTTHT- - MULTIPLY CELL TRANSMISSIBILITY IN THE -THETA DIRECTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

MULTTHT- multiples the transmissibility between two cell faces in the -Theta direction for all the cells in the model via an array, that is the keyword sets the transmissibility multiplier of block (I, J, K) between the cells (I, J-1, K) and (I, J, K). An alternative to defining the complete array is to use the BOX keyword to define an area of the grid and then use the MULTTHT- keyword to set the multipliers just for the area defined by the BOX keyword (see the example).

The keyword should only be used with radial and spider grids, as declared by the RADIAL or SPIDER keywords in the RUNSPEC section.

No.	Name	Description	Default
1	MULTTHT-	MULTTHT- is an array of real positive numbers assigning the transmissibility multipliers in the -Theta direction to each cell face in the model. Repeat counts may be used, for example 20*100.0.	1.0

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.

2) The keyword is terminated by a "/".

Table 6.82: MULTTHT- Keyword Description

Note that OPM Flow does not require the GRIDOPTS(TRANMULT) parameter in the RUNSPEC section to be set to YES, in order to use this and other negative directional dependent multiplier keywords in the input deck. Whereas, the commercial simulator will terminate with an error if the keyword is present, and the GRIDOPTS(TRANMULT) parameter has not been set to YES.

See also the MULTTHT, MULTR, MULTR-, MULTZ and MULTZ- keywords for scaling transmissible between I grid cells in the Theta direction.

Example

```
--
--       DEFINE INPUT BOX FOR EDITING INPUT ARRAYS
--
--       ---------- BOX ---------
--       I1  I2   J1  J2   K1  K2
BOX
         10  10   1   6    1   1                          / DEFINE BOX AREA
--
--       SET MULTX TRANSMISSIBILITY MULTIPLIERS CELLS
--
MULTTHT-
         6*0.500                                         /
--
--       DEFINE END OF INPUT BOX EDITING OF INPUT ARRAYS
--
ENDBOX
```

The above example defines a 0.5 scaling multiplier for the six cells defined by the preceding BOX statement. The ENDBOX keyword resets the input box to the full grid.

6.3.140 MULTX - MULTIPLY CELL TRANSMISSIBILITY IN THE +X DIRECTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

MULTX multiples the transmissibility between two cell faces in the +X direction for all the cells in the model via an array, that is the keyword sets the transmissibility multiplier of block (I, J, K) between the cells (I, J, K) and (I+I, J, K).

An alternative to defining the complete array is to use the BOX keyword to define an area of the grid and then use the MULTX keyword to set the multipliers just for the area defined by the BOX keyword (see the example).

The keyword can be used for all grid types, except for the Radial Grid geometry.

No.	Name	Description	Default
1	MULTX+	MULTX+ is an array of real positive numbers assigning the transmissibility multipliers in the +X direction to each cell face in the model. Repeat counts may be used, for example 20*100.0.	1.0

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) The keyword is terminated by a "/".

Table 6.83: MULTX Keyword Description

See also the MULTX-, MULTY, MULTY-, MULTZ and MULTZ- keywords for scaling transmissible between grid cells.

Example

```
--
--       DEFINE INPUT BOX FOR EDITING INPUT ARRAYS
--
--       ---------- BOX ---------
--       I1  I2   J1  J2   K1  K2
BOX
         10  10   1   6    1   3                          / DEFINE BOX AREA
--
--       SET MULTX+ TRANSMISSIBILITY MULTIPLIERS
--
MULTX
         18*0.300                                        /
--
--       DEFINE END OF INPUT BOX EDITING OF INPUT ARRAYS
--
ENDBOX
```

The above example defines a 0.3 scaling multiplier for the 18 cells defined by the preceding BOX statement. The ENDBOX keyword resets the input box to the full grid.

6.3.141 MULTX- - MULTIPLY CELL TRANSMISSIBILITY IN THE -X DIRECTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

MULTX- multiples the transmissibility between two cell faces in the -X direction for all the cells in the model via an array, that is the keyword sets the transmissibility multiplier of block (I, J, K) between the cells (I-1, J, K) and (I, J, K). An alternative to defining the complete array is to use the BOX keyword to define an area of the grid and then use the MULTX- keyword to set the multipliers just for the area defined by the BOX keyword (see the example).

The keyword can be used for all grid types, except for the Radial Grid geometry.

No.	Name	Description	Default
1	MULTX-	MULTX- is an array of real positive numbers assigning the transmissibility multipliers in the -X direction to each cell face in the model. Repeat counts may be used, for example 20*100.0.	1.0

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.

2) The keyword is terminated by a "/".

Table 6.84: MULTX- Keyword Description

Note that OPM Flow does not require the GRIDOPTS(TRANMULT) parameter in the RUNSPEC section to be set to YES, in order to use this and other negative directional dependent multiplier keywords in the input deck. Whereas, the commercial simulator will terminate with an error if the keyword is present, and the GRIDOPTS(TRANMULT) parameter has not been set to YES.

See also the MULTX, MULTY, MULTY-, MULTZ and MULTZ- keywords for scaling transmissible between grid cells.

Example

```
--
--       DEFINE INPUT BOX FOR EDITING INPUT ARRAYS
--
--       ---------- BOX ---------
--       I1  I2   J1  J2   K1  K2
BOX
         10  10   1   6    1   1                          / DEFINE BOX AREA
--
--       SET MULTX TRANSMISSIBILITY MULTIPLIERS CELLS
--
MULTX-
         6*0.500                                         /
--
--       DEFINE END OF INPUT BOX EDITING OF INPUT ARRAYS
--
ENDBOX
```

The above example defines a 0.5 scaling multiplier for the six cells defined by the preceding BOX statement. The ENDBOX keyword resets the input box to the full grid.

6.3.142 MULTY - MULTIPLY CELL TRANSMISSIBILITY IN THE +Y DIRECTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

MULTY multiples the transmissibility between two cell faces in the +Y direction for all the cells in the model via an array, that is the keyword sets the transmissibility multiplier of block (I, J, K) between the cells (I, J, K) and (I, J+1, K).

An alternative to defining the complete array is to use the BOX keyword to define an area of the grid and then use the MULTY keyword to set the multipliers just for the area defined by the BOX keyword (see the example).

The keyword can be used for all grid types, except for the Radial Grid geometry.

No.	Name	Description	Default
1	MULTY+	MULTY+ is an array of real positive numbers assigning the transmissibility multipliers in the +Y direction to each cell face in the model. Repeat counts may be used, for example 20*100.0.	1.0

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) The keyword is terminated by a "/".

Table 6.85: MULTY Keyword Description

See also the MULTY-, MULTX, MULTX-, MULTZ and MULTZ- keywords for scaling transmissible between grid cells.

Example

```
--
--       DEFINE INPUT BOX FOR EDITING INPUT ARRAYS
--
--       ---------- BOX ---------
--       I1  I2   J1  J2   K1  K2
BOX
         10  10   1   6    1   3                          / DEFINE BOX AREA
--
--       SET MULTX+ TRANSMISSIBILITY MULTIPLIERS
--
MULTY
         18*0.300                                        /
--
--       DEFINE END OF INPUT BOX EDITING OF INPUT ARRAYS
--
ENDBOX
```

The above example defines a 0.3 scaling multiplier for the 18 cells defined by the preceding BOX statement. The ENDBOX keyword resets the input box to the full grid.

6.3.143 MULTY- - MULTIPLY CELL TRANSMISSIBILITY IN THE -Y DIRECTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

MULTY- multiples the transmissibility between two cell faces in the -Y direction for all the cells in the model via an array, that is the keyword sets the transmissibility multiplier of block (I, J, K) between the cells (I, J-1, K) and (I, J, K). An alternative to defining the complete array is to use the BOX keyword to define an area of the grid and then use the MULTY- keyword to set the multipliers just for the area defined by the BOX keyword (see the example).

The keyword can be used for all grid types, except for the Radial Grid geometry.

No.	Name	Description	Default
1	MULTY-	MULTY- is an array of real positive numbers assigning the transmissibility multipliers in the -Y direction to each cell face in the model. Repeat counts may be used, for example 20*100.0.	1.0

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.

2) The keyword is terminated by a "/".

Table 6.86: MULTY- Keyword Description

Note that OPM Flow does not require the GRIDOPTS(TRANMULT) parameter in the RUNSPEC section to be set to YES, in order to use this and other negative directional dependent multiplier keywords in the input deck. Whereas, the commercial simulator will terminate with an error if the keyword is present, and the GRIDOPTS(TRANMULT) parameter has not been set to YES.

See also the MULTY, MULTX, MULTX-, MULTZ and MULTZ- keywords for scaling transmissible between grid cells.

Example

```
--
--       DEFINE INPUT BOX FOR EDITING INPUT ARRAYS
--
--       ---------- BOX ---------
--       I1  I2   J1  J2   K1  K2
BOX
         10  10   1   6    1   1                          / DEFINE BOX AREA
--
--       SET MULTX TRANSMISSIBILITY MULTIPLIERS CELLS
--
MULTY-
         6*0.500                                         /
--
--       DEFINE END OF INPUT BOX EDITING OF INPUT ARRAYS
--
ENDBOX
```

The above example defines a 0.5 scaling multiplier for the six cells defined by the preceding BOX statement. The ENDBOX keyword resets the input box to the full grid.

6.3.144 MULTZ - MULTIPLY CELL TRANSMISSIBILITY IN THE +Z DIRECTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

MULTZ multiples the transmissibility between two cell faces in the +Z direction for all the cells in the model via an array, that is the keyword sets the transmissibility multiplier of block (I, J, K) between the cells (I, J, K) and (I, J, K+1).

An alternative to defining the complete array is to use the BOX keyword to define an area of the grid and then use the MULTZ keyword to set the multipliers just for the area defined by the BOX keyword (see the example).

The keyword can be used for all grid types, except for the Radial Grid geometry.

No.	Name	Description	Default
1	MULTZ+	MULTZ+ is an array of real positive numbers assigning the transmissibility multipliers in the +Z direction to each cell face in the model. Repeat counts may be used, for example 20*100.0.	1.0

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) The keyword is terminated by a "/".

Table 6.87: MULTZ Keyword Description

See also the MULTZ-, MULTX, MULTX-, MULTY and MULTY- keywords for scaling transmissible between grid cells.

Example

```
--
--       DEFINE INPUT BOX FOR EDITING INPUT ARRAYS
--
--       ---------- BOX ---------
--       I1  I2   J1  J2   K1  K2
BOX
         10  10   1   18   1   1                          / DEFINE BOX AREA
--
--       SET MULTX+ TRANSMISSIBILITY MULTIPLIERS
--
MULTZ
         18*0.300                                        /
--
--       DEFINE END OF INPUT BOX EDITING OF INPUT ARRAYS
--
ENDBOX
```

The above example defines a 0.3 scaling multiplier for the 18 cells defined by the preceding BOX statement. The ENDBOX keyword resets the input box to the full grid.

6.3.145 MULTZ- - MULTIPLY CELL TRANSMISSIBILITY IN THE -Z DIRECTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

MULTZ- multiples the transmissibility between two cell faces in the -Z direction for all the cells in the model via an array, that is the keyword sets the transmissibility multiplier of block (I, J, K) between the cells (I, J, K-1) and (I, J, K). An alternative to defining the complete array is to use the BOX keyword to define an area of the grid and then use the MULTZ- keyword to set the multipliers just for the area defined by the BOX keyword (see the example).

The keyword can be used for all grid types, except for the Radial Grid geometry.

No.	Name	Description	Default
1	MULTZ-	MULTZ- is an array of real positive numbers assigning the transmissibility multipliers in the -X direction to each cell face in the model. Repeat counts may be used, for example 20*100.0.	1.0

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) The keyword is terminated by a "/".

Table 6.88: MULTZ- Keyword Description

Note that OPM Flow does not require the GRIDOPTS(TRANMULT) parameter in the RUNSPEC section to be set to YES, in order to use this and other negative directional dependent multiplier keywords in the input deck. Whereas, the commercial simulator will terminate with an error if the keyword is present, and the GRIDOPTS(TRANMULT) parameter has not been set to YES.

See also the MULTZ, MULTX, MULTX-, MULTY and MULTY- keywords for scaling transmissible between grid cells.

Example

```
--
--       DEFINE INPUT BOX FOR EDITING INPUT ARRAYS
--
--       ---------- BOX ---------
--       I1  I2   J1  J2   K1  K2
BOX
         10  10   1   6    1   1                          / DEFINE BOX AREA
--
--       SET MULTX TRANSMISSIBILITY MULTIPLIERS CELLS
--
MULTZ-
         6*0.500                                         /
--
--       DEFINE END OF INPUT BOX EDITING OF INPUT ARRAYS
--
ENDBOX
```

The above example defines a 0.5 scaling multiplier for the six cells defined by the preceding BOX statement. The ENDBOX keyword resets the input box to the full grid.

6.3.146 NEWTRAN – ACTIVATE IRREGULAR CORNER-POINT GRID TRANSMISSIBILITIES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword switches on Irregular Corner-Point Grid geometry transmissibility calculation, which is the default option for this type of grid. Grids defined with the COORD and ZCORN keywords will always invoke this option by default.

For Cartesian Regular Grids defined by the DX, DY, and DZ series of keywords the block center geometry transmissibility calculations should be activated via the OLDTRAN keyword. Again this is automatically invoked if this type of grid is being employed.

There is no data required for this keyword and there is no terminating "/" for this keyword.

Example

```
--
--       ACTIVATE IRREGULAR CORNER-POINT GRID TRANSMISSIBILITIES
--
NEWTRAN
```

The above example manually activates Irregular Corner-Point Grid transmissibility calculations.

6.3.147 NINENUM – DEFINE THE NINE-POINT DISCRETIZATION REGION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The NINENUM keyword defines areas in the grid that should use the Nine-Point Discretization formulation by setting a grid block's NINENUM value to one, or zero for the conventional standard five-point discretization formulation, for when the Nine-Point Discretization formulation has been activated by the NINEPOIN keyword in the RUNSPEC section. There should be a NINENUM value for each grid block in the model. Note that if the if the NINEPOIN keyword in the RUNSPEC section has been invoked and the NINENUM keyword has not been used in the input deck, then all the grid will use the nine-point scheme.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description	Default
1	NINENUM	NINENUM defines an integer array of zeros and ones assigning a grid cell to a particular discretization region, a value of zero for five-point or a value of one for nine-point discretization. Note that the default value of one implies a cell is included in the Nine-Point Discretization region; thus, if a cell is to use the conventional standard five-point finite difference discretization formulation, then NINENUM must be explicitly set to zero.	1

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.

2) The keyword is terminated by a "/".

Table 6.89: NINENUM Keyword Description

The NINENUM keyword cannot be used in models with Local Grid Refinements ("LGR") to set different discretization regions within the model, that is if LGRs are present in the model either all the grid uses nine-point discretization, if NINEPOIN is present in the RUNSPEC section, or five-point if NINEPOIN is absent.

Example

The example below sets a portion of the model to us the Nine-Point Discretization formulation.

```
--
--       DEFINE NINE-POINT DISCRETIZATION REGION FOR ALL CELLS
--
--       ARRAY       CONSTANT     ---------- BOX ---------
--                                I1  I2   J1  J2   K1  K2
EQUALS
         NINENUM'    0            1*  1*   1*  1*   1*  1* / FIVE-POINT
         NINENUM'    1            1*  1*   1*  1*   1   5  / NINE-POINT
/
```

Here the first line sets all the grid to us the five-point discretization formulation, all values set to zero, and then the second line sets all the cells in the layers one to five to use the nine-point discretization formulation.

6.3.148 NMATOPTS – DEFINE THE DISCRETIZED MATRIX DUAL POROSITY PARAMETERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The NMATOPTS keyword defines the Discretized Matrix Dual Porosity parameters for when the Discretized Matrix Dual Porosity option has been activated by NMATRIX keyword in the RUNSPEC section. The option allows the matrix grid blocks to be subdivided into smaller cells for more accurate flow calculations, in particular the modeling of transient flow within the matrix grid blocks.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

6.3.149 NNC – DEFINE NON-NEIGHBOR CONNECTIONS BETWEEN CELLS MANUALLY

Description

NNC enables Non-Neighbor Connections ("NNC") to be manually defined. This keyword is normally generated by static modeling software as opposed to be manually entered in the OPM Flow input deck due to the verbosity and complexity of calculating the required parameters for this keyword.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	I1	A positive integer that defines the first grid block in the I-direction in a non-neighbor connection, must be greater than or equal to one and less than or equal to NX on the DIMENS keyword in the RUNSPEC section.			None
2	J1	A positive integer that defines the first grid block in the J-direction in a non-neighbor connection, must be greater than or equal to one and less than or equal to NY on the DIMENS keyword in the RUNSPEC section.			None
3	K1	A positive integer that defines the first grid block in the K-direction in a non-neighbor connection, must be greater than or equal to one and less than or equal to NZ on the DIMENS keyword in the RUNSPEC section.			None
4	I2	A positive integer that defines the second grid block in the I-direction in a non-neighbor connection, must be greater than or equal to one and less than or equal to NX on the DIMENS keyword in the RUNSPEC section.			None
5	J2	A positive integer that defines the second grid block in the J-direction in a non-neighbor connection, must be greater than or equal to one and less than or equal to NY on the DIMENS keyword in the RUNSPEC section.			None
6	K2	A positive integer that defines the second grid block in the K-direction in a non-neighbor connection, must be greater than or equal to one and less than or equal to NZ on the DIMENS keyword in the RUNSPEC section.			None
7	TRANSNNC	TRANSNNC is a positive real number greater than or equal to zero that defines the transmissibility between the first grid block (I1, J1, K1) and the second grid block (I2, J2, K2). The default value of zero sets the transmissibility between the two cells to zero.			
		cP.rb/day/psia	cP.rm^3/day/bars	cP.rcc/hr/atm	0.0
8	ISATNUM1	ISATNUM1 is a positive integer defining which saturation table number (relative permeability table) to be used for flow from the first grid block to the second grid block. The default value of zero means the existing saturation table allocated to the upstream cell (I1,J1,K1).			0

No.	Name	Description			Default
		Field	Metric	Laboratory	
9	ISATNUM2	ISATNUM2 is a positive integer defining which saturation table number (relative permeability table) to be used for flow from the second grid block to the first grid block. The default value of zero means the existing saturation table allocated to the downstream cell (I2,J2,K2).			0
10	IPRSNUM1	IPRSNUM1 is a positive integer defining which pressure table number (PVT table) to be used for flow from the first grid block to the second grid block. The default value of zero means the existing PVT table allocated to the upstream cell (I1,J1,K1).			0
11	IPRSNUM2	IPRSNUM2 is a positive integer defining which pressure table number (PVT table) to be used for flow from the second grid block to the first grid block. The default value of zero means the existing PVT table allocated to the downstream cell (I2,J2,K2).			0
12	FACE1	FACE1 is a character string that defines the face associated with flow from the first grid block to the second grid block, where FACE1 can have values of: X+, X-, Y+, Y-, Z+, or Z-.			None
13	FACE2	FACE2 is a character string that defines the face associated with flow from the second grid block to the first grid block, where FACE2 can have values of: X+, X-, Y+, Y-, Z+, or Z-.			None
14	DIFFNNC	DIFFNNC is a positive real number that defines the diffusivity between the first grid block (I1, J1, K1) and the second grid block (I2, J2, K2).			
		feet	meters	cm	0.0
15	DISPNNC	DISPNNC s a positive real number that defines the dispersion coefficient $\frac{1}{(Area\ x\ Porosity)}$ between the first grid block (I1, J1, K1) and the second grid block (I2, J2, K2), used with the DISPERSE option.			
		ft^{-2}	m^{-2}	cm^{-2}	0.0
16	AREANNC	AREANNC is a positive real number that defines the area associated with the connection between the first grid block (I1, J1, K1) and the second grid block (I2, J2, K2).			
		ft^2	m^2	cm^2	None
17	PERMNNC	AREANNC is a positive real number that defines the permeability associated with the connection between the first grid block (I1, J1, K1) and the second grid block (I2, J2, K2). This used by the non-Darcy option.			
		mD	mD	mD	None

Notes:

1) Only functionality defined by items (1) to (7) are activated in OPM Flow.
2) Each record must be terminated by a "/" and the keyword is terminated by a "/".

Table 6.90: NNC Keyword Description

Note that although items (8) to (17) for this keyword are not available in OPM Flow, even if they were, it is strongly recommended that these items are defaulted if the data is being entered manually, as opposed to being generated by a pre-processing software.

Care should be taken that cells in different PVTNUM regions (see the PVTNUM keyword in the REGIONS section) are not connected, since the fluid properties are associated with a cell. If for example, a rbbl or a rm3 of oil flows from PVTNUM region 1 to PVTNUM region 2, then the oil properties of that oil will change from the PVT 1 data set to the PVT data set 2. This will result in material balance errors, that may or may not cause numerical issues.

If the transmissibility across a fault needs to be modified see the FAULTS and MULTFLT keywords in the GRID section for an alternative and less complicated method to modifying fault transmissibilities. Transmissibility between reservoir regions can be modified by using MULTREGT keyword, provided MULTNUM has been used to define the inter-region transmissibility region numbers for each grid block. Finally, the MULTX, MULTY and MULTZ series of keywords can be used to modify transmissibility between various cells. All the aforementioned keywords are described in the GRID section.

Example

```
--
--        MANUALLY DEFINE NON-NEIGHBOR CONNECTIONS
--
--        ------------ BOX -----------    -- TRANSNNC --
--        I1   J1   K1    I2   J2   K2
NNC
          1    1    1     1    1    2         0.2500     / SET NNC FOR FAULT
          1    1    2     1    1    3         0.2500     / SET NNC FOR FAULT
          1    1    3     1    1    4         0.2500     / SET NNC FOR FAULT
/
```

The above example defines the transmissibility between cells (1, 1, 1) and (1, 1, 2), (1, 1, 2) and (1, 1, 3) and finally between (1, 1, 3) and (1, 1, 4) to be 0.2500.

6.3.150 NODPPM – DEACTIVATE FRACTURE POROSITY-PERMEABILITY CALCULATION

The NODPPM keyword deactivates the default behavior of multiplying the fracture porosity by the fracture permeability to calculate the effective fracture permeability in dual porosity and dual permeability runs. Either the DUALPORO or DUALPERM keywords in the RUNSPEC section must be declared in the input file in order to use this keyword. If the default calculation is switched off by this keyword, then the effective fracture permeability is taken to be those entered for the fracture using the PERMX, PERMY and PERMZ keywords in the GRID section. If the keyword is absent from the input deck, then the entered PERMX, PERMY and PERMZ arrays for the fractures are multiplied by fracture PORO array values in order to obtain the effective fracture permeability.

See *NODPPM – Deactivate Fracture Porosity-Permeability Calculation* in the RUNSPEC section for a full description.

6.3.151 NOECHO – DEACTIVATE ECHOING OF USER INPUT FILES TO THE PRINT FILE

Turns off echoing of all the input files to the print file. Note by default echoing of the inputs files is active. but can subsequently be switched off by the NOECHO activation keyword.

See *NOECHO – Deactivate Echoing of User Input Files to the Print File* in the GLOBAL section for a full description.

6.3.152 NOGGF – DEACTIVATE OUTPUT OF GRID GEOMETRY FILE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword deactivates the output of a standard GRID or extended GRID file, as well as the extensible EGRID file for post-processing applications.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

There is no data required for this keyword and there is no terminating "/" for this keyword.

Example

```
--
--       DEACTIVATE GRID GEOMETRY OUTPUT
--
NOGGF
```

The above example switches off the default behavior of writing out the grid geometry files.

6.3.153 NOWARN – DEACTIVATE WARNING MESSAGES

Turns off warning messages to be printed to the print file; note that this keyword is deactivated by default and can subsequently be switched off by the WARN activation keyword. The warning messages may be turned on and off using keywords WARN and NOWARN.

See *NOWARN – Deactivate Warning Messages* in the GLOBAL section for a full description.

6.3.154 NTG – DEFINE THE NET-TO-GROSS RATIO FOR ALL THE CELLS

Description

NTG defines the Net-to-Gross Ratio ("NTG") for all the cells in the model via an array. The keyword can be used with all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	NTG	NTG is an array of real numbers greater than or equal to zero and less than or equal to one, that are assigned the net-to-gross ratio values for each cell in the model. The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword. Repeat counts may be used, for example 200*0.850.			
		dimensionless	dimensionless	dimensionless	None

Notes:

1) Setting a cells NTG value to zero will make the cell inactive, similar to setting the cells ACTNUM property to zero.
2) The keyword is terminated by a "/".

Table 6.91: NTG Keyword Description

See also the PORO, PERMX, PERMY and PERMZ keywords to fully define a grid's properties.

Example

```
--
--        DEFINE GRID BLOCK NTG DATA FOR ALL CELLS (BASED ON NX x NY x NZ = 300)
--
NTG
          100*1.000   100*0.850   100*0.500                                    /
```

The above example defines a constant NTG of 1.00 for the first 100 cells, then 0.85 for the second 100 hundred cells, and finally 0.500 for the last 100 cell, for the 300 cells in the model as defined by the DIMENS keyword in the RUNSPEC section.

6.3.155 NXFIN – DEFINE THE NUMBER OF LGR GRID BLOCKS IN THE X-DIRECTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

NXFIN defines the number of Local Grid Refinement ("LGR") cells within a global or host cell in the x-direction via a vector, as opposed to defining the size for each cell for a Cartesian LGR Grid. The LGR keyword in the RUNSPEC section should be activated to indicate an LGR is being used, and the keyword NXFIN should be placed in between the CARFIN and ENDFIN keywords in the GRID section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	NXFIN	NXFIN is a vector of integer numbers describing the number of LGR cells within each defined global or host grid block in the x-direction in a Cartesian LGR grid. Repeat counts may be used, for example 2*2.0.			
		dimensionless	dimensionless	dimensionless	None

Notes:

1) The number of entries should correspond to the NX parameter on the CARFIN keyword in the GRID section.
2) The keyword is terminated by a "/".

Table 6.92: NXFIN Keyword Description

See also the CARFIN, ENDFIN, NYFIN, and NZFIN keywords in the GRID section to fully define a Cartesian LGR grid model.

Example

```
--
--       CARFIN LGR GRID COMMANDS
--
--       LGR        ----- HOST GRID ------    -- CARFIN GRID --  MAX     HOST
--       NAME       I1  I2  J1  J2  K1  K2     NX    NY    NZ   WELLS   NAME
CARFIN
         LGR-OP01   24  25  87  87   1  50      8     1    50     1     GLOBAL /
--
--       DEFINE LGR GRID BLOCKS IN THE X-DIRECTION
--
NXFIN
         4  4                                                             /

ENDFIN
```

The above example splits the global cells (24-25,87, 1-50) into four and four LGR grid blocks in the x-direction, and since the HXFIN keyword has not been supplied, then the host cells will split into equal proportions.

6.3.156 NYFIN – DEFINE THE NUMBER OF LGR GRID BLOCKS IN THE Y-DIRECTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

NYFIN defines the number of Local Grid Refinement ("LGR") cells within a global or host cell in the x-direction via a vector, as opposed to defining the size for each cell for a Cartesian LGR Grid. The LGR keyword in the RUNSPEC section should be activated to indicate an LGR is being used, and the keyword NYFIN should be placed in between the CARFIN and ENDFIN keywords in the GRID section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	NYFIN	NYFIN is a vector of integer numbers describing the number of LGR cells within each defined global or host grid block in the y-direction in a Cartesian LGR grid. Repeat counts may be used, for example 2*2.0.			
		dimensionless	dimensionless	dimensionless	None

Notes:

1) The number of entries should correspond to the NY parameter on the CARFIN keyword in the GRID section.
2) The keyword is terminated by a "/".

Table 6.93: NYFIN Keyword Description

See also the CARFIN, ENDFIN, NXFIN, and NZFIN keywords in the GRID section to fully define a Cartesian LGR grid model.

Example

```
--
--       CARFIN LGR GRID COMMANDS
--
--       LGR        ----- HOST GRID ------    -- CARFIN GRID --   MAX     HOST
--       NAME       I1  I2  J1  J2  K1  K2     NX    NY    NZ   WELLS   NAME
CARFIN
         LGR-OP01   24  24  86  87   1  50      1     8    50     1     GLOBAL /
--
--       DEFINE LGR GRID BLOCKS IN THE Y-DIRECTION
--
NYFIN
         4  4                                                              /

ENDFIN
```

The above example splits the global cells (24, 86-87,1-50) into four and four LGR grid blocks in the y-direction and since the HYFIN keyword has not been supplied, then the host cells will split into equal proportions.

6.3.157 NZFIN – DEFINE THE NUMBER OF LGR GRID BLOCKS IN THE Z-DIRECTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

NZFIN defines the number of Local Grid Refinement ("LGR") cells within a global or host cell in the z-direction via a vector, as opposed to defining the size for each cell for a Cartesian LGR Grid. The LGR keyword in the RUNSPEC section should be activated to indicate an LGR is being used, and the keyword NXFIN should be placed in between the CARFIN and ENDFIN keywords in the GRID section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	NZFIN	NZFIN is a vector of integer numbers describing the number of LGR cells within each defined global or host grid block in the x-direction in a Cartesian LGR grid. Repeat counts may be used, for example 2*2.0.			
		dimensionless	dimensionless	dimensionless	None

Notes:

1) The number of entries should correspond to the NX parameter on the CARFIN keyword in the GRID section.
2) The keyword is terminated by a "/".

Table 6.94: NZFIN Keyword Description

See also the CARFIN, ENDFIN, NXFIN, and NYFIN keywords in the GRID section to fully define a Cartesian LGR grid model.

Example

```
--
--       CARFIN LGR GRID COMMANDS
--
--       LGR        ----- HOST GRID ------    -- CARFIN GRID --  MAX     HOST
--       NAME       I1  I2  J1  J2  K1  K2     NX    NY    NZ   WELLS   NAME
CARFIN
         LGR-OP01   24  24  86  86   1  50      8     1    100    1     GLOBAL /
--
--       DEFINE LGR GRID BLOCKS IN THE Z-DIRECTION
--
NZFIN
         50*2                                                            /

ENDFIN
```

The above example splits the global cells (24, 86, 1-50) into two LGR grid blocks per host cell in the z-direction, and since the HZFIN keyword has not been supplied, then the host cells will split into equal proportions.

6.3.158 OLDTRAN – ACTIVATE CARTESIAN REGULAR GRID TRANSMISSIBILITIES

Description

This keyword switches on Cartesian Regular Grids geometry transmissibility calculation (or block centered transmissibility calculations), which is the default option for this type of grid. Grids defined by the DX, DY, and DZ series of keywords will always invoke this option by default.

For Irregular Corner-Point Grids defined by the COORD and ZCORN keywords Irregular Corner-Point Grid geometry transmissibility calculations should be activated via the NEWTRAN keyword. Again this is automatically invoked if this type of grid is being employed.

There is no data required for this keyword and there is no terminating "/" for this keyword.

Example

```
--
--       ACTIVATE CARTESIAN REGULAR GRID TRANSMISSIBILITIES
--
OLDTRAN
```

The above example manually activates Cartesian Regular Grid transmissibility calculations.

6.3.159 OLDTRANR – ACTIVATE RADIAL REGULAR GRID TRANSMISSIBILITIES

Description

This keyword switches on Radial Regular Grids geometry transmissibility calculation (or block centered transmissibility calculations), which is the default option for this type of grid. Grids defined by the DR, DTHETA, and DZ series of keywords will always invoke this option by default.

For Irregular Corner-Point Grids defined by the COORD and ZCORN keywords Irregular Corner-Point Grid geometry transmissibility calculations should be activated via the NEWTRAN keyword. Again this is automatically invoked if this type of grid is being employed.

There is no data required for this keyword and there is no terminating "/" for this keyword.

Example

```
--
--       ACTIVATE RADIAL REGULAR GRID TRANSMISSIBILITIES
--
OLDTRANR
```

The above example manually activates Radial Regular Grid transmissibility calculations.

6.3.160 OPERATE – DEFINE MATHEMATICAL OPERATIONS ON ARRAYS

Description

This keyword, OPERATE, defines mathematical operations on property arrays (NTG, PORO etc.) and optionally using another property array as input to the function. The keyword allows for various mathematical functions and their associated variables to be defined and applied to the selected array data. Input constants can be real or integer depending on the array type; however, the arrays that can be operated on is dependent on which section the keyword is being used.

No.	Name	Description	Default
1	Y	The name of the array to be modified. This is the keyword name identifying the property and is up to eight characters in length and optionally enclosed in quotes.	None
2	I1	A positive integer that defines the lower bound of the array in the I-direction to be modified must be greater than or equal to one and less than or equal to I2 and NX.	1
3	I2	A positive integer that defines the upper bound of the array in the I-direction to be modified must be greater than or equal to I1 and less than or equal to NX	NX
4	J1	A positive integer that defines the lower bound of the array in the J-direction to be modified must be greater than or equal to one and less than or equal to J2 and NY.	1
5	J2	A positive integer that defines the upper bound of the array in the J-direction to be modified must be greater than or equal to J1 and less than or equal to NY.	NY
6	K1	A positive integer that defines the lower bound of the array in the K-direction to be modified must be greater than or equal to one and less than or equal to K2 and NZ.	1
7	K2	A positive integer that defines the upper bound of the array in the K-direction to be modified must be greater than or equal to K1 and less than or equal to NZ.	NZ
8	EQUATION	A defined character string enclosed in quotes that selects the mathematical function to be applied, using the X array and the ALPHA and BETA constants declared on this keyword. EQUATION should be set to one of the following character strings: 'MULTA' - $Y = \alpha X + \beta$ 'POLY' - $Y = Y + \alpha X^{\beta}$ 'SLOG' - $Y = 10^{\alpha X + \beta}$ 'LOG10' - $Y = \log(X)$ 'LOGE' - $Y = \ln(X)$ 'INV' - $Y = \frac{1}{X}$ 'MULTX' - $Y = \alpha X$ 'ADDX' - $Y = \alpha + X$ 'COPY' - $Y = X$ 'MAXLIM' - $Y = max(\alpha, X)$ 'MINLIM' - $Y = min(\alpha, X)$ 'MULTP' - $Y = \alpha X^{\beta}$ 'ABS' - $Y = \lvert(X)\rvert$ 'MULTIPLY' - $Y = XY$	None

No.	Name	Description	Default
9	X	The name of the array to be used as an input parameter. This is the keyword name identifying the property and is up to eight characters in length and optionally enclosed in quotes.	None
10	ALPHA	An integer or real value that is the α variable in the EQUATION function.	None
11	BETA	An integer or real value that is the β variable in the EQUATION function.	None

Notes:

1) Where NX, NY and NZ are the dimensions of the model as defined on the DIMENS keyword in the RUNSPEC section.
2) Each record must be terminated by a "/" and the keyword is terminated by a "/".

Table 6.95: OPERATE Keyword Description

The applicable arrays for each section are defined in Table 6.96 as shown below.

OPERATE Keyword And Variable Options By Section						
GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
DX	DEPTH	SWL	ENDNUM	PRESSURE		
DY	PORV	SWCR	EQLNUM	SWAT		
DZ	TRANX	SWU	FIPNUM	SGAS		
PERMX	TRANY	SGL	IMBNUM	RV		
PERMY	TRANZ	SGCR	MISCNUM	RS		
PERMZ	DIFFX	SGU	PVTNUM	TBLK		
MULTX	DIFFY	KRW	ROCKNUM	GI		
MULTY	DIFFZ	KRO	SATNUM	OILAPI		
MULTZ	TRANR	KRG	WH2NUM	SALT		
DR	TRANTHT	PCG		GASCONC		
THETA	DIFFR	PCW		SOLVCONC		
PERMR	DIFFTHT			SOLVFRAC		
PERMTHT				SFOAM		
DZNET				SPOLY		
PORO						
NTG						
FLUXNUM						
MULTNUM						
MPANUM						
DIFFX						
DIFFY						
DIFFZ						
DIFFR						
DIFFTHT						

Table 6.96: OPERATE Keyword Applicable Arrays by Section

Note that care should be exercised when performing operations on integer array data as all transforms are performed using floating point arithmetic operations. In addition, operations on any of the transmissibility arrays (TRANX, TRANX-, TRANY, TRANY-, TRANZ, and TRANZ-) may result in untended consequences as these arrays have zero values on the boundary of the grid. In this use OPM ResInsight to verify and visually inspect the results.

Example

The first example uses the MULTP function combined with the Net-to-Gross (NTG) array to re-scale the MULTX, MULTY and MULTZ arrays to reduce the transmissibility in three separation reservoirs based on the reservoir quality (NTG).

```
--
--        MATHEMATICAL OPERATIONS ON ARRAYS BY CELL
--
--        OUTPUT   ---------- BOX ---------   OPERATION   INPUT    ALPHA    BETA
--        ARRAY    I1   I2    J1   J2    K1   K2  ---------   ARRAY    -----    ----
OPERATE
          MULTX    1*   1*    1*   1*      1  32  'MULTP'     NTG      1.00     0.75 / RES1
          MULTY    1*   1*    1*   1*      1  32  'MULTP'     NTG      1.00     0.75 / RES1
          MULTZ    1*   1*    1*   1*      1  32  'MULTP'     NTG      1.00     0.75 / RES1

          MULTX    1*   1*    1*   1*     34  64  'MULTP'     NTG      1.00     0.85 / RES2
          MULTY    1*   1*    1*   1*     34  64  'MULTP'     NTG      1.00     0.85 / RES2
          MULTZ    1*   1*    1*   1*     34  64  'MULTP'     NTG      1.00     0.85 / RES2

          MULTX    1*   1*    1*   1*     67  96  'MULTP'     NTG      1.00     0.50 / RES3
          MULTY    1*   1*    1*   1*     67  96  'MULTP'     NTG      1.00     0.50 / RES3
          MULTZ    1*   1*    1*   1*     67  96  'MULTP'     NTG      1.00     0.50 / RES3
/
```

The next example shows how to set the maximum gas saturation (SGU) based on the minimum (lowest) water saturation (SWL) when using the End-Point Scaling option.

```
--
--        MATHEMATICAL OPERATIONS ON ARRAYS
--
--        OUTPUT   ---------- BOX ---------   OPERATION   INPUT    ALPHA    BETA
--        ARRAY    I1   I2    J1   J2    K1   K2  ---------   ARRAY    -----    ----
OPERATE
          SGU      1*   1*    1*   1*    1*   1*  'MULTA'      SWL     -1.0     1.0      /
/
```

The above example set the maximum gas saturation to be one minus the minimum water saturation.

6.3.161 OPERATER – DEFINE MATHEMATICAL OPERATIONS ON ARRAYS BY REGION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The OPERATER keyword is similar to the OPERATE keyword in the GRID section, except it applies the mathematical operation on specific regions, whereas, OPERATE applies the operations on a cell by cell basis. Here the OPERATER keyword defines mathematical operations on property arrays (NTG, PORO etc.) and optionally using another property array as input to the function. The keyword allows for various mathematical functions and their associated variables to be defined and applied to the selected region data. Input constants can be real or integer depending on the array type; however, the arrays that can be operated on is dependent on which section the keyword is being used.

No.	Name	Description	Default
1	Y	The name of the array to be modified. This is the keyword name identifying the property and is up to eight characters in length and optionally enclosed in quotes.	None
2	REGION	REGION is a positive integer representing the region for which the EQUATION should be applied. The default is to use the region number from the OPERNUM keyword; however this can be reset to another region array via the ARRAY item on this keyword, provided the array exists at the time the keyword is declared in the input deck. Note also the OPERNUM keyword must precede the use of the OPERATER keyword.	0
3	EQUATION	A defined character string enclosed in quotes that selects the mathematical function to be applied, using the X array and the ALPHA and BETA constants declared on this keyword. EQUATION should be set to one of the following character strings: 'MULTA' - $Y = \alpha X + \beta$ 'POLY' - $Y = Y + \alpha X^{\beta}$ 'SLOG' - $Y = 10^{\alpha X + \beta}$ 'LOG10' - $Y = \log(X)$ 'LOGE' - $Y = \ln(X)$ 'INV' - $Y = \frac{1}{X}$ 'MULTX' - $Y = \alpha X$ 'ADDX' - $Y = \alpha + X$ 'COPY' - $Y = X$ 'MAXLIM' - $Y = max(\alpha, X)$ 'MINLIM' - $Y = min(\alpha, X)$ 'MULTP' - $Y = \alpha X^{\beta}$ 'ABS' - $Y = \lvert(X)\rvert$ 'MULTIPLY' - $Y = XY$	None
4	X	The name of the array to be used as an input parameter. This is the keyword name identifying the property and is up to eight characters in length and optionally enclosed in quotes.	None
5	ALPHA	An integer or real value that is the α variable in the EQUATION function.	None
6	BETA	An integer or real value that is the β variable in the EQUATION function.	None

No.	Name	Description	Default
7	ARRAY	The name of the array for which the REGION variable references. This can be any standard region array as declared in the REGION section (FIPNUM, PVTNUM, etc.), provided the array exists at the time the OPERATER keyword is invoked. In addition, the MULTNUM, FLUXNUM and OPERNUM may be used. Only the default value of OPERNUM is supported by OPM Flow.	OPERNUM

Notes:

1) Where the REGION should be less than or equal to the maximum number of regions as defined on the REGDIMS keyword for the FIPNUM and OPERNUM arrays or the GRIDOPTS keyword for the MULTNUM array in the RUNSPEC section..
2) Each record must be terminated by a "/" and the keyword is terminated by a "/".

Table 6.97: OPERATE Keyword Description

The applicable arrays for each section are defined in Table 6.98 as shown below.

OPERATE Keyword And Variable Options By Section						
GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
DX	DEPTH	SWL	ENDNUM	PRESSURE		
DY	PORV	SWCR	EQLNUM	SWAT		
DZ	TRANX	SWU	FIPNUM	SGAS		
PERMX	TRANY	SGL	IMBNUM	RV		
PERMY	TRANZ	SGCR	MISCNUM	RS		
PERMZ	DIFFX	SGU	PVTNUM	TBLK		
MULTX	DIFFY	KRW	ROCKNUM	GI		
MULTY	DIFFZ	KRO	SATNUM	OILAPI		
MULTZ	TRANR	KRG	WH2NUM	SALT		
DR	TRANTHT	PCG		GASCONC		
THETA	DIFFR	PCW		SOLVCONC		
PERMR	DIFFTHT			SOLVFRAC		
PERMTHT				SFOAM		
DZNET				SPOLY		
PORO						
NTG						
FLUXNUM						
MULTNUM						
MPANUM						
DIFFX						
DIFFY						
DIFFZ						
DIFFR						
DIFFTHT						

Table 6.98: OPERATE Keyword Applicable Arrays by Section

Note that care should be exercised when performing operations on integer array data as all transforms are performed using floating point arithmetic operations. In addition, operations on any of the transmissibility arrays (TRANX, TRANX-, TRANY, TRANY-, TRANZ, and TRANZ-) may result in unintended consequences as these arrays have zero values on the boundary of the grid. In this use OPM ResInsight to verify and visually inspect the results.

Note

The OPERATER and OPERATE keywords allow for great flexibility in generating or modifying the simulator's input arrays. In addition, OPM Flow also has a Python facility to manipulate and calculate data that offers even greater flexibility, but note that this feature is not compatible with the commercial simulator.

Finally, OPM ResInsight, the post-processing plotting software, has both Octave and Python scripting facilities that enable both data generation and visual inspection of the results. The resulting calculated arrays can then be exported from OPM ResInsight and "included" back into OPM Flow, thus maintaining compatibility with the commercial simulator.

Example

The first example uses the MULTP function combined with the Net-to-Gross (NTG) array to re-scale the MULTX, MULTY and MULTZ arrays to reduce the transmissibility in three separate reservoirs based on the reservoir quality (NTG). This keyword sequence should be in the GRID section.

```
--
--        MATHEMATICAL OPERATIONS ON ARRAYS BY REGION
--
--        OUTPUT    REGN    OPERATION    SOURCE    ALPHA    BETA     REGN
--        ARRAY     NUM     TYPE         ARRAY     CONST    CONST    ARRAY
OPERATER
          MULTX     1      'MULTP'       NTG       1.00     0.75         / RES1
          MULTY     1      'MULTP'       NTG       1.00     0.75         / RES1
          MULTZ     1      'MULTP'       NTG       1.00     0.75         / RES1

          MULTX     2      'MULTP'       NTG       1.00     0.85         / RES2
          MULTY     2      'MULTP'       NTG       1.00     0.85         / RES2
          MULTZ     2      'MULTP'       NTG       1.00     0.85         / RES2

          MULTX     3      'MULTP'       NTG       1.00     0.50         / RES3
          MULTY     3      'MULTP'       NTG       1.00     0.50         / RES3
          MULTZ     3      'MULTP'       NTG       1.00     0.50         / RES3
/
```

Notice that the ARRAY variable has been defaulted, resulting in OPERNUM being the regional array for the REGION variable.

The next example shows how to set the maximum gas saturation (SGU) based on the minimum (lowest) water saturation (SWL) when using the End-Point Scaling option, in the PROPS section.

```
--
--        MATHEMATICAL OPERATIONS ON ARRAYS
--
--        OUTPUT    REGN    OPERATION    SOURCE    ALPHA    BETA     REGN
--        ARRAY     NUM     TYPE         ARRAY     CONST    CONST    ARRAY
OPERATER
          SGU       1      'MULTA'       SWL       -1.0     1.0          /
          SGU       2      'MULTA'       SWL       -1.0     1.0          /
          SGU       3      'MULTA'       SWL       -1.0     1.0          /
/
```

The above example set the maximum gas saturation to be one minus the minimum water saturation for regions one to three.

The final example shows how to reset the FIPNUM array when the exported array from the earth model does not correspond to the simulator's desired numbering scheme.

```
--
--        MATHEMATICAL OPERATIONS ON ARRAYS BY REGION
--
--        RESET FIPNUM BASED ON MULTNUM AND OPERNUM
--
--        DESTIN    REGN    OPERATION    SOURCE    ALPHA    BETA  INPUT  SEGNUM    EQUIL
--        ARRAY     NUM     TYPE         ARRAY     CONST    CONST ARRAY  NUMBER    NUMBER
OPERATER
          FIPNUM     26     'MULTA'    'MULTNUM'  0.00      1           /  26        1
          FIPNUM     44     'MULTA'    'MULTNUM'  0.00      2           /  44        2
          FIPNUM     62     'MULTA'    'MULTNUM'  0.00      3           /  62        3
          FIPNUM     98     'MULTA'    'MULTNUM'  0.00      4           /  98        4
          FIPNUM    116     'MULTA'    'MULTNUM'  0.00      5           / 116        5
          FIPNUM    134     'MULTA'    'MULTNUM'  0.00      6           / 134        6
          FIPNUM     46     'MULTA'    'MULTNUM'  0.00      7           /  46        7
          FIPNUM     64     'MULTA'    'MULTNUM'  0.00      8           /  64        8
          FIPNUM     82     'MULTA'    'MULTNUM'  0.00      9           /  82        9
          FIPNUM    226     'MULTA'    'MULTNUM'  0.00     10           / 226       10
          FIPNUM    262     'MULTA'    'MULTNUM'  0.00     11           / 262       11
          FIPNUM    280     'MULTA'    'MULTNUM'  0.00     12           / 280       12
          FIPNUM    298     'MULTA'    'MULTNUM'  0.00     13           / 298       13
          FIPNUM     33     'MULTA'    'MULTNUM'  0.00     14           /  33       14
          FIPNUM     51     'MULTA'    'MULTNUM'  0.00     15           /  51       15
          FIPNUM    105     'MULTA'    'MULTNUM'  0.00     16           / 105       16
          FIPNUM    159     'MULTA'    'MULTNUM'  0.00     17           / 159       17
          FIPNUM    195     'MULTA'    'MULTNUM'  0.00     18           / 195       18
          FIPNUM    267     'MULTA'    'MULTNUM'  0.00     19           / 267       19
          FIPNUM    303     'MULTA'    'MULTNUM'  0.00     20           / 303       20
          FIPNUM    321     'MULTA'    'MULTNUM'  0.00     21           / 321       21
          FIPNUM    339     'MULTA'    'MULTNUM'  0.00     22           / 339       22
          FIPNUM     54     'MULTA'    'MULTNUM'  0.00     23           /  54       23
          FIPNUM     72     'MULTA'    'MULTNUM'  0.00     24           /  72       24
          FIPNUM    108     'MULTA'    'MULTNUM'  0.00     25           / 108       25
          FIPNUM    144     'MULTA'    'MULTNUM'  0.00     26           / 144       26
          FIPNUM    270     'MULTA'    'MULTNUM'  0.00     27           / 270       27
/
```

Note that operation can only be done in the REGION section as FIPNUM is only available for use in this section and that the ARRAY variable has been defaulted, resulting in OPERNUM being the regional array for the REGION variable.

6.3.162 OPERNUM – DEFINE REGIONS FOR MATHEMATICAL OPERATIONS ON ARRAYS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword defines the OPERATER region numbers for each grid block. The OPERNUM keyword defines the region numbers for each grid block, as such there must be one entry for each cell in the model. The array can also be used with the EQUALREG, ADDREG, COPYREG, MULTIREG, MULTREGP and MULTREGT keywords, as well as the OPERATER keyword in calculating various grid properties in the GRID and REGION section.

No.	Name	Description	Default
1	OPERNUM	OPERNUM defines an array of positive integers greater than or equal to one that assigns a grid cell to a particular OPERNUM region. The maximum number of OPERNUM regions is set by the NOPREG variable on the REGDIMS keyword in the RUNSPEC section. Note that the default value of zero implies that the calculations requested by the OPERATER keyword will not be performed.	0

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.

2) The keyword is terminated by a "/".

Table 6.99: OPERNUM Keyword Description

Examples

The example below sets three OPERNUM regions for a 4 x 5 x 2 model.

```
--
--        DEFINE OPERNUM REGIONS FOR ALL CELLS
--
OPERNUM
          2  2  1  1  2  2  1  1  1  1  1  1  1  1  1  1  1  1  1  1
          3  3  1  1  3  3  1  1  1  1  1  1  1  1  1  1  1  1  1  1
/
```

Alternatively the EQUALS keyword could be employed to accomplish the same task, that is:

```
--
--        ARRAY       CONSTANT     ---------- BOX ---------
--                                 I1  I2   J1  J2   K1  K2
EQUALS
          OPERNUM     1            1*  1*   1*  1*   1*  1* / SET REGION 1
          OPERNUM     2            1   2    1   2    1   1  / SET REGION 2
          OPERNUM     3            1   2    1   2    2   2  / SET REGION 3
/
```

One can then increase PERMX by 25% in region three only.

```
--
--        MULTIPLY AN ARRAY BY A CONSTANT BASED ON A REGION NUMBER
--
--        ARRAY     CONSTANT  REGION   REGION ARRAY
--                  VALUE     NUMBER    M / F / O
MULTIREG
          PERMX     1.25      3         O                  /
/
```

6.3.163 OUTRAD - DEFINE THE OUTER RADIUS OF A RADIAL GRID

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

OUTRAD defines the OUTER radius of the reservoir model for a radial or spider grid geometry. The RADIAL or SPIDER keyword in the RUNSPEC should be activated to indicate that radial or spider geometry is being used.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	OUTRAD	A single real positive number greater than INRAD defining the outer radius of a radial grid.			
		feet	m	cm	None
Notes: 1) The keyword is terminated by a "/".					

Table 6.100: OUTRAD Keyword Description

Note that the SPIDER keyword activates OPM Flow's radial grid geometry option for the model. This keyword will create a spiderweb-shaped grid based on a corner-point grid using the standard radial grid keywords: INRAD, DRV, DTHETAV, DZ/DZV etc. in the GRID the section. A spider grid can be viewed in 3D in OPM ResInsight unlike radial grids that cannot be viewed in the software. To overcome this, the simulator now converts radial grids to Irregular Corner-Point Grids and adjusts the model's pore volume to reflect radial coordinates; thus, overcoming the display limitation.

The keyword allows for an alternative method of entering the size of the R direction grid cells instead of entering the data using the DR or DRV keywords in the GRID section. Given the internal radius set by the INRAD keyword, the external radius set by the OUTRAD keyword and the number of grid cells in the R direction set by the NX variable on the DIMENS keyword in the RUNSPEC section, the R direction cells sizes are computed automatically on a geometric spacing, as defined by:

$$\frac{R_i}{R_{i-1}} = \left(\frac{OUTRAD}{R_{i_j-1}}\right)^{\frac{1}{(NX - i_j + 1)}} \tag{6.12}$$

or

$$R_i = \left(R_{i_j-1}\right)\left(\frac{OUTRAD}{R_{i_j-1}}\right)^{\frac{(i - i_j + 1)}{(NX - i_j + 1)}} \tag{6.13}$$

and the DR value for the i^{th} cell, that is the value that can also be manually entered on the DR keyword in the GRID section, is given by:

$$DR_i = R_i - R_{i-1} \tag{6.14}$$

Where:

DR_i = DR value for the i^{th} cell
R_i = current total radius for the i radii.
Ri-1 = total radius for the i – 1 radii.

NX (NR) = number of radial grid cells excluding the inner radius
OUTRAD = the outer radius of the radial grid, the value includes the inner radius.

For example, given an inner radius set to 0.25, an outer radius of 2,050 and the number of cells in the R direction set to ten, then Table 6.101 shows the grid size calculations.

OUTRAD Radial Grid Example			
INRAD	0.25		
OUTRAD	2050.0		
NX	10		
NX	**Ri**	**DR**	**Ratio**
0	0.250	0.250	
1	0.616	0.366	1.463
2	1.516	0.900	2.463
3	3.733	2.217	2.463
4	9.193	5.460	2.463
5	22.638	13.445	2.463
6	55.748	33.109	2.463
7	137.281	81.533	2.463
8	338.058	200.777	2.463
9	832.477	494.420	2.463
10	2050.000	1217.523	2.463
Total		**2050.000**	

Table 6.101: OUTRAD Radial Grid Example

See also the DR, DRV, DTHETA, DTHETAV and TOPS keywords to fully define a Radial or Spider Grid.

Example

```
--
--       INNER RADIUS OF FIRST GRID BLOCK IN THE RADIAL DIRECTION
--
INRAD
         0.25                                                                  /
--
--       OUTER RADIUS OF FIRST GRID BLOCK IN THE RADIAL DIRECTION
--
OUTRAD
         2050.0                                                                /
```

The above example defines the inner radius to be 0.25 and the outer radius to be 2,050 feet. Note that the outer radius includes the inner radius.

6.3.164 PARAOPTS – DEFINE PARALLEL RUN OPTIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The PARAOPTS keyword defines various options for parallel runs, for when the Parallel option has been invoked by the PARALLEL keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

See section *2.2 Running OPM Flow 2023-10 From The Command Line* on how to run OPM Flow in parallel mode.

6.3.165 PEBI – ACTIVATE AND DEFINED PEBI GRID OPTIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

PEBI activates the unstructured Perpendicular Bisector ("PEBI")[125] and [126] loading of grid data generated by an external pre-processing program for generating simulation grids.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description	Default
1	OPTION1	A defined character string that activates or deactivates the checking of negative transmissibility values. OPTION1 should be set to YES to check for negative values, or NO switches off this option.	NO
2	OPTION2	A defined character string that activates or deactivates the calculation of pore volumes and transmissibilities. OPTION2 should be set to YES if the pore volumes and transmissibilities are provided, or NO for the values to calculated by the simulator.	NO

Table 6.102: PEBI Keyword Description

Example

```
--
--      OPTION1  OPTION2
--      CHECK    CALCULATE
PEBI
        NO       YES                                                    /
```

The above example switches off the negative transmissibility check and requests that the simulator calculates pore volumes and transmissibilities as they are not provided by the input data.

[125] *Heinemann, Z.E. and Brand, C.W. 1988. Gridding Techniques in Reservoir Simulation. Proc., First Intl. Forum on Reservoir Simulation, Alpbach, Austria, 339.*

[126] *Heinemann, Z.E., Brand, C.W., Munka, M. et al. 1991. Modeling Reservoir Geometry With Irregular Grids. SPE Res Eng 6 (2): 225–232. SPE-18412-PA. http://dx.doi.org/10.2118/18412-PA*

6.3.166 PERMAVE – DEFINE AVERAGE TRANSMISSIBILITY COEFFICIENTS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The PERMAVE keyword defines the three directional exponent coefficients used to average the grid block permeabilities between two neighboring cells when calculating the transmissibility between the two blocks. The keyword can be used to change from the default weighted harmonic averaging (coefficient set equal to -1), to geometric (coefficient equal to zero), or to arithmetic averaging (coefficient equal to 1). The three coefficients represent the averaging in the x-, y- and z-directions.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

6.3.167 PERMJFUN – DEFINE LEVERETT J-FUNCTION PERMEABILITY FOR ALL CELLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

PERMJFUN defines the permeability to be used in de-normalizing the Leverett J-Functions[127] for when the PERM variable on the JFUNC or the JFUNCR keyword in the GRID section has been set to "U", as oppose to using PERMX, PERMY, PERMZ arrays etc.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	PERMJFUN	PERMJFUN is an array of real positive numbers assigning the permeability to be used in de-normalizing the Leverett J-Function to each cell in the model. Repeat counts may be used, for example 20*100.0.			
		mD	mD	mD	None

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.

2) The keyword is terminated by a "/".

Table 6.103: PERMJFUN Keyword Description

For grid blocks that have not been assigned a PERMJFUN value the default directional permeability will be used, that is the average of PERMX and PERMY.

See also the PERMX, PERMY and PERMZ keywords to fully define the permeability for the model.

Example

```
--
--       DEFINE GRID BLOCK PERMJFUN FOR ALL CELLS (BASED ON NX x NY x NZ = 300)
--
PERMJFUN
         100*500.0   100*50.0   100*200.0                                    /
```

The above example defines the PERMJFUN to be 500.0, 50.0, and 200.0 for the first, second and third layers in the model for all 300 cells, as defined by the DIMENS keyword in the RUNSPEC section.

[127] *Leverett, M. C.; "Capillary Behaviour in Porous Solids", Trans. AIME (1941) 142, 152-168.*

6.3.168 PERMR – DEFINE THE PERMEABILITY FOR EACH CELL IN THE R DIRECTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

PERMR sets the permeability for each cell in the R direction in a radial geometry grid. The RADIAL or SPIDER keywords in the RUNSPEC should be activated to indicate that radial or spider geometry is being used.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	PERMR	PERMR is an array of real positive numbers assigning the permeability in the R direction to each cell in the model. This equivalent to PERMX in a Cartesian grid. Repeat counts may be used, for example 20*100.0.			
		mD	mD	mD	None

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.

2) The keyword is terminated by a "/".

Table 6.104: PERMR Keyword Description

Note that the SPIDER keyword activates OPM Flow's radial grid geometry option for the model. This keyword will create a spiderweb-shaped grid based on a corner-point grid using the standard radial grid keywords: INRAD, DRV, DTHETAV, DZ/DZV etc. in the GRID the section. A spider grid can be viewed in 3D in OPM ResInsight unlike radial grids that cannot be viewed in the software. To overcome this, the simulator now converts radial grids to Irregular Corner-Point Grids and adjusts the model's pore volume to reflect radial coordinates; thus, overcoming the display limitation.

See also the PERMTHT and PERMZ keywords to fully define the permeability for a radial or spider grid model.

Example

```
--
--       DEFINE GRID BLOCK PERMR DATA FOR ALL CELLS (BASED ON NR x NY x NZ = 300)
--
PERMR
         100*500.0   100*50.0   100*200.0                                    /
```

The above example defines the PERMR to be 500.0, 50.0, and 200.0 for the first, second and third layers in the model for all 300 cells, as defined by the DIMENS keyword (10, 10, 3) in the RUNSPEC section.

6.3.169 PERMTHT – DEFINE THE PERMEABILITY FOR EACH CELL IN THE THETA DIRECTION

Description

PERMTHT sets the permeability for each cell in the THETA direction in a radial geometry grid. The RADIAL or SPIDER keyword in the RUNSPEC should be activated to indicate that radial or spider geometry is being used.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	PERMTHT	PERMTHT is an array of real positive numbers assigning the permeability in the THETA direction to each cell in the model. This equivalent to PERMY in a Cartesian grid. Repeat counts may be used, for example 20*100.0.			
		mD	mD	mD	None

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.

2) The keyword is terminated by a "/".

Table 6.105: PERMTHT Keyword Description

Note that the SPIDER keyword activates OPM Flow's radial grid geometry option for the model. This keyword will create a spiderweb-shaped grid based on a corner-point grid using the standard radial grid keywords: INRAD, DRV, DTHETAV, DZ/DZV etc. in the GRID the section. A spider grid can be viewed in 3D in OPM ResInsight unlike radial grids that cannot be viewed in the software. To overcome this, the simulator now converts radial grids to Irregular Corner-Point Grids and adjusts the model's pore volume to reflect radial coordinates; thus, overcoming the display limitation.

See also the PERMR and PERMZ keywords to fully define the permeability for a radial or spider grid model.

Example

```
--
--       DEFINE GRID BLOCK PERMTHT DATA FOR ALL CELLS (NX x NY x NZ = 300)
--
PERMTHT
         100*500.0   100*50.0   100*200.0                                  /
```

The above example defines the PERMTHT to be 500.0, 50.0, and 200.0 for the first, second and third layers in the model for all 300 cells, as defined by the DIMENS keyword (10, 10, 3) in the RUNSPEC section.

6.3.170 PERMX - DEFINE THE PERMEABILITY IN THE X DIRECTION FOR ALL THE CELLS

Description

PERMX defines the permeability in the X direction for all the cells in the model via an array. The keyword can be used for all grid types, except for the Radial Grid geometry.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	PERMX	PERMX is an array of real positive numbers assigning the permeability in the X direction to each cell in the model. Repeat counts may be used, for example 20*100.0.			
		mD	mD	mD	None

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.

2) The keyword is terminated by a "/".

Table 6.106: PERMX Keyword Description

See also the PERMY and PERMZ keywords to fully define the permeability for the model.

Example

```
--
--       DEFINE GRID BLOCK PERMX DATA FOR ALL CELLS (BASED ON NX x NY x NZ = 300)
--
PERMX
         100*500.0   100*50.0   100*200.0                                    /
```

The above example defines the PERMX to be 500.0, 50.0, and 200.0 for the first, second and third layers in the model for all 300 cells, as defined by the DIMENS keyword in the RUNSPEC section.

6.3.171 PERMY - DEFINE THE PERMEABILITY IN THE Y DIRECTION FOR ALL THE CELLS

Description

PERMY defines the permeability in the Y direction for all the cells in the model via an array. The keyword can be used for all grid types, except for the Radial Grid geometry.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	PERMY	PERMY is an array of real positive numbers assigning the permeability in the Y direction to each cell in the model. Repeat counts may be used, for example 20*100.0.			
		mD	mD	mD	None

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) The keyword is terminated by a "/".

Table 6.107: PERMY Keyword Description

See also the PERMX and PERMZ keywords to fully define the permeability for the model.

Example

```
--
--       DEFINE GRID BLOCK PERMY DATA FOR ALL CELLS (BASED ON NX x NY x NZ = 300)
--
PERMY
         100*500.0   100*50.0   100*200.0                                    /
```

The above example defines the PERMY to be 500.0, 50.0, and 200.0 for the first, second and third layers in the model for all 300 cells, as defined by the DIMENS keyword in the RUNSPEC section.

6.3.172 PERMZ - DEFINE THE PERMEABILITY IN THE Z DIRECTION FOR ALL THE CELLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

PERMZ defines the permeability in the Z direction for all the cells in the model via an array. The keyword can be used for all grid types, except for the Radial Grid geometry.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	PERMZ	PERMZ is an array of real positive numbers assigning the permeability in the Z direction to each cell in the model. Repeat counts may be used, for example 200*50.0.			
		mD	mD	mD	None

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.

2) The keyword is terminated by a "/".

Table 6.108: PERMZ Keyword Description

Note

Although PERMX and PERMY are commonly set to be equal, PERMZ is typically not equal to either PERMX or PERMY. Normally PERMZ is set as a fraction of PERMX with typical values ranging from 0.1 to 0.5 times PERMX.

See also the PERMX and PERMY keywords to fully define the permeability for the model.

Example

The example below defines the PERMZ to be 50.0, 5.0, and 20.0 for the first, second and third layers in the model for all 300 cells, as defined by the DIMENS keyword in the RUNSPEC section.

```
--
--       DEFINE GRID BLOCK PERMZ DATA FOR ALL CELLS (BASED ON NX x NY x NZ = 300)
--
PERMZ
         100*50.0    100*5.0   100*20.0                                    /
```

The next example sets PERMX to be 500.0, 50.0, and 200.0 for the first, second and third layers in the model for all 300 cells, as defined by the DIMENS keyword in the RUNSPEC section. It then copies the PERMX values to the PERMY and PERMZ arrays, and finally multiplies PERMZ by 0.1 times to get the final values for PERMZ.

```
--
--       DEFINE GRID BLOCK PERMX DATA FOR ALL CELLS
--
PERMX
         100*500.0   100*50.0   100*200.0                                    /
--
--       SOURCE      DESTIN.      ---------- BOX ---------
--                                I1  I2   J1  J2   K1  K2
COPY
         PERMX       PERMY        1*  1*   1*  1*   1*  1* / CREATE PERMY
         PERMX       PERMZ        1*  1*   1*  1*   1*  1* / CREATE PERMZ
/
--
--       ARRAY       CONSTANT     ---------- BOX ---------
--                                I1  I2   J1  J2   K1  K2
MULTIPLY
         PERMZ       0.10000      1*  1*   1*  1*   1*  1* / PERMZ * 0.1
/
```

The above sequence of keywords is quite common in input decks, that is copying the PERMX data to the PERMY and PERMZ arrays and then adjusting the PERMY and PERMZ arrays as required using the MULTIPLY keyword.

6.3.173 PETGRID – LOAD A GENERIC SIMULATION GRID FILE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The PETGRID keyword instructs the simulator to load a Generic Simulation Grid (*.GSG) file that contains grid geometry data.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

6.3.174 PINCH – DEFINE PINCH-OUT LAYER OPTIONS

Description

The PINCH keyword defines the parameters used to control the generation of Non-Neighbor Connections ("NNCs") in the vertical (K) direction due to layers pinching out. This keyword is applied to all layers in the model as opposed to the PINCHREG keyword that offers more flexibility by applying the pinch-out controls to various regions in the model defined by the PINCHNUM keyword.

OPM Flow will automatically generate connections between non neighbor cells in the vertical direction based on the parameters on this keyword.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	PINCHTHK	A real number defining the pinch-out threshold thickness for any cell. NNCs are generated across inactive cells having a vertical thickness less than PINCHTHK.			
		ft. 0.001	m 0.001	cm 0.001	Defined
2	PINCHOPT	A character string controlling the generation of pinch-outs when the MINPV keyword has been used to deactivate cells with small pore volumes. PINCHOPT can either be set to: 1) GAP to allow the generation of NNCs across cells that have been made inactive with the MINPV keyword when the thickness is greater than PINCHTHK threshold. 2) NOGAP to enforce the strict adherence to the PINCHTHK threshold whether or not cells have been made inactive due to the MINPV keyword.			GAP
3	PINCHGAP	A real number defining the maximum "empty" thickness allowed between grid blocks in adjacent grid layers for a non-zero transmissibility to exist between them.			
		ft. 1.0E20	m 1.0E20	cm 1.0E20	Defined
4	PINCHCAL	A character string controlling the calculation of the pinch-out transmissibilities. PINCHCAL can either be set to: 1) TOPBOT results in the pinch-out transmissibility being calculated from the half-cell Z-direction transmissibilities of the active cells on either side of the pinched-out layers. 2) ALL results in the pinch-out transmissibility being calculated from the Z-direction transmissibilities harmonic average of all the cells between the active cells on either side of the pinched-out layers.			TOPBOT

No.	Name	Description			Default
		Field	Metric	Laboratory	
5	PINCHMUL	A character string controlling the calculation of the pinch-out transmissibilities when adjustments have been made by the MULTZ keyword. PINCHMUL can either be set to: 1) TOP results in the pinch-out transmissibility being calculated from the active cell at the top of the pinch-out. 2) ALL results in the pinch-out transmissibility being calculated from the minimum value of the MULTZ of the active cell at the top of the pinch-out and all the inactive cells in the pinch-out vertical column. Note if PINCHCAL has been set equal to ALL then PINCHMUL is reset to TOP, irrespective of the entered value for PINCHMUL.			TOP
Notes: 1) The keyword is terminated by a "/".					

Table 6.109: PINCH Keyword Description

Examples

The first example below will create NNCs between the cells above and below any cell having vertical thickness less than 0.01 in either feet or metres.

```
--
--       SET PINCH-OUT PARAMETERS FOR CALCULATING PINCH-OUT PROPERTIES
--
PINCH
--       THRESHOLD   GAP      EMPTY   TRANS    MULTZ
--       THICKNESS   NO GAP   GAP     CALC     CALC
         0.01        1*       1*      1*       1*                    /
```

For the second example, the MINPV keyword is used to set the minimum pore volume to 500 m3 (metric units) and then the PINCH keyword is invoked with PINCHGAP set equal to GAP, as follows:

```
--
--       MINIMUM PORE VOLUME FOR ACTIVE CELLS
--
MINPV
         500.0                                                         /
--
--       SET PINCH-OUT CRITERIA FOR THE MODEL
--
PINCH
--       THRESHOLD   GAP      EMPTY   TRANS    MULTZ
--       THICKNESS   NO GAP   GAP     CALC     CALC
         0.1         GAP       1*    1*       1*                    /
```

In the above example the MINPV keyword will deactivate all cells with pore volumes less than 500 m3. These deactivated cells are inactive in the model and therefore are not included in the flow calculations; however, by default they will result in no-flow barriers but may not be thin enough for PINCH to create NNCs across them. By setting PINCHGAP equal to GAP on the PINCH keyword (the default setting), then OPM Flow generates NNCs across the cells that have been deactivated by the MINPV keyword. However, in this case there may be grid blocks in the model with a pore volume greater than MINPV but a thickness less than the pinch-out threshold. These cells will not be deactivated by the PINCH keyword.

6.3.175 PINCHNUM – DEFINE PINCH-OUT REGIONS FOR THE PINCHREG KEYWORD

Description

The PINCHNUM keyword defines the pinch-out region numbers for each grid block, as such there must be one entry for each cell in the model. The array is used with the PINCHREG keyword to set the pinch-out options and threshold thickness for each region.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description	Default
1	PINCHNUM	PINCHNUM defines an array of positive integers assigning a grid cell to a particular PINCHNUM region. The maximum number of PINCHNUM regions is set by the NRPINC variable on the GRIDOPTS keyword in the RUNSPEC section.	1

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.

2) The keyword is terminated by a "/".

Table 6.110: PINCHNUM Keyword Description

Example

The example below sets defines three PINCHNUM regions for various layers in a model based on the model's layering.

```
--
--        ARRAY       CONSTANT     ---------- BOX ---------
--                                 I1  I2   J1  J2   K1  K2
EQUALS
          MULTNUM     1            1*  1*   1*  1*   1*  1* / SET REGION 1
          MULTNUM     2            1   2    1   2    10  50 / SET REGION 2
          MULTNUM     3            1   2    1   2    51 100 / SET REGION 3
/
```

One can then set the pinch-out criteria for each region.

```
--
--        SET PINCH-OUT CRITERA VIA THE PINCHNUM REGION
--
PINCHREG
--        THRESHOLD   GAP      EMPTY    TRANS
--        THICKNESS   NO GAP   GAP      CALC
          0.1         1*       1*       1*                  / PINCHNUM 01
          1.0         1*       10       1*                  / PINCHNUM 02
          1.0         NOGAP    20       1*                  / PINCHNUM 03
```

The above example sets the default pinch-out criteria for grid blocks defined as region one via the PINCHNUM array and different criteria for regions two and three.

6.3.176 PINCHOUT - DEFINE PINCH-OUT LAYERS OPTION (FIXED)

Description

The PINCHOUT keyword activates the generation of Non-Neighbor Connections ("NNCs") in the vertical (K) direction due to layers pinching out, using a constant threshold thickness of 0.001 for all unit systems. See also the PINCH keyword in the GRID section that allows for specifying the threshold thickness and other parameters on a layer basis, and the PINCHREG keyword that applies the pinch-out controls to various regions in the model defined by the PINCHNUM keyword.

There is no data required for this keyword and there is no terminating "/" for this keyword.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

Example

The example will create NNCs between the cells above and below any cell having vertical thickness less than 0.001 in either feet or metres.

```
--
--       SET PINCH-OUT CRITERA WITH CONSTANT THRESHOLD THICKNESS OF 0.001
--
PINCHOUT
```

6.3.177 PINCHREG - DEFINE PINCH-OUT REGION OPTIONS

Description

The PINCHREG keyword defines the parameters used to control the generation of Non-Neighbor Connections ("NNCs") in the vertical (K) direction due to layers pinching out in combination with the PINCHNUM keyword. This allows different regions in the model to use different criteria in controlling the how pinch-outs are generated. The keyword should contain NRPINC records defining the criteria for each pinch-out region defined with the PINCHNUM keyword. NRPINC is the maximum number of PINCHNUM regions defined via the GRIDOPTS keyword in the RUNSPEC section.

An alternative method to set the pinch-out criteria is to use the PINCH keyword, that applies the criteria to the whole model.

OPM Flow will automatically generate connections between non neighbor cells in the vertical direction based on the parameters on this keyword.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	PINCHTHK	A real number defining the pinch-out threshold thickness for any cell. NNCs are generated across inactive cells having a vertical thickness less than PINCHTHK.			
		ft. 0.001	m 0.001	cm 0.001	Defined
2	PINCHOPT	A character string controlling the generation of pinch-outs when the MINPV keyword has been used to deactivate cells with small pore volumes. PINCHOPT can either be set to: 1) GAP to allow the generation of NNCs across cells that have been made inactive with the MINPV keyword when the thickness is greater than PINCHTHK threshold. 2) NOGAP to enforce the strict adherence to the PINCHTHK threshold whether or not cells have been made inactive due to the MINPV keyword.			GAP
3	PINCHGAP	A real number defining the maximum "empty" thickness allowed between grid blocks in adjacent grid layers for a non-zero transmissibility to exist between them.			
		ft. 1.0E20	m 1.0E20	cm 1.0E20	Defined
4	PINCHCAL	A character string controlling the calculation of the pinch-out transmissibilities. PINCHCAL can either be set to: 1) TOPBOT results in the pinch-out transmissibility being calculated from the half-cell Z-direction transmissibilities of the active cells on either side of the pinched-out layers. 2) ALL results in the pinch-out transmissibility being calculated from the Z-direction transmissibilities harmonic average of all the cells between the active cells on either side of the pinched-out layers.			TOPBOT

No.	Name	Description			Default
		Field	Metric	Laboratory	
5	PINCHMUL	A character string controlling the calculation of the pinch-out transmissibilities when adjustments have been made by the MULTZ keyword. PINCHMUL can either be set to: 1) TOP results in the pinch-out transmissibility being calculated from the active cell at the top of the pinch-out. 2) ALL results in the pinch-out transmissibility being calculated from the minimum value of the MULTZ of the active cell at the top of the pinch-out and all the inactive cells in the pinch-out vertical column. Note if PINCHCAL has been set equal to ALL then PINCHMUL is reset to TOP, irrespective of the entered value for PINCHMUL.			TOP
Notes: 1) The keyword should contain NRPINC records defining the criteria for each pinch-out region defined with the PINCHNUM keyword. NRPINC is the maximum number of PINCHNUM regions defined via the GRIDOPTS keyword in the RUNSPEC section. 2) Each record must be terminated by a "/" there is no keyword terminating "/".					

Table 6.111: PINCHREG Keyword Description

Example

```
--
--       SET PINCH-OUT CRITERA VIA THE PINCHNUM REGION
--
PINCHREG
--       THRESHOLD   GAP      EMPTY   TRANS
--       THICKNESS   NO GAP   GAP     CALC
         0.1         1*       1*      1*                  / PINCHNUM 01
         1.0         1*       10      1*                  / PINCHNUM 02
         1.0         NOGAP    20      1*                  / PINCHNUM 03
```

The above example sets the default pinch-out criteria for grid blocks defined as region one via the PINCHNUM array and different values for regions two and three.

6.3.178 PINCHXY – DEFINE PINCH-OUT AREAL OPTIONS

Description

The PINCHXY keyword defines the x-direction and y-direction threshold thickness used to control the generation of Non-Neighbor Connections ("NNCs") in the x- and y- directions for missing cells in the areal plane.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	PINCHTHX	A real number defining the pinch-out threshold width for any cell in the x-direction. NNCs are generated across inactive cells having a width less than PINCHTHX in the x-direction.			
		ft. 0.001	m 0.001	cm 0.001	Defined
2	PINCHTHY	A real number defining the pinch-out threshold width for any cell in the y-direction. NNCs are generated across inactive cells having a width less than PINCHTHY in the y-direction.			
		ft. 0.001	m 0.001	cm 0.001	Defined
Notes: 1) The keyword is terminated by a "/".					

Table 6.112: PINCHXY Keyword Description

Example

The example below will create NNCs between the cells in the areal plane having cell widths less than 0.01 in either feet or metres in both the x- and y-directions.

```
--
--        SET PINCH-OUT PARAMETERS FOR AREAL PLANE
--
PINCHXY
--        X-DIRC      Y-DIRC
--        THRESHOLD   THRESHOLD
--
          0.01        0.01                                    /
```

6.3.179 PORO - DEFINE THE POROSITY VALUES FOR ALL THE CELLS

Description

PORO defines the porosity for all the cells in the model via an array. The keyword can be used with all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	PORO	PORO is an array of real positive numbers that are greater than or equal to zero and less than or equal to one that are the porosity values for each cell in the model. Repeat counts may be used, for example 3000*0.15			
		dimensionless	dimensionless	dimensionless	None

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) The keyword is terminated by a "/".

Table 6.113: PORO Keyword Description

See also the NTG, PERMX, PERMY and PERMX keywords to fully define a grid's properties

Example

```
--
--    DEFINE GRID BLOCK POROSITY DATA FOR ALL CELLS (BASED ON NX x NY x NZ = 300)
--
PORO
      300*0.300                                                              /
```

The above example defines a constant porosity of 0.300 to all 300 cells in the model as defined by the DIMENS keyword in the RUNSPEC section.

6.3.180 PYEND – END THE DEFINITION OF A PYINPUT SECTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The PYINPUT and PYEND keywords are a part of OPM Flow's Python scripting facility that processes standard Python commands that can be used to manipulate and define the simulators input parameters during processing of the input deck. The main purpose of the facility is to script the construction of the various keywords.

PYINPUT declares the start of a PYINPUT Definition Section on a single separate line, which is then followed by various standard Python commands, with one command per line. A PYINPUT Definition Section is terminated by a PYEND keyword (this keyword) on a separate single line.

There is no data required for this keyword and there is no terminating "/" for this keyword.

Although this keyword is read by OPM Flow and the script processing has been implemented, one should use caution when using this facility as it may result in OPM Flow aborting. This is because the PYINPUT facility allows the user to implement complex functionality and the implementation is new for the 2020-04 release. Users should therefore use caution when using this facility.

Note

This is an OPM Flow specific keyword for the simulator's scripting facility using the standard Python interpreter, as such it gives more flexibility than the commercial simulator's data editing keywords (ADD, EQUAL, MULTIPLY, etc.), although OPM Flow also supports these keywords as well.

The PYINPUT facility should be considered experimental as details of the OPM Flow - Python interface might change for future releases. In particular, the current implementation is quite minimal; however, future releases are expected to add more entry points in the simulator's deck class which can be used to manipulate the input deck as the data is loaded. As a user you are encouraged to come with wishes in this regard.

The PYINPUT facility is very powerful and allows for any piece of Python code to be included and run, including potentially malicious code. The important point is to scrutinize the Python code in between PYINPUT and PYEND in a deck you receive from other parties.

See also the PYACTION keyword in the SCHEDULE section which is also part of OPM Flow's Python scripting facility, that loads a standard Python script file that can be used to define a series of conditions and actions as the simulation proceeds through time.

Example

The example shows how to construct the DX variable in the GRID section and to add the resulting DX array as part of the input deck.

```
--
--       START OF PYINPUT SECTION
--
PYINPUT
#
# Import Numpy Model
#
import numpy as np
#
# Define DX and Get the Input Decks Unit Systems
#
dx = np.array([100.0, 100.0, 100.0, 100.0])
active_unit_system  = context.deck.active_unit_system()
default_unit_system = context.deck.default_unit_system()
#
# Set DX in the Input Deck
#
kw = context.DeckKeyword( context.parser['DX'], dx, active_unit_system,
default_unit_system )
context.deck.add(kw)

PYEND
```

The active Parser objects are accessible as context.parser and the active Deck object is available as context.deck.

6.3.181 PYINPUT – DEFINE THE START OF A PYINPUT SECTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The PYINPUT and PYEND keywords are a part of OPM Flow's Python scripting facility that processes standard Python commands that can be used to manipulate and define the simulators input parameters during processing of the input deck. The main purpose of the facility is to script the construction of the various keywords used by the simulator.

PYINPUT declares the start of a PYINPUT Definition Section on a single separate line, which is then followed by various standard Python commands, with one command per line. A PYINPUT Definition Section is terminated by a PYEND keyword on a separate single line.

There is no data required for this keyword and there is no terminating "/" for this keyword.

Although this keyword is read by OPM Flow and the script processing has been implemented, one should use caution when using this facility as it may result in OPM Flow aborting. This is because the PYINPUT facility allows the user to implement complex functionality and the implementation is new for the 2020-04 release. Users should therefore use caution when using this facility.

Note

This is an OPM Flow specific keyword for the simulator's scripting facility using the standard Python interpreter, as such it gives more flexibility than the commercial simulator's data editing keywords (ADD, EQUAL, MULTIPLY, etc.), although OPM Flow also supports these keywords as well.

The PYINPUT facility should be considered experimental as details of the OPM Flow - Python interface might change for future releases. In particular, the current implementation is quite minimal; however, future releases are expected to add more entry points in the simulator's deck class which can be used to manipulate the input deck as the data is loaded. As a user you are encouraged to come with wishes in this regard.

The PYINPUT facility is very powerful and allows for any piece of Python code to be included and run, including potentially malicious code. The important point is to scrutinize the Python code in between PYINPUT and PYEND in a deck you receive from other parties.

No.	Name	Description	Default
PYINPUT		PYINPUT declares the start of a PYINPUT Definition Section. This is then followed by any number Python commands.	Not Applicable
1-1	PYTHON	A series of standard Python commands with one line per command. The active Parser objects are accessible as context.parser and the active Deck object is available as context.deck.	
PYEND		PYEND declares the end of a PYINPUT Definition Section. The Python code between PYINPUT and PYEND is read and executed, and the simulator thenreturns to reading the normal simulation input deck.	

Notes:

1) There is no terminating "/" for this keyword, instead the PYEND keyword terminates the keyword.

Table 6.114: PYINPUT Keyword Description

The PYINPUT/PYEND set of keywords is a result of combining two programming languages, the interactive Python interpreter and OPM Flow's source code language C++. When combining two languages one extends and embeds one into the other. When extending Python with C++ the functionality implemented in C++ is made available to Python applications, when embedding Python in C++ one can call Python functions from within C++. The PYINPUT/PYEND set of keywords is based on embedding a Python interpreter in the C++ OPM Flow simulator, but the Python code actually runs as part of the PYINPUT keyword is based on wrapping C++ objects in Python, that is extending Python.

The Python code in between the PYINPUT and PYEND keywords are imported during processing of the input deck and as such this implies that basic Python syntax checking is performed during reading the Python script.

See also the PYACTION keyword in the SCHEDULE section which is also part of OPM Flow's Python scripting facility, that loads a standard Python script file that can be used to define a series of conditions and actions as the simulation proceeds through time.

Example

The example shows how to construct the DX variable in the GRID section and to add the resulting DX array as part of the input deck.

```
--
--       START OF PYINPUT SECTION
--
PYINPUT
#
# Import Numpy Model
#
import numpy as np
#
# Define DX and Get the Input Decks Unit Systems
#
dx = np.array([100.0, 100.0, 100.0, 100.0])
active_unit_system  = context.deck.active_unit_system()
default_unit_system = context.deck.default_unit_system()
#
# Set DX in the Input Deck
#
kw = context.DeckKeyword( context.parser['DX'], dx, active_unit_system,
default_unit_system )
context.deck.add(kw)

PYEND
```

The active Parser objects is accessible as context.parser and the active Deck object is available as context.deck.

6.3.182 QMOBIL ACTIVATE OR DEACTIVATE LGR END-POINT MOBILITY CORRECTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The QMOBIL keyword activates or deactivates the end-point mobility correction for Local Grid Refinements ("LGR"), for when LGRs have been activated for the input deck using the LGR keyword in the RUNSPEC section. QMOBIL should be placed in between the LGR definition keywords CARFIN, or RADIN (or RAFDIN4) and the ENDFIN keyword in the GRID section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

6.3.183 RADFIN – DEFINE A RADIAL LOCAL GRID REFINEMENT WITH ONE COLUMN

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword defines a radial local grid refinement using one columns Local grid refinement is currently not supported by OFM Flow.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

6.3.184 RADFIN4 – DEFINE A RADIAL LOCAL GRID REFINEMENT WITH FOUR COLUMNS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword defines a radial local grid refinement using four columns. Local grid refinement is currently not supported by OFM Flow.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

6.3.185 REFINE – START THE DEFINITION OF A LOCAL GRID REFINEMENT

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The REFINE keyword defines the start of a Cartesian or radial Local Grid Refinement ("LGR") definition that sets the properties of the selected LGR. The keyword is then followed by the property keywords associated with the section where the keyword is being invoked. For example, if the REFINE keyword is used in the GRID section then most of the keywords in that section can be used to set the grid properties for the LGR.

The ENDFIN keyword is used to terminate the LGR definition.

There is no data required for this keyword.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

6.3.186 RESVNUM – DEFINE RESERVOIR COORDINATE DATA SET

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The RESVNUM keyword is used to define the start of a reservoir coordinate data set and stipulates the reservoir number for the data set. The keyword is used in conjunction with the COORD keyword in the GRID section, that specifies a set of coordinate lines or pillars for a reservoir grid via an array. Note that the COORD keyword should immediately follow the RESVNUM keyword.

Although the keyword is processed by OPM Flow, the keyword is effectively ignored as only the default value of one is supported.

No.	Name	Description	Default
1	RESVNUM	A positive integer values that defines the reservoir coordinate data set, or the independent reservoir, for which the subsequent COORD data is to be associated with. RESVNUM should be less than or equal to NUMRES on the NUMRES keyword in the RUNSPEC section. OPM Flow currently only accepts a single data set, that is the default value of one.	1
Notes: 1) The keyword is terminated by a "/".			

Table 6.115: RESVNUM Keyword Description

See the NUMRES keyword in the RUNSPEC section that defines the number of reservoir grids (COORD data sets) that the simulator should process.

The facility is useful to combine two separate reservoir grids into one model in the simulator.

Example

```
--
--       NUMRES
--       NUMBER
RESVNUM
         1                                                                  /
--
-- SPECIFY VERTICAL COORDINATE LINES FOR A REGULAR 3 x 2 GRID
--(DX = 100 and DY = 200)
--
-- X1     Y1    Z1      X2     Y2    Z2
-- ---    ---   ----    ---    ---   ----
COORD
     0      0   1000      0      0   5000
   100      0   1000    100      0   5000
   200      0   1000    200      0   5000
   300      0   1000    300      0   5000
     0    200   1000      0    200   5000
   100    200   1000    100    200   5000
   200    200   1000    200    200   5000
   300    200   1000    300    200   5000
     0    400   1000      0    400   5000
   100    400   1000    100    400   5000
   200    400   1000    200    400   5000
   300    400   1000    300    400   5000
/
--
--       NUMRES
--       NUMBER
RESVNUM
         2                                                                  /
--
-- SPECIFY VERTICAL COORDINATE LINES FOR A REGULAR 3 x 2 GRID
--(DX = 100 and DY = 200)
--
-- X1     Y1    Z1      X2     Y2    Z2
-- ---    ---   ----    ---    ---   ----
COORD
     0      0   1000      0      0   5000
   100      0   1000    100      0   5000
   200      0   1000    200      0   5000
   300      0   1000    300      0   5000
     0    200   1000      0    200   5000
   100    200   1000    100    200   5000
   200    200   1000    200    200   5000
   300    200   1000    300    200   5000
     0    400   1000      0    400   5000
   100    400   1000    100    400   5000
   200    400   1000    200    400   5000
   300    400   1000    300    400   5000
/
```

6.3.187 ROCKFRAC - DEFINE THE ROCK VOLUME TO BULK VOLUME FRACTION FOR ALL THE CELLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

ROCKFRAC defines the rock volume to bulk volume fraction for all the cells, The keyword can be used with all grid types. Rock volume of a grid block is calculated by multiply a cell's bulk volume by it's ROCKFRAC volume. A cell's rock volume is used in the Coal option to calculate the adsorbed gas in the rock (coal), as well as the Thermal and Temp options to calculate the energy is stored in the rock.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	ROCKFRAC	ROCKFRAC is an array of real numbers greater than or equal to zero and less than or equal to one, that are assigned the rock volume to bulk volume fraction values for each cell in the model. The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword. Repeat counts may be used, for example 200*0.850.			
		dimensionless	dimensionless	dimensionless	None
Notes: 1) Setting a cells ROCKFRAC value to zero will make the cell inactive, similar to setting the cells ACTNUM property to zero. 2) The keyword is terminated by a "/".					

Table 6.116: NTG Keyword Description

See also the PORO, PERMX, PERMY, PERMZ and NTG keywords to fully define a grid's properties.

Example

```
--
--       DEFINE GRID ROCKFRAC DATA FOR ALL CELLS (BASED ON NX x NY x NZ = 300)
--
ROCKFRAC
         100*1.000   100*0.850   100*0.500                                    /
```

The above example defines a constant ROCKFRAC of 1.00 for the first 100 cells, then 0.85 for the second 100 hundred cells, and finally 0.500 for the last 100 cell, for the 300 cells in the model as defined by the DIMENS keyword in the RUNSPEC section.

6.3.188 RPTGRID – DEFINE GRID SECTION REPORTING

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword defines the data in the GRID section that is to be printed to the output print file in human readable format. The keyword has two distinct forms, the first of which consists of the keyword followed by a series of integers on the next line indicating the data to be printed (see the first example). This is the original formal in the commercial simulator and was subsequently superseded by the second format. The second format consists of the keyword followed by a series of character strings that indicate the data to be printed. In most cases the character string is the keyword used to load the data in the OPM Flow input deck, for example PORO for the porosity array. Its is anticipated that OPM Flow will eventually support the functionality of the second format only, the first format although recognized will be completely ignored.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

No.	Name	Description	Default
1	ALLNCC	Print all the non-neighbor connections.	N/A
2	COORD	Print the coordinate lines.	N/A
3	COORDYS	Print the coordinate systems.	N/A
4	DEPTH	Print grid cells center depths.	N/A
….		….	N/A

Notes:

1) The keyword is terminated by a "/".

Table 6.117: RPTGRID Keyword Description

Note

This keyword has the potential to produce very large print files that some text editors may have difficulty loading, coupled with the fact that reviewing the data in this format is very cumbersome.

A more efficient solution is to load the *.INIT file into OPM ResInsight to view the data graphically, this also has the benefit of being able to filter the grid based on I, J, K ranges and grid properties.

Examples

The first example shows the original format of this keyword; although the keyword and format are recognized by OPM Flow, the format is ignored and is unlikely to be implemented in in the simulator.

```
--
--       DEFINE GRID SECTION REPORT OPTION (ORIGINAL FORMAT)
-
RPTGRID
         1        2*0      1        3*1                                        /
```

The next example shows the second format of the keyword which may be supported in a future release of OPM Flow.

```
--       DEFINE GRID SECTION REPORT OPTIONS
--
RPTGRID
         DX       DY       DZ       DEPTH    PORO     PERMX                   /
```

6.3.189 RPTGRIDL – DEFINE GRID SECTION REPORTING FOR LGRS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword defines the data in the GRID section that is to be printed to the output print file in human readable format for Local Grid Refinements ("LGRs"), for when LGRs have been activated for the input deck using the LGR keyword in the RUNSPEC section.

The keyword has two distinct forms, the first of which consists of the keyword followed by a series of integers on the next line indicating the data to be printed (see the first example). This is the original formal in the commercial simulator and was subsequently superseded by the second format. The second format consists of the keyword followed by a series of character strings that indicate the data to be printed. In most cases the character string is the keyword used to load the data in the OPM Flow input deck, for example PORO for the porosity array. Its is anticipated that OPM Flow will eventually support the functionality of the second format only, the first format although recognized will be completely ignored.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

No.	Name	Description	Default
1	ALLNCC	Print all the non-neighbor connections.	N/A
2	COORD	Print the coordinate lines.	N/A
3	COORDYS	Print the coordinate systems.	N/A
4	DEPTH	Print grid cells center depths.	N/A
….		….	N/A
24	ALLNNC	ALLNNC is a defined positive integer that specifies the type of Non-Neighbor Connections ("NNC") to be printed, and should be set to one of the follow: 1) To print the NNCs within the LGRs, and the connections between the local and host cells to the print file (*.PRT). 2) To print the NNCs within the LGRs, and the connections between the local and host cells to the print (*.PRT) and debug files (*.DBG). 3) Same as (2) but the data in the debug file (*.DBG) is written out in an alternative format.	N/A
….		….	N/A
57	EXTHOST	EXTHOSTS outputs host cells for Perpendicular Bisector ("PEBI")[128] and [129] LGRs.	
….		….	N/A
Notes:			

[128] *Heinemann, Z.E. and Brand, C.W. 1988. Gridding Techniques in Reservoir Simulation. Proc., First Intl. Forum on Reservoir Simulation, Alpbach, Austria, 339.*

[129] *Heinemann, Z.E., Brand, C.W., Munka, M. et al. 1991. Modeling Reservoir Geometry With Irregular Grids. SPE Res Eng 6 (2): 225–232. SPE-18412-PA. http://dx.doi.org/10.2118/18412-PA*

No.	Name	Description	Default
1)	The keyword is terminated by a "/".		

Table 6.118: RPTGRIDL Keyword Description

Note

This keyword has the potential to produce very large print files that some text editors may have difficulty loading, coupled with the fact that reviewing the data in this format is very cumbersome.

A more efficient solution is to load the *.INIT file into OPM ResInsight to view the data graphically, this also has the benefit of being able to filter the grid based on I, J, K ranges and grid properties.

Examples

The first example shows the original format of this keyword; although the keyword and format are recognized by OPM Flow, the format is ignored and is unlikely to be implemented in in the simulator.

```
--
--       DEFINE LGR GRID SECTION REPORT OPTION (ORIGINAL FORMAT)
-
RPTGRIDL
         1        2*0      1        3*1                                    /
```

The next example shows the second format of the keyword which may be supported in a future release of OPM Flow.

```
--       DEFINE LGR GRID SECTION REPORT OPTIONS
--
RPTGRIDL
         DX       DY       DZ       DEPTH    PORO     PERMX                  /
```

6.3.190 RPTINIT – DEFINE OUTPUT TO THE INIT FILE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword defines the data in the GRID and EDIT sections that is to be written out to the INIT file (*.INIT or *.FINIT). The format consists of the keyword followed by a series of character strings that indicate the data to be written. In most cases the character string is the keyword used to load the data into the OPM Flow input deck, for example PORO for the porosity array in the GRID section. In addition, values either read or calculated by the simulator in the EDIT section can also be written to the INIT file. Again the keyword or property name is used as the mnemonic for the character string, for example the PORV, TRANX keywords etc. If the RPTINIT keyword is not used in the input deck then a default set of data array are written to the file, in this case the actual data written is dependent on the model's configuration and the options being used.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

6.3.191 RPTISOL – ACTIVATE ISOLATED RESERVOIR NUMBER REPORTING

Description

The RPTISOL keyword activates the isolated reservoir report that generates an array of isolated region numbers that is printed in the debug file (*.DBG). The main purpose of this facility is to use the generated array as input to the ISOLNUM keyword in the GRID section in conjunction with the Independent Reservoir Regions option. If the model can be divided into isolated reservoirs then the individual reservoirs may be solved independently, resulting in increased computational efficient, compared with solving the model as a whole.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

There is no data required for this keyword and there is no terminating "/" for this keyword.

Example

```
--
--       ACTIVATE ISOLATED RESERVOIR NUMBER REPORTING
--
RPTISOL
```

The above example activates the isolated reservoir report that generates an array of isolated region numbers to the debug file (*.DBG).

6.3.192 SIGMA – DUAL POROSITY MATRIX TO FRACTURE SIGMA (ALL CELLS)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The SIGMA keyword defines the dual porosity matrix to fracture transmissibility multiplier, sigma, that is applied to all cells, for when the Dual Porosity model has been activated by either the DUALPORO or the DUALPERM keywords in the RUNSPEC section. Sigma (σ) takes into account the matrix-fracture interface area per unit volume and was defined by Kazemi et al[130] to be:

$$\sigma = 4\left(\frac{1}{l_x^2} + \frac{1}{l_y^2} + \frac{1}{l_z^2}\right) \tag{6.15}$$

Where lx, ly and lz are not the grid block dimensions in the model in the respective directions, but the dimensions of the blocks of the matrix material. In practice, σ is used as a tuning parameter in dual porosity runs to match reservoir and well performance.

See also the SIGMAV keyword in the GRID section that supplies the sigma values on an individual cells basis.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

6.3.193 SIGMAGD – DUAL POROSITY MATRIX TO FRACTURE SIGMA FOR GRAVITY DRAINAGE (ALL CELLS)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The SIGMAGD keyword defines the dual porosity matrix to fracture transmissibility multiplier, sigma, that is applied to all cells, for when the Dual Porosity model has been activated by either the DUALPORO or the DUALPERM keywords in the RUNSPEC section. In addition, the GRAVDR keyword in the RUNSPEC section should be used to enable the Gravity Drainage model for the run. Sigma (σ) takes into account the matrix-fracture interface area per unit volume and was defined by Kazemi et al[131] to be:

$$\sigma = 4\left(\frac{1}{l_x^2} + \frac{1}{l_y^2} + \frac{1}{l_z^2}\right) \tag{6.16}$$

Where lx, ly and lz are not the grid block dimensions in the model in the respective directions, but the dimensions of the blocks of the matrix material. In practice, σ is used as a tuning parameter in dual porosity runs to match reservoir and well performance.

Note that SIGMAGD keyword data is used for areas being swept by gas and the SIGMA keyword data is used when the area is being invaded by water. See also the SIGMAGDV keyword in the GRID section that supplies the sigma values on an individual cells basis

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

[130] *Kazemi, H., Merrill JR., L. S., Porterfield, K. L., and Zeman, P. R. "Numerical Simulation of Water-Oil Flow in Naturally Fractured Reservoirs," paper SPE 5719, Society of Petroleum Engineers Journal (1976) 16, No. 6, 317-326.*

[131] *Kazemi, H., Merrill JR., L. S., Porterfield, K. L., and Zeman, P. R. "Numerical Simulation of Water-Oil Flow in Naturally Fractured Reservoirs," paper SPE 5719, Society of Petroleum Engineers Journal (1976) 16, No. 6, 317-326.*

6.3.194 SIGMAGDV – DUAL POROSITY MATRIX TO FRACTURE SIGMA GRAVITY DRAINAGE (INDIVIDUAL CELLS)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The SIGMAGD keyword defines the dual porosity matrix to fracture transmissibility multiplier, sigma, that is applied to individual cells, for when the Dual Porosity model has been activated by either the DUALPORO or the DUALPERM keywords in the RUNSPEC section. In addition, the GRAVDR keyword in the RUNSPEC section should be used to enable the Gravity Drainage model for the run. Sigma (σ) takes into account the matrix-fracture interface area per unit volume and was defined by Kazemi et al[132] to be:

$$\sigma = 4\left(\frac{1}{l_x^2} + \frac{1}{l_y^2} + \frac{1}{l_z^2}\right) \tag{6.17}$$

Where lx, ly and lz are not the grid block dimensions in the model in the respective directions, but the dimensions of the blocks of the matrix material. In practice, σ is used as a tuning parameter in dual porosity runs to match reservoir and well performance.

Note that SIGMAGDV keyword data is used for areas being swept by gas and the SIGMA keyword data is used when the area is being invaded by water. See also the SIGMAGD keyword in the GRID section that supplies a constant sigma value for all cells.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

[132] *Kazemi, H., Merrill JR., L. S., Porterfield, K. L., and Zeman, P. R. "Numerical Simulation of Water-Oil Flow in Naturally Fractured Reservoirs," paper SPE 5719, Society of Petroleum Engineers Journal (1976) 16, No. 6, 317-326.*

6.3.195 SIGMAV – DUAL POROSITY MATRIX TO FRACTURE SIGMA (INDIVIDUAL CELLS)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The SIGMAV keyword defines a dual porosity matrix to fracture multiplier, sigma, that is applied to individual cells, for when the Dual Porosity model has been invoked by either the DUALPORO or the DUALPERM keywords in the RUNSPEC section. Sigma (σ) takes into account the matrix-fracture interface area per unit volume and was defined by Kazemi et al[133] to be:

$$\sigma = 4\left(\frac{1}{l_x^2} + \frac{1}{l_y^2} + \frac{1}{l_z^2}\right) \tag{6.18}$$

Where lx, ly and lz are not the grid block dimensions in the model in the respective directions, but the dimensions of the blocks of the matrix material. In practice, σ is used as a tuning parameter in dual porosity runs to match reservoir and well performance.

See also the SIGMA keyword in the GRID section that supplies a constant sigma to all cells.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

[133] *Kazemi, H., Merrill JR., L. S., Porterfield, K. L., and Zeman, P. R. "Numerical Simulation of Water-Oil Flow in Naturally Fractured Reservoirs," paper SPE 5719, Society of Petroleum Engineers Journal (1976) 16, No. 6, 317-326.*

6.3.196 SKIP – ACTIVATE SKIPPING OF ALL KEYWORDS AND INPUT DATA

The SKIP keyword turns on skipping of keywords until the ENDSKIP activation keyword is encountered. All keywords and between the SKIP and ENDSKIP keywords are ignored by OPM Flow.

See *SKIP – Activate Skipping of All Keywords and Input Data* in the GLOBAL section for a full description.

6.3.197 SKIP100 – ACTIVATE SKIPPING OF BLACK-OIL KEYWORDS AND INPUT DATA

This keyword turns on skipping of black-oil keywords until the ENDSKIP activation keyword is encountered. All black-oil keywords between the SKIP100 and ENDSKIP keywords are ignored by OPM Flow.

See *SKIP100 – Activate Skipping of Black-Oil Keywords and Input Data* in the GLOBAL section for a full description.

6.3.198 SKIP300 – ACTIVATE SKIPPING OF "COMPOSITIONAL" KEYWORDS AND INPUT DATA

Turns on skipping of "compositional" keywords until the ENDSKIP activation keyword is encountered. All "compositional" keywords between the SKIP300 and ENDSKIP keywords are ignored by OPM Flow.

See *Error: Reference source not found* in the GLOBAL section for a full description.

6.3.199 SMULTX - MULTIPLY CELL TRANSMISSIBILITY IN THE +X DIRECTION (AUTO-REFINEMENT)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

SMULTX multiples the transmissibility between two cell faces in the +X direction between cells in a host base grid and the connecting auto-refined grid cells, via an array, that is the keyword sets the transmissibility multiplier of block (Ihost, Jhost, Khost) in the host base grid, multiplies the transmissibility all the cells (Iauto, Jauto, Kauto) and (I+Iauto, Jauto, Kauto) in the auto-refinement grid. The Auto Refinement option must be enabled to use this keyword via the AUTOREF keyword in the RUNSPEC section.

An alternative to defining the complete array is to use the BOX keyword to define an area of the grid and then use the SMULTX keyword to set the multipliers just for the area defined by the BOX keyword (see the example).

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description	Default
1	SMULTX+	SMULTX+ is an array of real positive numbers assigning the transmissibility multipliers in the +X direction to each cell face in the model. Repeat counts may be used, for example 20*100.0.	1.0

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.

2) The keyword is terminated by a "/".

Table 6.119: SMULTX Keyword Description

See also the MULTX, MULTY and MULTZ keywords for scaling transmissible between grid cells.

Example

```
--
--       DEFINE INPUT BOX FOR EDITING INPUT ARRAYS
--
--       ---------- BOX ---------
--       I1  I2   J1  J2   K1  K2
BOX
         10  10   1   6    1   3                          / DEFINE BOX AREA
--
--       SET SMULTX+ TRANSMISSIBILITY MULTIPLIERS
--
SMULTX
         18*0.300                                        /
--
--       DEFINE END OF INPUT BOX EDITING OF INPUT ARRAYS
--
ENDBOX
```

The above example defines a 0.3 scaling multiplier for the 18 cells defined by the preceding BOX statement. The ENDBOX keyword resets the input box to the full grid.

6.3.200 SMULTY - MULTIPLY CELL TRANSMISSIBILITY IN THE +Y DIRECTION (AUTO-REFINEMENT)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

SMULTY multiples the transmissibility between two cell faces in the +Y direction between cells in a host base grid and the connecting auto-refined grid cells, via an array, that is the keyword sets the transmissibility multiplier of block (Ihost, Jhost, Khost) in the host base grid, multiplies the transmissibility all the cells (Iauto, Jauto, Kauto) and (Iauto, J+1auto, Kauto) in the auto-refinement grid. The Auto Refinement option must be enabled to use this keyword via the AUTOREF keyword in the RUNSPEC section.

An alternative to defining the complete array is to use the BOX keyword to define an area of the grid and then use the SMULTY keyword to set the multipliers just for the area defined by the BOX keyword (see the example).

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description	Default
1	SMULTY+	SMULTY+ is an array of real positive numbers assigning the transmissibility multipliers in the +X direction to each cell face in the model. Repeat counts may be used, for example 20*100.0.	1.0

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.

2) The keyword is terminated by a "/".

Table 6.120: SMULTY Keyword Description

See also the MULTX, MULTY and MULTZ keywords for scaling transmissible between grid cells.

Example

```
--
--       DEFINE INPUT BOX FOR EDITING INPUT ARRAYS
--
--       ---------- BOX ---------
--       I1  I2   J1  J2   K1  K2
BOX
         10  10   1   6    1   3                          / DEFINE BOX AREA
--
--       SET SMULTY+ TRANSMISSIBILITY MULTIPLIERS
--
SMULTY
         18*0.300                                        /
--
--       DEFINE END OF INPUT BOX EDITING OF INPUT ARRAYS
--
ENDBOX
```

The above example defines a 0.3 scaling multiplier for the 18 cells defined by the preceding BOX statement. The ENDBOX keyword resets the input box to the full grid.

6.3.201 SMULTZ - MULTIPLY CELL TRANSMISSIBILITY IN THE +Z DIRECTION (AUTO-REFINEMENT)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

SMULTZ multiples the transmissibility between two cell faces in the +Z direction between cells in a host base grid and the connecting auto-refined grid cells, via an array, that is the keyword sets the transmissibility multiplier of block (Ihost, Jhost, Khost) in the host base grid, multiplies the transmissibility all the cells (Iauto, Jauto, Kauto) and (Iauto, Jauto, K+1auto) in the auto-refinement grid. The Auto Refinement option must be enabled to use this keyword via the AUTOREF keyword in the RUNSPEC section.

An alternative to defining the complete array is to use the BOX keyword to define an area of the grid and then use the SMULTZ keyword to set the multipliers just for the area defined by the BOX keyword (see the example).

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description	Default
1	SMULTZ+	SMULTZ+ is an array of real positive numbers assigning the transmissibility multipliers in the +X direction to each cell face in the model. Repeat counts may be used, for example 20*100.0.	1.0

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.

2) The keyword is terminated by a "/".

Table 6.121: SMULTX Keyword Description

See also the MULTX, MULTY and MULTZ keywords for scaling transmissible between grid cells.

Example

```
--
--       DEFINE INPUT BOX FOR EDITING INPUT ARRAYS
--
--       ---------- BOX ---------
--       I1  I2   J1  J2   K1  K2
BOX
         10  10   1   6    1   3                          / DEFINE BOX AREA
--
--       SET SMULTZ+ TRANSMISSIBILITY MULTIPLIERS
--
SMULTZ
         18*0.300                                        /
--
--       DEFINE END OF INPUT BOX EDITING OF INPUT ARRAYS
--
ENDBOX
```

The above example defines a 0.3 scaling multiplier for the 18 cells defined by the preceding BOX statement. The ENDBOX keyword resets the input box to the full grid.

6.3.202 SOLVDIRS – DEFINE LINEAR SOLVER PRINCIPAL DIRECTIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The SOLVDIRS keyword defines the linear solver principal directions, which should be set to XY, XZ, YX, YX, ZX, or ZY. The default direction is based on the direction of the highest transmissibility and SOLVDIRS allows for over writing the default direction for when linear convergence of the equations are problematic.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

See section *2.2 Running OPM Flow 2023-10 From The Command Line* on how to invoke various numerical schemes via the OPM Flow command line interface.

6.3.203 SOLVNUM – DEFINE PEBI GRID CORRESPONDENCE TO SOLVER ORDER

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The SOLVNUM defines the unstructured Perpendicular Bisector ("PEBI")[134] and [135] grid correspondence to the nested factorization solver order, for when the grid has been entered as a PEBI list. This keyword is generated by an external pre-processing program for generating simulation grids.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

[134] *Heinemann, Z.E. and Brand, C.W. 1988. Gridding Techniques in Reservoir Simulation. Proc., First Intl. Forum on Reservoir Simulation, Alpbach, Austria, 339.*

[135] *Heinemann, Z.E., Brand, C.W., Munka, M. et al. 1991. Modeling Reservoir Geometry With Irregular Grids. SPE Res Eng 6 (2): 225–232. SPE-18412-PA. http://dx.doi.org/10.2118/18412-PA*

6.3.204 SPECGRID- DEFINE THE DIMENSIONS OF A CORNER-POINT GRID

Description

SPECGRID defines the dimensions of corner-point and radial grids in the x, y, and z directions as well as the number of reservoirs, where each reservoir has its own set of corner-point geometry data.

The keyword can only be used with Irregular Corner-Point Grids and Radial Grids.

No.	Name	Description	Default
1	NDIVIX	A positive integer value that defines the number of cells in the X or R direction	1
2	NDIVIY	A positive integer value that defines the number of cells in the Y or THETA direction	1
3	NDIVZ	A positive integer value that defines the number of cells in the Z direction	1
4	NUMRES	A positive integer values that defines number of coordinate data sets, or independent reservoirs in the model. OPM Flow currently only accepts a single data set, that is the default value of one.	1
5	TYPE	A character string set to either T of F that defines the type of grid to be defined by subsequent keywords: 1) T = Radial grid with radial coordinates 2) F = Cartesian grid Only the default option F is supported by OPM Flow.	F
Notes: 1) The dimensions are also entered on the DIMENS keyword in the RUNSPEC section and the two sets of numbers should be consistent. 2) The keyword is terminated by a "/".			

Table 6.122: SPECGRID Keyword Description

See also the COORD, COORDSYS and ZCORN keywords to fully define an Irregular Corner-Point Grid.

Example

```
--
--        MAX     MAX     MAX     MAX     GRID
--        NDIVIX  NDIVIY  NDIVIZ  NUMRES  TYPE
SPECGRID
          46      112     22      1       F                                  /
```

The above example defines a 46 x 112 x 22 grid with one set of irregular corner-point data.

6.3.205 THCGAS – DEFINE GAS PHASE THERMAL CONDUCTIVITY FOR ALL CELLS

Description

The THCGAS keyword defines the gas phase thermal conductivity for when the thermal calculation is activated by the THERMAL keyword in the RUNSPEC section, and should be used in conjunction with THCROCK keyword in the GRID section.

This keyword can only be used if the thermal option has been activated by the THERMAL keyword in the RUNSPEC section. Note this is different to the commercial simulator that uses the TEMP keyword in the RUNSPEC section to activate the black-oil thermal model.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	THCGAS	THCGAS is an array of real positive numbers that define the thermal conductivity of the gas phase in each grid block. Repeat counts may be used, for example 3000*20.0			
		Btu/ft/day/°R	kJ/m/day/K	J/cm/hr/K	None
Notes: 1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement. 2) The keyword is terminated by a "/".					

Table 6.123: THCGAS Keyword Description

Note that there two ways to define the rock and in situ fluids thermal conductivity:

1) Either by using the THCONR keyword to define the combined rock and fluid conductivity, and optionally the THCONSF keyword in the GRID section, or
2) by specifying the rock and fluid conductivities individually using the THCROCK, THCOIL, THCGAS, and THCWATER keywords in the GRID section.

Hence, the THCROCK and THCONR keywords are mutually exclusive.

Here, the THCGAS keyword is used in conjunction with the other thermal conductivity arrays to calculate the porosity weighted thermal conductivity of a grid block using:

$$\text{Average Thermal Conductivity} = \frac{PORO \times (THCOIL + THCGAS + THCWATER + THCSOLID)}{\text{NUMBER OF PHASES IN THE MODEL}} \times (1 - PORO) \times THCROCK \quad (6.19)$$

See also the THCOIL, and THCWATER, and THCROCK keywords in the GRID section. The commercial compositional simulator's THCSOLID keyword is not supported or required by OPM Flow.

Example

```
--
--       DEFINE GRID BLOCK GAS PHASE THERMAL CONDUCTIVITY
-        FOR ALL CELLS (BASED ON NX x NY x NZ = 300)
--
THCGAS
         300*20.0                                                    /
```

The above example defines the gas phase thermal conductivity of 20.0 for each cell in the 300 grid block model as defined by the DIMENS keyword in the RUNSPEC section.

6.3.206 THCOIL – DEFINE OIL PHASE THERMAL CONDUCTIVITY FOR ALL CELLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The THCOIL keyword defines the oil phase thermal conductivity for when the thermal calculation is activated by the THERMAL keyword in the RUNSPEC section, and should be used in conjunction with THCROCK keyword in the GRID section.

This keyword can only be used if the thermal option has been activated by the THERMAL keyword in the RUNSPEC section. Note this is different to the commercial simulator that uses the TEMP keyword in the RUNSPEC section to activate the black-oil thermal model.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	THCOIL	THCOIL is an array of real positive numbers that define the thermal conductivity of the oil phase in each grid block. Repeat counts may be used, for example 3000*20.0			
		Btu/ft/day/°R	kJ/m/day/K	J/cm/hr/K	None

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) The keyword is terminated by a "/".

Table 6.124: THCOIL Keyword Description

Note that there two ways to define the rock and in situ fluids thermal conductivity:

1) Either by using the THCONR keyword to define the combined rock and fluid conductivity, and optionally the THCONSF keyword in the GRID section, or
2) by specifying the rock and fluid conductivities individually using the THCROCK, THCOIL, THCGAS, and THCWATER keywords in the GRID section.

Hence, the THCROCK and THCONR keywords are mutually exclusive.

Here, the THCOIL keyword is used in conjunction with the other thermal conductivity arrays to calculate the porosity weighted thermal conductivity of a grid block using:

$$\text{Average Thermal Conductivity} = \frac{PORO \times (THCOIL + THCGAS + THCWATER + THCSOLID)}{\text{NUMBER OF PHASES IN THE MODEL}} \times (1 - \text{PORO}) \times \text{THCROCK} \quad (6.20)$$

See also the THCGAS, and THCWATER, and THCROCK keywords in the GRID section. The commercial compositional simulator's THCSOLID keyword is not supported or required by OPM Flow.

Example

```
--
--       DEFINE GRID BLOCK OIL PHASE THERMAL CONDUCTIVITY
-        FOR ALL CELLS (BASED ON NX x NY x NZ = 300)
--
THCOIL
         300*20.0                                                    /
```

The above example defines the oil phase thermal conductivity of 20.0 for each cell in the 300 grid block model, as defined by the DIMENS keyword in the RUNSPEC section.

6.3.207 THCONR – DEFINE ROCK AND FLUID THERMAL CONDUCTIVITY FOR ALL CELLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The THCONR keyword defines the reservoir rock plus fluid thermal conductivity for all cells for when the thermal calculation is activated by the THERMAL keywords in the RUNSPEC section.

This keyword can only be used if the thermal option has been activated by the THERMAL keyword in the RUNSPEC section. Note this is different to the commercial simulator that uses the TEMP keyword in the RUNSPEC section to activate the black-oil thermal model.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	THCONR	THCONR is an array of real positive numbers that define the combined rock and fluid conductivity of a grid block. Repeat counts may be used, for example 3000*25.0			
		Btu/ft/day/°R	kJ/m/day/K	J/cm/hr/K	None
Notes: 1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement. 2) The keyword is terminated by a "/".					

Table 6.125: THCONR Keyword Description

Note that there two ways to define the rock and in situ fluids thermal conductivity:

1) Either by using the THCONR keyword to define the combined rock and fluid conductivity, and optionally the THCONSF keyword in the GRID section, or
2) by specifying the rock and fluid conductivities individually using the THCROCK, THCOIL, THCGAS, and THCWATER keywords in the GRID section.

Hence, the THCROCK and THCONR keywords are mutually exclusive.

Example

```
--
--       DEFINE GRID BLOCK ROCK-FLUID THERMAL CONDUCTIVITY
--       FOR ALL CELLS (BASED ON NX x NY x NZ = 300)
--
THCONR
         300*25.0                                                    /
```

The above example defines the combined rock and fluid thermal conductivity of 25.0 for each cell in the 300 grid block model, as defined by the DIMENS keyword in the RUNSPEC section.

6.3.208 THCONSF – DEFINE GAS SATURATION DEPENDENT THERMAL CONDUCTIVITY SCALING FACTOR FOR ALL CELLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The THCONSF keyword defines a gas saturation dependent scaling factor to the fluid and reservoir rock thermal conductivities entered via the THCONR keyword in the GRID section, for when the thermal calculation is activated by the THERMAL keyword in the RUNSPEC.

This keyword can only be used if the thermal option has been activated by the THERMAL keyword in the RUNSPEC section. Note this is different to the commercial simulator that uses the TEMP keyword in the RUNSPEC section to activate the black-oil thermal model.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	THCONSF	THCONSF is an array of real positive numbers, greater than zero and less than or equal to one, that define the gas saturation dependent scaling factor that is applied to the THCONR data, entered via the THCONR keyword, to adjust the thermal conductivity of the reservoir cells in each grid block. Repeat counts may be used, for example 3000*0.15			
		dimensionless	dimensionless	dimensionless	None

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.

2) The keyword is terminated by a "/".

Table 6.126: THCROCK Keyword Description

Note that there two ways to define the rock and in situ fluids thermal conductivity:

1) Either by using the THCONR keyword to define the combined rock and fluid conductivity, and optionally the THCONSF keyword in the GRID section, or

2) by specifying the rock and fluid conductivities individually using the THCROCK, THCOIL, THCGAS, and THCWATER keywords in the GRID section.

Hence, the THCROCK and THCONR keywords are mutually exclusive.

Here, the THCONSF keyword defines a scaling factor which is a function of the gas saturation that scales a cells total thermal conductivity (reservoir fluids plus reservoir rock) entered via the THCONR keyword in the GRID section. This combination of keywords, THCONSF and THCONR implies that the oil and water phase thermal conductivities are saturation independent with respect to the liquid phase, and that only the gas saturation influences a cell's thermal conductivity as entered via the THCONR keyword.

Thus, THCONSF scales the THCONR values via a multiplier Ω, by:

$$\Omega_{i,j,k} = \left(1 - \text{THCONSF x Gas Saturation}\right)_{i,j,k} \quad \textbf{(6.21)}$$

See also the THCGAS, THCOIL, THCWATER and THROCK keywords in the GRID section, for an alternative way to enter the thermal conductivity properties. However, the THCONSF keyword cannot be used with the THCGAS, THCOIL, THCWATER and THCROCK keywords. Secondly, the commercial compositional simulator's THCSOLID keyword is not supported or required by OPM Flow.

Example

```
--
--       DEFINE GRID SGAS DEPENDENT SCALING FACTOR FOR THE THCONR ARRAY
--       FOR ALL CELLS (BASED ON NX x NY x NZ = 300)
--       (OPM FLOW THERMAL OPTION ONLY)
--
THCONSF
         300*0.12                                                              /
```

The above example defines the gas saturation thermal conductivity scaling factor to be applied to the THCONR to be 0.12 for all 300 cells in the model, as defined by the DIMENS keyword in the RUNSPEC section.

6.3.209 THCROCK – DEFINE RESERVOIR ROCK THERMAL CONDUCTIVITY FOR ALL CELLS

Description

The THCROCK keyword defines the reservoir rock thermal conductivity for when the thermal calculation is activated by the THERMAL keyword in the RUNSPEC section.

This keyword can only be used if the thermal option has been activated by the THERMAL keyword in the RUNSPEC section. Note this is different to the commercial simulator that uses the TEMP keyword in the RUNSPEC section to activate the black-oil thermal model.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	THCROCK	THCROCK is an array of real positive numbers that define the thermal conductivity of the reservoir rock in each grid block. Repeat counts may be used, for example 3000*20.0			
		Btu/ft/day/°R	kJ/m/day/K	J/cm/hr/K	None
Notes: 1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement. 2) The keyword is terminated by a "/".					

Table 6.127: THCROCK Keyword Description

Note that there two ways to define the rock and in situ fluids thermal conductivity:

1) Either by using the THCONR keyword to define the combined rock and fluid conductivity, and optionally the THCONSF keyword in the GRID section, or
2) by specifying the rock and fluid conductivities individually using the THCROCK, THCOIL, THCGAS, and THCWATER keywords in the GRID section.

Hence, the THCROCK and THCONR keywords are mutually exclusive.

Here, the THCROCK keyword is used in conjunction with the other thermal conductivity arrays to calculate the porosity weighted thermal conductivity of a grid block using:

$$\text{Average Thermal Conductivity} = \frac{PORO \times (THCOIL + THCGAS + THCWATER + THCSOLID)}{\text{NUMBER OF PHASES IN THE MODEL}} \times (1 - PORO) \times THCROCK \quad (6.22)$$

See also the THCOIL, and THCGAS, and THCWATER keywords in the GRID section. The commercial compositional simulator's THCSOLID keyword is not supported or required by OPM Flow.

Example

```
--
--       DEFINE GRID BLOCK RESERVOIR ROCK THERMAL CONDUCTIVITY
-        FOR ALL CELLS (BASED ON NX x NY x NZ = 300)
--
THCROCK
         300*20.0                                                    /
```

The above example defines the reservoir rock thermal conductivity of 20.0 for each cell in the 300 grid block model, as defined by the DIMENS keyword in the RUNSPEC section.

6.3.210 THCWATER – DEFINE WATER PHASE THERMAL CONDUCTIVITY FOR ALL CELLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The THCWATER keyword defines the water phase thermal conductivity for when the thermal calculation is activated by the THERMAL keyword in the RUNSPEC section, and should be used in conjunction with THCROCK keyword in the GRID section.

This keyword can only be used if the thermal option has been activated by the THERMAL keyword in the RUNSPEC section. Note this is different to the commercial simulator that uses the TEMP keyword in the RUNSPEC section to activate the black-oil thermal model.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	THCWATER	THCWATER is an array of real positive numbers that define the thermal conductivity of the water phase in each grid block. Repeat counts may be used, for example 3000*20.0			
		Btu/ft/day/°R	kJ/m/day/K	J/cm/hr/K	None

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.

2) The keyword is terminated by a "/".

Table 6.128: THCWATER Keyword Description

Note that there two ways to define the rock and in situ fluids thermal conductivity:

1) Either by using the THCONR keyword to define the combined rock and fluid conductivity, and optionally the THCONSF keyword in the GRID section, or

2) by specifying the rock and fluid conductivities individually using the THCROCK, THCOIL, THCGAS, and THCWATER keywords in the GRID section.

Hence, the THCROCK and THCONR keywords are mutually exclusive.

Here, the THCWATER keyword is used in conjunction with the other thermal conductivity arrays to calculate the porosity weighted thermal conductivity of a grid block using:

$$\text{Average Thermal Conductivity} = \frac{PORO \times (THCOIL + THCGAS + THCWATER + THCSOLID)}{\text{NUMBER OF PHASES IN THE MODEL}} \times (1 - PORO) \times THCROCK \quad (6.23)$$

See also the THCGAS, and THCOIL, and THCROCK keywords in the GRID section. The commercial compositional simulator's THCSOLID keyword is not supported or required by OPM Flow

Example

```
--
--       DEFINE GRID BLOCK WATER PHASE THERMAL CONDUCTIVITY
-        FOR ALL CELLS (BASED ON NX x NY x NZ = 300)
--
THCWATER
         300*20.0                                                    /
```

The above example defines the water phase thermal conductivity of 20.0 for each cell in the 300 grid block model, as defined by the DIMENS keyword in the RUNSPEC section.

6.3.211 THPRESFT - DEFINE FAULT THRESHOLD PRESSURES

Description

The THPRESFT keyword defines a fault threshold pressures that prevents fluid flow from occurring across the fault plane until the threshold pressure is exceeded, for when the threshold pressure option has been activated via the THRPRES variable on the EQLOPTS keyword in the RUNSPEC section.

Each row entry in the THPRESFT keyword defines a fault threshold pressure.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	FLTNAME	FLTNAME is a character string enclosed in quotes with a maximum length of eight characters, that defines the name of the fault. FLTNAME must have been previously defined using the FAULTS keyword in the GROD section, otherwise an error will occur.			None
2	PRESS	PRESS is a single positive real value that defines the threshold pressure for the fault (FLTNAME). If PRESS is defaulted then the simulator will set the threshold pressure to zero, that is the fault is open to flow along the fault plane.			
		psia	barsa	atma	0
Notes: 1) If there are multiple entries for FLTNAME only the last entry is applied. 2) Each record is terminated by a "/" and the keyword should be terminated by a "/".					

Table 6.129: THPRESFT Keyword Description

See also the FAULTS keyword in the GRID section that is used to created named faults. Normally this fault data is automatically generated by pre-processing software that generates the static model.

Example

The example below defines two fault traces, 'M_WEST' and 'BC' fault having threshold pressures of 1000.0 and 2000 psis respectively.

```
--
--        DEFINE FAULTS IN THE GRID GEOMETRY
--
--        FAULT        ------------ FAULT TRACE -------------
--        NAME          I1   I2    J1   J2    K1   K2    FACE
FAULTS
          'M_WEST'       5    5     3    3     1    22   'X'   /
          'M_WEST'       5    5     4    4     1    22   'X'   /
          'M_WEST'       5    5     5    5     1    22   'X'   /
……..

          'BC'          43   43     8    8     1    22   'Y'   /
          'BC'          42   42     9    9     1    22   'X'   /
          'BC'          44   44     8    8     1    22   'Y'   /
……..
/
--
--        DEFINE FAULT THRESHOLD PRESSURES
--
--        FAULT         THRESHOLD
--        NAME          PRESSURE
THPRESFT
          'M_WEST'      1000.0                                  /
          'BC'          1200.0                                  /
/
```

6.3.212 TOPS - DEFINE THE DEPTH AT THE CENTER OF THE TOP FACE FOR EACH CELL

Description

TOPS defines the depth of the top face of each cell in the model.

It can only be used with the Cartesian Regular Grid or Radial Grid models.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	TOPS	TOPS is an array of real numbers defining the depth at the top face of each cell in the model. One can either just enter the TOPS for the first layer only based on NX x NY entries and OPM Flow will calculate the remaining TOPS based on either DZ or DZV. Alternatively NX x NY x NZ TOPS may be entered for each cell in the model. See the DIMENS keyword in the RUNSPEC section for the definition of NX, NY and NZ. Repeat counts may be used, for example 10*5201.0.			
		feet	m	cm	None
Notes: 1) The keyword is terminated by a "/".					

Table 6.130: TOPS Keyword Description

See also the DEPTHS keyword to define the structural depth for the cells.

Examples

The example below defines the TOPS of the cells for each cell for NX = 5, NY = 5 and NZ = 3 model, as well as the X and Y direction cells sizes.

```
--
--       DEFINE GRID BLOCK TOPS FOR THE TOP LAYER (NX=5, NY=5, and NZ=3)
--
TOPS
         25*3100  25*3105  25*3110                                          /
--
--       DEFINE GRID BLOCK X DIRECTION CELL SIZE (BASED ON NX = 5)
--
DXV
         5*100                                                            /
--
--       DEFINE GRID BLOCK X DIRECTION CELL SIZE (BASED ON NY = 5)
--
DYV
         5*100                                                            /
```

A second example is shown on the following page.

This example defines the same grid as before but with the TOPS keyword only defining the top layer and DZV keyword defining the cells thickness.

```
--
--       DEFINE GRID BLOCK TOPS FOR THE TOP LAYER (NX = 5, NY = 5, NZ = 3)
--
TOPS
         25*3100                                                              /
--
--       DEFINE GRID BLOCK Z DIRECTION CELL SIZE (BASED ON NZ = 3)
--
DZV
         3*5.0                                                                /
--
--       DEFINE GRID BLOCK X DIRECTION CELL SIZE (BASED ON NX = 5)
--
DXV
         5*100                                                                /
--
--       DEFINE GRID BLOCK Y DIRECTION CELL SIZE (BASED ON NY = 5)
--
DYV
         5*100                                                                /
```

6.3.213 TRANGL – DEFINE NON-NEIGHBOR CONNECTIONS BETWEEN GLOBAL AND LGR CELLS MANUALLY

Description

TRANGL enables Non-Neighbor Connections ("NNC") between the global cells and the Local Grid Refinement ("LGR") cells to be manually specified, as oppose to the simulator calculating the transmissibilities. The LGR keyword in the RUNSPEC section should be utilized to define the presence of LGRs in the model and to define various LGR dimension parameters.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	I1	A positive integer that defines the LGR grid block in the I-direction in a non-neighbor connection, must be greater than or equal to one and less than or equal to NX on the CARFIN keyword in the GRID section.			None
2	J1	A positive integer that defines the LGR grid block in the J-direction in a non-neighbor connection, must be greater than or equal to one and less than or equal to NY on the CARFIN keyword in the GRID section.			None
3	K1	A positive integer that defines the LGR grid block in the K-direction in a non-neighbor connection, must be greater than or equal to one and less than or equal to NZ on the CARFIN keyword in the GRID section.			None
4	I2	A positive integer that defines the GLOBAL grid block in the I-direction in a non-neighbor connection, must be greater than or equal to one and less than or equal to NX on the DIMENS keyword in the RUNSPEC section.			None
5	J2	A positive integer that defines the GLOBAL grid block in the J-direction in a non-neighbor connection, must be greater than or equal to one and less than or equal to NY on the DIMENS keyword in the RUNSPEC section.			None
6	K2	A positive integer that defines the GLOBAL grid block in the K-direction in a non-neighbor connection, must be greater than or equal to one and less than or equal to NZ on the DIMENS keyword in the RUNSPEC section.			None
7	TRANSNNC	TRANSNNC is a positive real number greater than or equal to zero that defines the transmissibility between the GLOBAL grid block (I1, J1, K1) and the LGR grid block (I2, J2, K2). The default value of zero sets the transmissibility between the two cells to zero.			
		cP.rb/day/psia	cP.rm^3/day/bars	cP.rcc/hr/atm	0.0

Notes:

1) Each record must be terminated by a "/" and the keyword is terminated by a "/".

Table 6.131: TRANGL Keyword Description

Example

```
--
--       MANUALLY DEFINE LGR-GLOBAL GRID NON-NEIGHBOR CONNECTIONS
--
--       ----LGR-----   ---GLOBAL----   -- TRANSNCC --
--       I1   J1   K1    I2   J2   K2
TRANGL
         1    1    1     1    1    2        0.2500     /
         1    1    2     1    1    3        0.2500     /
         1    1    3     1    1    4        0.2500     /
/
```

The above example defines the transmissibility between LGR cell (1, 1, 1) and global cell (1, 1, 2), LGR cell (1, 1, 2) and global cell (1, 1, 3) and finally between LGR cell (1, 1, 3) and global cell (1, 1, 4) to be 0.2500.

6.3.214 USEFLUX – ACTIVATE FLUX BOUNDARY MODEL AND DEFINE FLUX FILE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The USEFLUX keyword activates the Flux Boundary model and defines the name of the FLUX file. Only grid blocks that have been declared by the FLUXREG keyword in the GRID section to be in an active flux region, are active for the run.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

6.3.215 USENOFLO – ACTIVATE FLUX BOUNDARY MODEL WITHOUT A FLUX FILE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The USENOFLUX keyword activates the Flux Boundary model without a FLUX file. The USEFLUX keyword should still be in the input deck, but in this case the FLUX filename is ignored. The option is useful when the no-flow boundary condition is a reasonable assumption and avoids the pre-cursor run used to generate the FLUX file via the DUMPFLUX keyword in the GRID section. Only grid blocks that have been declared by the FLUXREG keyword in the GRID section to be in an active flux region, are active for the run.

There is no data required for this keyword and there is no terminating "/" for this keyword.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

Example

```
--
--       ACTIVATE FLUX BOUNDARY MODEL WITHOUT A FLUX FILE
--
USEFLUX
/

USENOFLO
```

The above example activates the Flux Boundary model without a FLUX file.

6.3.216 VEDEBUG – VERTICAL EQUILIBRIUM DEBUG DATA OUTPUT

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword defines the debug Vertical Equilibrium ("VE") data to be written to the debug file (*.DBG), for when the VE model has been activated by the VE keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

6.3.217 VEFIN – ACTIVATE VERTICAL EQUILIBRIUM MODEL (LGR)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

If the VE keyword in the RUNSPEC section has been used to activate the Vertical Equilibrium ("VE") model for the global grid, then the VEFIN keyword may used to set various options for the Local Grid Refinements ("LGR"). The LGR keyword in the RUNSPEC section should be activated to indicate the presence of LGRs and the keyword VEFIN should be placed in between the CARFIN and ENDFIN keywords in the GRID section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

6.3.218 WARN – ACTIVATE WARNING MESSAGES

Turns on warning messages to be printed to the print file (*.PRT); note that this keyword is activated by default and can subsequently be switched off by the NOWARN activation keyword. The warning messages may be turned on and off using keywords WARN and NOWARN. OPM Flow always prints error messages.

See *WARN – Activate Warning Messages* in the GLOBAL section for a full description.

6.3.219 ZCORN – DEFINE THE DEPTH OF EACH CORNER-POINT OF A GRID BLOCK

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

ZCORN defines the depth of each corner point of a grid block on the pillars defining the reservoir grid. A total of 8 x NX x NY x NZ values are needed to fully define all the depths in the model. The depths specifying the top of the first layer are entered first with one point for each pillar for each grid block. The points are entered with the X axis cycling fastest. Next come the depths of the bottom of the first layer. The top of layer two follows etc.

The keyword can be only used be used with Irregular Corner-Point Grids.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	ZCORN	An array of depths with 8 depths for each cell, for a total of 8 x Nx x NY x NZ entries			
		feet	metres	cm	None

Notes:

1) Regular Cartesian and Radial grid keywords cannot be used with this keyword, that is: DR, DRV, DTHETA, DTHETAV, DX, DXV, DY, DYV, DZ, INRAD, and TOPS.

2) The keyword is terminated by a "/".

Table 6.132: ZCORN Keyword Description

See also the SPECGRID, COORD and COORDSYS keywords to fully define an Irregular Corner-Point Grid.

Example

```
--
--       SPECIFY CORNER-POINT DEPTHS FOR A 3 x 2 x 2 GRID,
--       WITH CONSTANT SLOPE IN THE X AND Y DIRECTIONS
--       SUCH THAT ALL CORNER POINTS OF NEIGHBOURING BLOCKS ALIGN
ZCORN
--
--                    top of layer 1
--
         1450   1500   1500   1550   1550   1600
         1500   1550   1550   1600   1600   1650
         1500   1550   1550   1600   1600   1650
         1550   1600   1600   1650   1650   1700
--
--                     bottom of layer 1
--
         1460   1510   1510   1560   1560   1610
         1510   1560   1560   1610   1610   1660
         1510   1560   1560   1610   1610   1660
         1560   1610   1610   1660   1660   1710
--
--                     top of layer 2
--
         1460   1510   1510   1560   1560   1610
         1510   1560   1560   1610   1610   1660
```

```
          1510   1560   1560   1610   1610   1660
          1560   1610   1610   1660   1660   1710
--
--                    bottom of layer 2
--
          1470   1520   1520   1570   1570   1620
          1520   1570   1570   1620   1620   1670
          1520   1570   1570   1620   1620   1670
          1570   1620   1620   1670   1670   1720
/
```

The above example defines depths of the vertical coordinate lines for a regular 3 by 2 by 2 grid with a constant slope in the x and y directions such that all the corner points of neighboring blocks are aligned.

CHAPTER 7: EDIT SECTION

7.1 INTRODUCTION

This section enables user defined changes to be applied after OPM Flow has processed the data in the GRID section, that is the resulting pore volume (PORV) and transmissibility arrays (TRANX, TRANY and TRANX). The entered primary static arrays (PORO, PERMX etc.) in the GRID section are no longer available and all modifications are applied to the pore volume and transmissibility arrays.

Historically the intention of this section was allow for the editing of the processed data; however, the features available in this section have, through time, migrated to the GRID section. For example the array operator keywords like ADD, COPY, MULTIPLY, etc. are available in the GRID section and thus enabling editing of the primary static arrays (PORO, PERMX etc.). This increased in capability in the GRID section has therefore made the EDIT section some what redundant.

Nevertheless the section is sometimes used by users to incorporate history matching parameter changes and by static earth modeling software packages to import directly the pore volumes and transmissibilities calculated in the static model directly into the numerical model via the EDIT section. Although the latter work flow is not very common.

7.2 DATA REQUIREMENTS

As the primary purpose of this section is to modify the simulator's calculated pore volumes and transmissibilities, then the properties used to define these arrays must have been fully defined in the GRID section. The arrays available for modification in the EDIT section are listed in Table 7.1 together with the associated GRID arrays used to generate the EDIT property array.

Cartesian And Irregular Corner-Point Grids Keywords		Radial Grid Keywords	
GRID	**EDIT**	**GRID**	**EDIT**
TOPS	DEPTH	TOPS	DEPTH
DX	PORV	DR	PORV
DY		THETA	
DZ		DZ	
DZNET		DZNET	
PORO		PORO	
NTG		NTG	
PERMX	TRANX	PERMR	TRANR
MULTX		MULTR	
PERMY	TRANY	PERMTHT	TRANTHT
MULTY		MULTTHT	
PERMZ	TRANZ	PERMZ	TRANZ
MULTZ		MULTZ	

Notes:

1) Currently Radial Grids have not been implemented in OPM Flow.
2) The GRID property association to the EDIT property is only indicative as several variables, DZNET and NTG for example, are also used in the transmissibility calculations.

Table 7.1: EDIT Section Arrays Available for Modification

An example pore volume array (PORV property) from the Volve[136] field is shown in Figure 7.1 and Figure 7.2 illustrates the model's transmissibility in the x-direction (TRANX).

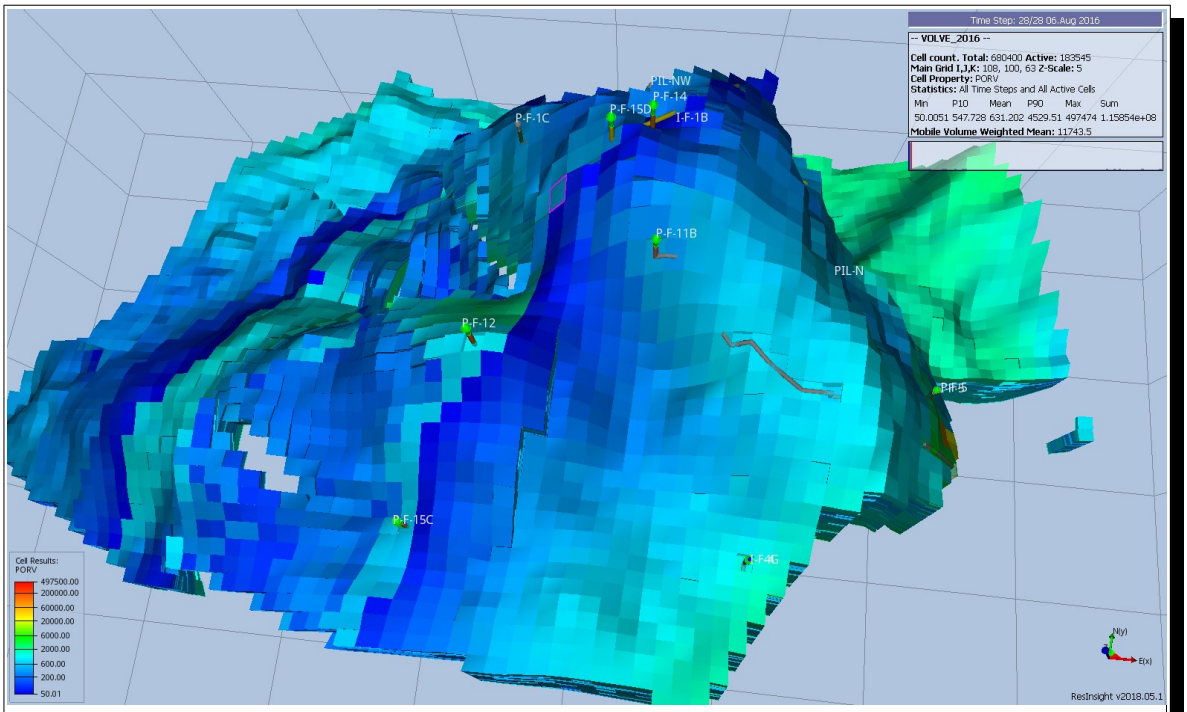


Figure 7.1: Volve Full Field Model PORV Array

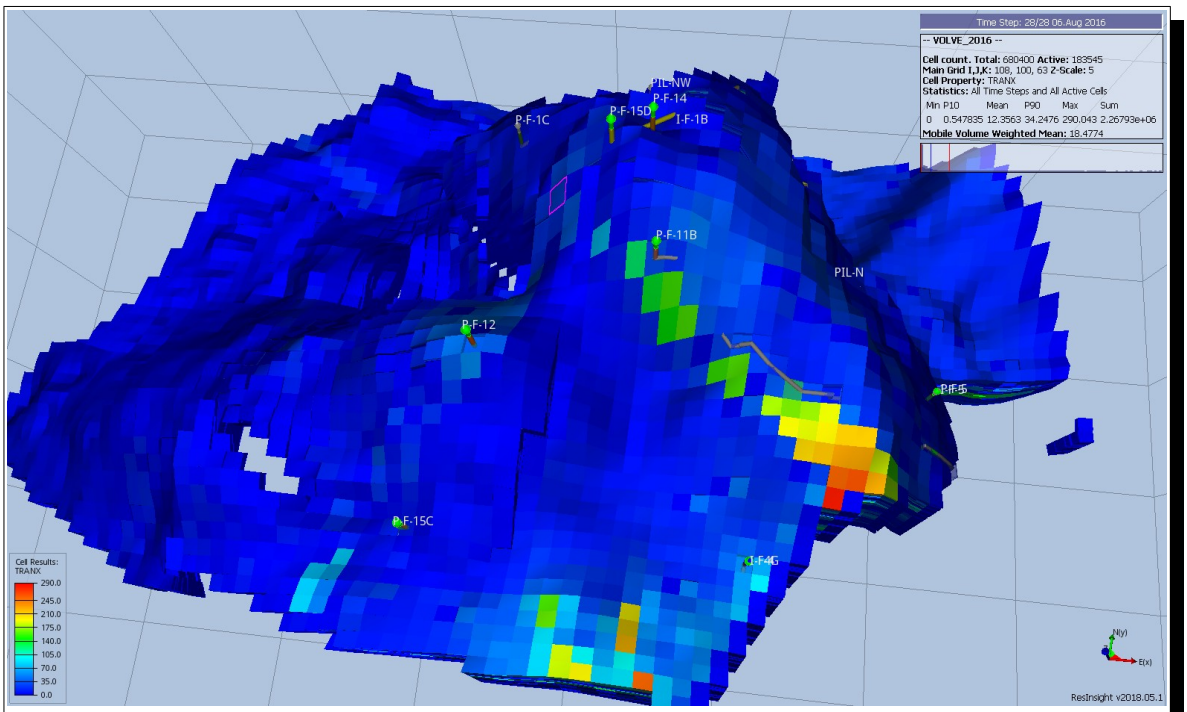


Figure 7.2: Volve Full Field Model TRANX Array

[136] *The Volve Data was approved for data sharing in 2018 by the initiative of the last Operating company, Equinor and approved by the license partners ExxonMobil E&P Norway AS and Bayerngas Norge AS in the end of 2017.*

7.3 KEYWORD DEFINITIONS

7.3.1 ADD – ADD A CONSTANT TO A SPECIFIED ARRAY

The ADD keyword adds a constant to a specified array or part of an array. The constant can be real or integer depending on the array type; however, the arrays that can be operated on is dependent on which section the ADD keyword is being applied.

See *ADD – Add a Constant to a Specified Array* in the GRID section for a full description.

7.3.2 ADDREG – ADD A CONSTANT TO AN ARRAY BASED ON A REGION NUMBER

The ADDREG keyword adds a constant to a specified array or part of an array based on cells with a specific region number. The region number array can be FLUXNUM, MULTNUM or OPERNUM and these arrays must be defined and be available before the ADDREG keyword is read by the simulator. The constant can be real or integer depending on the property array type; however, the property arrays that can be operated on is dependent on which section the ADDREG keyword is being used.

See *ADDREG – Add a Constant to an Array based on a Region Number* in the GRID section for a full description.

7.3.3 BOUNDARY – DEFINE A BOUNDARY BOX FOR PRINTING

The BOUNDARY keyword defines a rectangular grid for printing various arrays to the output print file (*.PRT); thus, avoiding printing all the elements in the selected array.

See *BOUNDARY – Define a Boundary Box for Printing* in the GRID section for a full description.

7.3.4 BOX - DEFINE A RANGE OF GRID BLOCKS TO ENTER PROPERTY DATA

BOX defines a range of grid blocks for which subsequent data is assigned for all the cells in the defined BOX. Note that the BOX grid is reset by the keyword ENDBOX by resetting the current defined BOX to be the whole grid. The keyword can be used for any array and for all grid types.

See *BOX - Define a Range of Grid Blocks to Enter Property Data* in the GRID section for a full description.

7.3.5 COLUMNS – DEFINE INPUT FILE COLUMN MARGINS

The COLUMNS keyword defines the input file column margins; characters outside the margins are ignored by the input parser.

See *COLUMNS – Define Input File Column Margins* in the GLOBAL section for a full description.

7.3.6 COPY – COPY ARRAY DATA TO ANOTHER ARRAY

The COPY keyword copies an array (or part of an array) to another array or part of an array. The arrays can be real or integer depending on the array type; however, the arrays that can be operated on is dependent on which section the COPY keyword is being used.

See *COPY – Copy Array Data to Another Array* in the GRID section for a full description.

7.3.7 COPYREG – COPY AN ARRAY TO ANOTHER ARRAY BASED ON A REGION NUMBER

The COPYREG keyword copies a specified array or part of an array based on cells with a specific region number to another array. The region number array can be FLUXNUM, MULTNUM or OPERNUM and these arrays must be defined and be available before the COPYREG keyword is read by the simulator. The property arrays can be real or integer depending on the property array type; however, the property arrays that can be operated on is dependent on which section the COPYREG keyword is being used.

See *COPYREG – Copy an Array to Another Array based on a Region Number* in the GRID section for a full description.

7.3.8 DEBUG – DEFINE THE DEBUG DATA TO BE PRINTED TO FILE

This keyword defines the debug data to be written to the debug file (*.DBG). This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

See *DEBUG – Define the Debug Data to be Printed to File* in the GLOBAL section for a full description.

7.3.9 DEPTH - EDITS THE DEPTH AT THE CENTER OF EACH CELL

Description

The DEPTH keywords modifies the depth at the center of selected cells in the model. The cells DEPTH are calculated by OPM Flow at the end of the GRID section and this keyword allows the user to adjust the calculated depths in the EDIT section. The area to be modified can be defined via the various grid selection keywords, ADD, BOX, EQUALS, etc., and areas that are not selected remain unchanged.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	DEPTH	DEPTH is an array of real numbers defining the depth at the center of each cell in the model. Only the values in the currently defined input BOX needed be entered. Repeat counts may be used, for example 30*5201.0.			
		feet	m	cm	None
Notes: 1) The keyword is terminated by a "/".					

Table 7.2: Depth Keyword Description

See also the TOPS keyword to define the top structural depth for the cells.

Examples

The example below modifies the DEPTH of the cells for a selection of 10 cells from an NX = 10, NY = 11 and NZ = 20 model.

```
--
--        DEFINE INPUT BOX FOR EDITING INPUT ARRAYS
--
--        ---------- BOX ---------
--        I1  I2   J1  J2   K1  K2
BOX
          1  10   11  11   20  20 / SET BOX AREA TO BE MODIFIED
/
--
--        SET GRID BLOCK CENTER DEPTH FOR THE GRID BLOCKS
--
DEPTH     10*3500.0                                                    /
--
--        DEFINE END OF INPUT BOX EDITING OF INPUT ARRAYS
--
ENDBOX
```

Alternatively the EQUALS keyword can be used to perform the same edit.

```
--
--        ARRAY       CONSTANT     ---------- BOX ---------
--                                 I1  I2   J1  J2   K1  K2
EQUALS
          DEPTH       3500.0        1  10   11  11   20  20 / RESET DEPTH
/
```

7.3.10 DIFFR – DEFINE GRID BLOCK RADIAL DIRECTION DIFFUSIVITY VALUES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The DIFFR keyword defines the radial direction diffusivity values for cells in the current input box for when the Diffusivity option has been activated by the DIFFUSE keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

7.3.11 DIFFTHT – DEFINE GRID BLOCK THETA DIRECTION DIFFUSIVITY VALUES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The DIFFTHT keyword defines the theta direction diffusivity values for cells in the current input box for when the Diffusivity option has been activated by the DIFFUSE keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

7.3.12 DIFFX – DEFINE GRID BLOCK X-DIRECTION DIFFUSIVITY VALUES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The DIFFX keyword defines the x-direction diffusivity values for cells in the current input box for when the Diffusivity option has been activated by the DIFFUSE keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

7.3.13 DIFFY – DEFINE GRID BLOCK Y-DIRECTION DIFFUSIVITY VALUES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The DIFFY keyword defines they y-direction diffusivity values for cells in the current input box for when the Diffusivity option has been activated by the DIFFUSE keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

7.3.14 DIFFZ – DEFINE GRID BLOCK Z-DIRECTION DIFFUSIVITY VALUES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The DIFFZ keyword defines the z-direction diffusivity values for cells in the current input box for when the Diffusivity option has been activated by the DIFFUSE keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

7.3.15 ECHO – ACTIVATE ECHOING OF USER INPUT FILES TO THE PRINT FILE

Turns on echoing of all the input files to the print file; note that this keyword is activated by default and can subsequently be switched off by the NOECHO activation keyword.

See *ECHO – Activate Echoing of User Input Files to the Print File* in the GLOBAL section for a full description.

7.3.16 EDIT - DEFINE THE START OF THE EDIT SECTION OF KEYWORDS

Description

The EDIT activation keyword marks the end of the GRID section and the start of the EDIT section that enables modifications to the OPM Flow calculated properties derived from the data entered in the GRID section, for example grid block pore volumes via the PORV array and the transmissibilities via the TRANX, TRANY and TRANZ family of keywords.

There is no data required for this keyword.

Example

```
-- ==========================================================================
--
-- EDIT SECTION
--
-- ==========================================================================
EDIT
```

The above example marks the end of the GRID section and the start of the EDIT section in the OPM Flow data input file.

7.3.17 EDITNNC – SCALE NON-NEIGHBOR CONNECTIONS BETWEEN CELLS MANUALLY

Description

EDITNNC enables Non-Neighbor Connections ("NNC"), entered via the NNC keyword or calculated by the simulator, to be multiplied (re-scaled) by a constant. For example, if the existing transmissibility between non-neighbor connections is Told and the multiplier is C, then the resulting transmissibility, Tnew, will be $T_{new} = C \ x \ T_{old}$. Only previously defined NNC's entered via the NNC keyword or calculated by the simulator can be edited, otherwise a warning message will be printed.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	I1	A positive integer that defines the first grid block in the I-direction in a non-neighbor connection, must be greater than or equal to one and less than or equal to NX on the DIMENS keyword in the RUNSPEC section.			None
2	J1	A positive integer that defines the first grid block in the J-direction in a non-neighbor connection, must be greater than or equal to one and less than or equal to NY on the DIMENS keyword in the RUNSPEC section.			None
3	K1	A positive integer that defines the first grid block in the K-direction in a non-neighbor connection, must be greater than or equal to one and less than or equal to NZ on the DIMENS keyword in the RUNSPEC section.			None
4	I2	A positive integer that defines the second grid block in the I-direction in a non-neighbor connection, must be greater than or equal to one and less than or equal to NX on the DIMENS keyword in the RUNSPEC section.			None
5	J2	A positive integer that defines the second grid block in the J-direction in a non-neighbor connection, must be greater than or equal to one and less than or equal to NY on the DIMENS keyword in the RUNSPEC section.			None
6	K2	A positive integer that defines the second grid block in the K-direction in a non-neighbor connection, must be greater than or equal to one and less than or equal to NZ on the DIMENS keyword in the RUNSPEC section.			None
7	TRANSMUL	TRANSMUL is a positive real number greater than or equal to zero that defines a constant that scales the transmissibility between the first grid block (I1, J1, K1) and the second grid block (I2, J2, K2). The default value of one means no scaling will be applied.			
		dimensionless	dimensionless	dimensionless	1
8	ISATNUM1	The default value of zero means the existing saturation table allocated to the upstream cell (I1,J1,K1).			0
9	ISATNUM2	ISATNUM2 is a positive integer defining which saturation table number (relative permeability table) to be used for flow from the second grid block to the first grid block. The default value of zero means the existing saturation table allocated to the downstream cell (I2,J2,K2).			0

No.	Name	Description			Default
		Field	Metric	Laboratory	
10	IPRSNUM1	IPRSNUM1 is a positive integer defining which pressure table number (PVT table) to be used for flow from the first grid block to the second grid block. The default value of zero means the existing PVT table allocated to the upstream cell (I1,J1,K1).			0
11	IPRSNUM2	IPRSNUM2 is a positive integer defining which pressure table number (PVT table) to be used for flow from the second grid block to the first grid block. The default value of zero means the existing PVT table allocated to the downstream cell (I2,J2,K2).			0
12	FACE1	FACE1 is a character string that defines the face associated with flow from the first grid block to the second grid block, where FACE1 can have values of: X+, X-, Y+, Y-, Z+, or Z-. FACE1 is used with the commercial simulator's Vertical Equilibrium option which is not supported by OPM Flow.			None
13	FACE2	FACE2 is a character string that defines the face associated with flow from the second grid block to the first grid block, where FACE2 can have values of: X+, X-, Y+, Y-, Z+, or Z-. FACE2 is used with the commercial simulator's Vertical Equilibrium option which is not supported by OPM Flow.			None
14	DIFFNNC	DIFFNNC is a positive real number greater than or equal to zero that scales the diffusivity between the first grid block (I1, J1, K1) and the second grid block (I2, J2, K2).			
		dimensionless	dimensionless	dimensionless	0.0

Notes:

1) Each record must be terminated by a "/" and the keyword is terminated by a "/".

Table 7.3: EDITNNC Keyword Description

Note that although items (8) to (14) for this keyword are not available in OPM Flow, even if they were, it is strongly recommended that these items are defaulted if the data is being entered manually, as opposed to being generated by pre-processing software.

Care should be taken that cells in different PVTNUM regions (see the PVTNUM keyword in the REGIONS section) are not connected, since the fluid properties are associated with a cell. If for example, a rbbl or a rm^3 of oil flows from PVTNUM region 1 to PVTNUM region 2, then the oil properties of that oil will change from the PVT 1 data set to the PVT data set 2. This will result in material balance errors, that may or may not cause numerical issues.

If the transmissibility across a fault needs to be modified see the FAULTS and MULTFLT keywords in the GRID section for an alternative and less complicated method to modifying fault transmissibilities. Transmissibility between reservoir regions can be modified by using MULTREGT keyword, provided MULTNUM has been used to define the inter-region transmissibility region numbers for each grid block. Finally, the MULTX, MULTY and MULTZ series of keywords can be used to modify transmissibility between various cells. All the aforementioned keywords are described in the GRID section.

See also the EDITNNCR keyword in the EDIT section that resets an existing NNC to a user defined value.

Example

```
--
--       MANUALLY RESCALE NON-NEIGHBOR CONNECTIONS
--
--       ------------ BOX -----------    -- TRANSMUL --
--       I1   J1   K1    I2   J2   K2
EDITNCC
         1    1    1     1    1    2        0.2000     / SET NNC FOR FAULT
         1    1    2     1    1    3        0.2000     / SET NNC FOR FAULT
         1    1    3     1    1    4        0.2000     / SET NNC FOR FAULT
/
```

The above example multiplies the transmissibility between cells (1, 1, 1) and (1, 1, 2), (1, 1, 2) and (1, 1, 3) and finally between (1, 1, 3) and (1, 1, 4) by 0.200.

7.3.18 EDITNNCR – RESET NON-NEIGHBOR CONNECTIONS BETWEEN CELLS MANUALLY

Description

EDITNNCR enables Non-Neighbor Connections ("NNC"), entered via the NNC keyword or calculated by the simulator, to be reset to a user defined value. Only previously defined NNC's entered via the NNC keyword or calculated by the simulator can be edited, otherwise a warning message will be printed. See also the EDITNNC keyword in the EDIT section that scales an existing NNC.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	I1	A positive integer that defines the first grid block in the I-direction in a non-neighbor connection, must be greater than or equal to one and less than or equal to NX on the DIMENS keyword in the RUNSPEC section.			None
2	J1	A positive integer that defines the first grid block in the J-direction in a non-neighbor connection, must be greater than or equal to one and less than or equal to NY on the DIMENS keyword in the RUNSPEC section.			None
3	K1	A positive integer that defines the first grid block in the K-direction in a non-neighbor connection, must be greater than or equal to one and less than or equal to NZ on the DIMENS keyword in the RUNSPEC section.			None
4	I2	A positive integer that defines the second grid block in the I-direction in a non-neighbor connection, must be greater than or equal to one and less than or equal to NX on the DIMENS keyword in the RUNSPEC section.			None
5	J2	A positive integer that defines the second grid block in the J-direction in a non-neighbor connection, must be greater than or equal to one and less than or equal to NY on the DIMENS keyword in the RUNSPEC section.			None
6	K2	A positive integer that defines the second grid block in the K-direction in a non-neighbor connection, must be greater than or equal to one and less than or equal to NZ on the DIMENS keyword in the RUNSPEC section.			None
7	TRANSNNC	TRANSNNC is a positive real number greater than or equal to zero that defines the transmissibility between the first grid block (I1, J1, K1) and the second grid block (I2, J2, K2). This value cannot be defaulted and must be defined.			
		cP.rb/day/psia	cP.rm^3/day/bars	cP.rcc/hr/atm	None
8	ISATNUM1	ISATNUM1 is a positive integer defining which saturation table number (relative permeability table) to be used for flow from the first grid block to the second grid block. The default value of zero means the existing saturation table allocated to the upstream cell (I1,J1,K1).			0
9	ISATNUM2	ISATNUM2 is a positive integer defining which saturation table number (relative permeability table) to be used for flow from the second grid block to the first grid block. The default value of zero means the existing saturation table allocated to the downstream cell (I2,J2,K2).			0

No.	Name	Description			Default
		Field	Metric	Laboratory	
10	IPRSNUM1	IPRSNUM1 is a positive integer defining which pressure table number (PVT table) to be used for flow from the first grid block to the second grid block. The default value of zero means the existing PVT table allocated to the upstream cell (I1,J1,K1).			0
11	IPRSNUM2	IPRSNUM2 is a positive integer defining which pressure table number (PVT table) to be used for flow from the second grid block to the first grid block. The default value of zero means the existing PVT table allocated to the downstream cell (I2,J2,K2).			0
12	FACE1	FACE1 is a character string that defines the face associated with flow from the first grid block to the second grid block, where FACE1 can have vales of: X+, X-, Y+, Y-, Z+, or Z-.			None
13	FACE2	FACE2 is a character string that defines the face associated with flow from the second grid block to the first grid block, where FACE2 can have vales of: X+, X-, Y+, Y-, Z+, or Z-.			None
14	DIFFNNC	DIFFNNC is a positive real number greater than or equal to zero that scales the diffusivity between the first grid block (I1, J1, K1) and the second grid block (I2, J2, K2). The default value is the value calculated in the GRID section.			
		feet	meters	cm	1*
Notes: 1) Each record must be terminated by a "/" and the keyword is terminated by a "/".					

Table 7.4: EDITNNCR Keyword Description

Note that although items (8) to (14) for this keyword are not available in OPM Flow, even if they were, it is strongly recommended that these items are defaulted if the data is being entered manually, as opposed to being generated by pre-processing software.

If the transmissibility across a fault needs to be modified see the FAULTS and MULTFLT keywords in the GRID section for an alternative and less complicated method to modifying fault transmissibilities. Transmissibility between reservoir regions can be modified by using MULTREGT keyword, provided MULTNUM has been used to define the inter-region transmissibility region numbers for each grid block. Finally, the MULTX, MULTY and MULTZ series of keywords can be used to modify transmissibility between various cells. All the aforementioned keywords are described in the GRID section.

Example

```
--
--       MANUALLY RESET NON-NEIGHBOR CONNECTIONS
--
--       ------------ BOX -----------    -- TRANSNNC --
--       I1   J1   K1    I2   J2   K2
EDITNCCR
         1    1    1     1    1    2        0.2500    / RESET NNC TRANS FOR FAULT
         1    1    2     1    1    3        0.2500    / RESET NNC TRANS FAULT
         1    1    3     1    1    4        0.2500    / RESET NNC TRANS FAULT
/
```

The above example res-sets the transmissibility between cells (1, 1, 1) and (1, 1, 2), (1, 1, 2) and (1, 1, 3) and (1, 1, 3) and (1, 1, 4) to be 0.2500.

7.3.19 END – DEFINE THE END OF THE INPUT FILE

This keyword marks the end of the input file and can occur in any section. Any keywords and data after the END keyword are ignored.

See *END – Define the End of the Input File* in the GLOBAL section for a full description.

7.3.20 ENDBOX – DEFINE THE END OF THE BOX DEFINED GRID

This keyword marks the end of a previously defined BOX sub-grid as defined by a previously entered BOX keyword. The keyword resets the input grid to be the full grid as defined by the NX, NY, and NZ variables on the DIMENS keyword in the RUNSPEC section.

See *ENDBOX – Define the End of the BOX Defined Grid* in the GRID section for a full description.

7.3.21 ENDFIN – END THE DEFINITION OF A LOCAL GRID REFINEMENT

NDFIN defines the end of a Cartesian or radial local grid refinement ("LGR") definition and a LGR property definition data set.

See *ENDFIN – End the Definition of a Local Grid Refinement* in the GRID section for a full description.

7.3.22 ENDINC – DEFINE THE END OF AN INCLUDE FILE

This keyword marks the end of an include file specified on the INCLUDE keyword. When the ENDINC keyword is encountered in the INCLUDE file, input data is read from the next keyword in the current file. Any keywords and data after the ENDINC keyword in the INCLUDE file are ignored.

See *ENDINC – Define the End of an Include File* in the GLOBAL section for a full description.

7.3.23 ENDSKIP – DEACTIVATE SKIPPING OF KEYWORDS AND INPUT DATA

Turns off skipping of keywords that was activated by the SKIP, SKIP100, or SKIP300 keywords. Note that for each SKIP keyword activated there must be a corresponding ENDSKIP keyword.

See *ENDSKIP – DeActivate Skipping of Keywords and Input Data* in the GLOBAL section for a full description.

7.3.24 EQUALREG – SETS AN ARRAY TO A CONSTANT BY REGION NUMBER

The EQUALREG keyword sets a specified array to a constant for cells with a specific region number. The region number array can be FLUXNUM, MULTNUM or OPERNUM and these arrays must be defined and be available before the EQUALREG keyword is read by the simulator. The constant can be real or integer depending on the property array type; however, the property arrays that can be operated on is dependent on which section the EQUALREG keyword is being used.

See *EQUALREG – Sets an Array to a Constant by Region Number* in the GRID section for a full description.

7.3.25 EQUALS – SETS A SPECIFIED ARRAY TO A CONSTANT

The EQUALS keyword sets a specified array or part of an array to a constant. The constant can be real or integer depending on the array type; however, the arrays that can be operated on is dependent on which section the EQUALS keyword is being used.

See *EQUALS – Sets a Specified Array to a Constant* in the GRID section for a full description.

7.3.26 EXTRAPMS – ACTIVATE EXTRAPOLATION WARNING MESSAGES

The EXTRAPMS keyword activates extrapolation warning messages for when OPM Flow extrapolates the PVT or VFP tables. Frequent extrapolation warning messages should be investigated and resolved as this would indicate possible incorrect data and may result in the simulator extrapolating to unrealistic values.

See *EXTRAPMS – Activate Extrapolation Warning Messages* in the GLOBAL section for a full description.

7.3.27 FILEUNIT – ACTIVATE UNIT CONSISTENCY VERIFICATION

The FILEUNIT keyword defines the units of the of the data set, and is used to verify that the units in the input deck and any associated include files are consistent grid data. The keyword does not provide for the conversion between different sets of units.

See *FILEUNIT – Activate Unit Consistency Checking* in the GRID section for a full description.

7.3.28 FORMFEED – DEFINED THE PRINT FILE FORM-FEED CHARACTER

The FORMFEED keyword defines the form-feed character, or carriage control character, for the output print (*.PRT) run summary (*.RSM) files. The keyword should be place at the very top of the input file.

See *FORMFEED – Defined the Print File Form-Feed Character* in the GLOBAL section for a dull description.

7.3.29 GETDATA – LOAD AND ASSIGN DATA ARRAY FROM INIT OR RESTART FILE

The GETDATA keyword loads a data array from a previously generated INIT or RESTART file and assigns the loaded array to either same array in the run or another array name.

See *GETDATA – Load and Assign Data Array from INIT or RESTART Files* in the GRID section for a full description

7.3.30 HMMULT – HISTORY MATCH GRID TRANSMISSIBILITY & PORE VOLUME GRADIENT CUMULATIVE MULTIPLIERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The HMMULT series of keywords defines the history match gradient cumulative permeability multipliers, for when the History Match Gradient option has been activated by the HMDIMS keyword in the RUNSPEC section. The keyword consists of the first six characters of "HMMULT" followed by a one or two character string shown in Table 7.5, that determines the transmissibility direction, for example, HMMULTX.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

Mnemonic	Cartesian Grid		Radial Grid	
	Grid Keyword	HMMULT Keyword	Grid Keyword	HMMULT Keyword
X/R	MULTX	HMMULTX	MULTR	HMMULTR
XY	MULTXY	HMMULTXY		
Y/HT	MULTY	HMMULTY	MULTTHT	HMMULTTH
Z	MULTZ	HMMULTZ	MULTZ	HMMULTZ
PV	MULTPV	HMMULTPV	MULTPV	HMMULTPV

Table 7.5: HMMULT Keyword List

See also the HMMLT keyword in the GRID section.

7.3.31 IMPORT – IMPORT GRID FILE DATA AT THE CURRENT POSITION

The IMPORT keyword informs the simulator to continue reading input data from the specified IMPORT file. When the end of the IMPORT file is reached, input data is read from the next keyword in the current file. Normally IMPORT files are generated by grid pre-processing software and the keyword allows for both formatted and unformatted (binary) files to be loaded.

See *IMPORT – Import Grid File Data at the Current Position* in the GRID section for a full description.

7.3.32 INCLUDE – LOAD ANOTHER DATA FILE AT THE CURRENT POSITION

The INCLUDE keyword informs OPM Flow to continue reading input data from the specified INCLUDE file. When the end of the INCLUDE file is reached, or the ENDINC keyword is encountered in the included file, input data is read from the next keyword in the current file.

See *INCLUDE – Load Another Data File at the Current Position* in the GLOBAL section for a full description.

7.3.33 LGRCOPY – ACTIVATE LOCAL GRID REFINEMENT INHERITANCE

The LGRCOPY keyword actives Local Grid Refinement ("LGR") Inheritance option that allows the LGR to inherit the properties of the global or host cell containing a LGR grid block at the start of the GRID section, as oppose to the normal process of applying this transform at the end of the GRID section. LGRCOPY can be used in the RUNSPEC, GRID and EDIT sections. If used in the RUNSPEC section then the option is applied to all LGRs defined in the input file, whereas if used in the GRID and EDIT sections the keyword must be placed inside a LGR definition section, that is between a CARFIN (Cartesian LGR grid) or RADIN/RADIN4 (radial LGR grid) and the ENDFIN keyword. In the latter case inheritance is applied on an individual LGR basis.

See *LGRCOPY – Activate Local Grid Refinement Inheritance* in the RUNSPEC section for a full description.

7.3.34 MAXVALUE – SETS A MAXIMUM VALUE FOR AN ARRAY ELEMENT

The MAXVALUE keyword sets a maximum value for the specified array or part of an array. The constant can be real or integer depending on the array type; however, the arrays that can be operated on is dependent on which section the MAXVALUE keyword is being used.

See *MAXVALUE – Sets a Maximum Value for an Array Element* in the GRID section for a full description.

7.3.35 MESSAGE – OUTPUT USER MESSAGE

The MESSAGE keyword outputs a user message to the terminal, as well as to the print (*.PRT) and debug (*.DBG) files. Note this is different to the MESSAGES keyword, that defines OPM Flows message print limits and stop limits generated by the simulator.

See *MESSAGE – Output User Message* in the GLOBAL section for a full description.

7.3.36 MESSAGES – DEFINE MESSAGE PRINT LIMITS AND STOP LIMITS

The MESSAGES keyword defines the print and stops levels for various messages. The "print limits" set the maximum number of messages that will be printed, after which no more messages will be printed and the "stop limits" terminate the run when these limits are exceeded. There are six levels of message that increase in severity from informative all the way to programming errors, as outlined in Error: Reference source not found.

See *MESSAGES – Define Message Print Limits and Stop Limits* in the GLOBAL section for a full description.

7.3.37 MINVALUE – SET A MINIMUM VALUE FOR AN ARRAY ELEMENT

The MINVALUE keyword sets a minimum value for the specified array or part of an array. The constant can be real or integer depending on the array type; however, the arrays that can be operated on is dependent on which section the MINVALUE keyword is being used.

See *MINVALUE – Set a Minimum Value for an Array Element* in the GRID section for a full description.

7.3.38 MULTFLT – MULTIPLY THE TRANSMISSIBILITY OF A DEFINED FAULT BY A CONSTANT

MULTFLT enables the transmissibilities across defined faults, as declared by the FAULTS keyword, to be modified. They keyword allows for the re-scaling of the existing fault transmissibilities calculated by OPM Flow, for example setting a fault to be completely sealing by setting the multiplier to zero..

See *MULTFLT – Multiply the Transmissibility of a Defined Fault by a Constant* in the GRID section for a full description.

7.3.39 MULTIPLY – MULTIPLY A SPECIFIED ARRAY BY A CONSTANT

The MULTIPLY keyword multiplies a specified array or part of an array by a constant. The constant can be real or integer depending on the array type; however, the arrays that can be operated on is dependent on which section the keyword is being used.

See *MULTIPLY – Multiply a Specified Array by a Constant* in the GRID section for a full description.

7.3.40 MULTIREG – MULTIPLY AN ARRAY BY A CONSTANT BASED ON A REGION NUMBER

The MULTIREG keyword multiplies an array or part of an array by a constant for cells with a specific region number. The region number array can be FLUXNUM, MULTNUM or OPERNUM and these arrays must be defined and be available before the MULTIREG keyword is read by the simulator. The constant can be real or integer depending on the property array type; however, the property arrays that can be operated on is dependent on which section the MULTIREG keyword is being used.

See *MULTIREG – Multiply an Array by a Constant based on a Region Number* in the GRID section for a full description.

7.3.41 MULTPV – MULTIPLY CELL PORE VOLUMES BY A CONSTANT

MULTPV multiples the pore volumes of a cell by a real positive constant for all the cells in the model via an array. An alternative to defining the complete array is to use the BOX keyword to define an area of the grid and then use the MULTPV keyword to set the multipliers just for the area defined by the BOX keyword (see the example).

See *MULTPV – Multiply Cell Pore Volumes by a Constant*in the GRID section for a full description.

7.3.42 MULTR - MULTIPLY CELL TRANSMISSIBILITY IN THE +R DIRECTION

MULTR multiples the transmissibility between two cell faces in the +R direction for all the cells in the model via an array, that is the keyword sets the transmissibility multiplier of block (I, J, K) between the cells (I, J, K) and (I+I, J, K). The keyword should only be used with radial grids, as declared by the RADIAL keyword in the RUNSPEC section.

See *MULTR - Multiply Cell Transmissibility in the +R Direction* in the GRID section for a full description.

7.3.43 MULTR- - MULTIPLY CELL TRANSMISSIBILITY IN THE -R DIRECTION

MULTX- multiples the transmissibility between two cell faces in the -X direction for all the cells in the model via an array, that is the keyword sets the transmissibility multiplier of block (I, J, K) between the cells (I-1, J, K) and (I, J, K). The keyword should only be used with radial grids, as declared by the RADIAL keyword in the RUNSPEC section.

See *MULTR- - Multiply Cell Transmissibility in the -R Direction* in the GRID section for a full description.

7.3.44 MULTREGD – MULTIPLY DIFFUSIVITIES BETWEEN REGIONS

The MULTREGT keyword multiplies the diffusivity between two regions by a constant. The region number array can be FLUXNUM, MULTNUM or OPERNUM and these arrays must be defined and be available before the MULTREGT keyword is read by the simulator. The constant should be a real number.

See *MULTREGD – Multiply Diffusivities Between Regions* in the GRID section for a full description.

7.3.45 MULTREGH – MULTIPLY THERMAL CONDUCTIVITIES BETWEEN REGIONS

The MULTREGH keyword multiplies the thermal conductivity between two regions by a constant. The region number array can be FLUXNUM, MULTNUM or OPERNUM and these arrays must be defined and be available before the MULTREGT keyword is read by the simulator. The constant should be a real number.

See *MULTREGH – Multiply Thermal Conductivities Between Regions* in the GRID section for a full description.

7.3.46 MULTREGP– MULTIPLY PORE VOLUMES BASED ON REGION NUMBER

The MULTREGP keyword multiplies the pore volume of a cell by a constant for all cells with a specific region number. The region number array can be FLUXNUM, MULTNUM or OPERNUM and these arrays must be defined and be available before the MULTREGP keyword is read by the simulator. The constant should be a real number.

See *MULTREGP – Multiply Pore Volumes Based On Region Number* in the GRID section for a full description.

7.3.47 MULTREGT– MULTIPLY TRANSMISSIBILITIES BETWEEN REGIONS

The MULTREGT keyword multiplies the transmissibility between two regions by a constant. The region number array can be FLUXNUM, MULTNUM or OPERNUM and these arrays must be defined and be available before the MULTREGT keyword is read by the simulator. The constant should be a real number.

See *MULTREGT – Multiply Transmissibilities Between Regions*in the GRID section for a full description.

7.3.48 MULTTHT - MULTIPLY CELL TRANSMISSIBILITY IN THE +THETA DIRECTION

MULTTHT multiples the transmissibility between two cell faces in the +Theta direction for all the cells in the model via an array, that is the keyword sets the transmissibility multiplier of block (I, J, K) between the cells (I, J, K) and (I, J+1, K). The keyword should only be used with radial grids, as declared by the RADIAL keyword in the RUNSPEC section.

See *MULTTHT - Multiply Cell Transmissibility in the +Theta Direction* in the GRID section for a full description.

7.3.49 MULTTHT- - MULTIPLY CELL TRANSMISSIBILITY IN THE -THETA DIRECTION

MULTTHT- multiples the transmissibility between two cell faces in the -Theta direction for all the cells in the model via an array, that is the keyword sets the transmissibility multiplier of block (I, J, K) between the cells (I, J-1, K) and (I, J, K). The keyword should only be used with radial grids, as declared by the RADIAL keyword in the RUNSPEC section.

See *MULTTHT- - Multiply Cell Transmissibility in the -Theta Direction* in the GRID section for a full description.

7.3.50 MULTX - MULTIPLY CELL TRANSMISSIBILITY IN THE +X DIRECTION

MULTX multiples the transmissibility between two cell faces in the +X direction for all the cells in the model via an array, that is the keyword sets the transmissibility multiplier of block (I, J, K) between the cells (I, J, K) and (I+1, J, K).

See *MULTX - Multiply Cell Transmissibility in the +X Direction* in the GRID section for a full description.

7.3.51 MULTX- - MULTIPLY CELL TRANSMISSIBILITY IN THE -X DIRECTION

MULTX- multiples the transmissibility between two cell faces in the -X direction for all the cells in the model via an array, that is the keyword sets the transmissibility multiplier of block (I, J, K) between the cells (I-1, J, K) and (I, J, K).

See *MULTX- - Multiply Cell Transmissibility in the -X Direction* in the GRID section for a full description.

7.3.52 MULTY - MULTIPLY CELL TRANSMISSIBILITY IN THE +Y DIRECTION

MULTY multiples the transmissibility between two cell faces in the +Y direction for all the cells in the model via an array, that is the keyword sets the transmissibility multiplier of block (I, J, K) between the cells (I, J, K) and (I, J+1, K).

See *MULTY - Multiply Cell Transmissibility in the +Y Direction* in the GRID section for a full description.

7.3.53 MULTY- - MULTIPLY CELL TRANSMISSIBILITY IN THE -Y DIRECTION

MULTY- multiples the transmissibility between two cell faces in the -Y direction for all the cells in the model via an array, that is the keyword sets the transmissibility multiplier of block (I, J, K) between the cells (I, J-1, K) and (I, J, K).

See *MULTY- - Multiply Cell Transmissibility in the -Y Direction* in the GRID section for a full description.

7.3.54 MULTZ - MULTIPLY CELL TRANSMISSIBILITY IN THE +Z DIRECTION

MULTZ multiples the transmissibility between two cell faces in the +Z direction for all the cells in the model via an array, that is the keyword sets the transmissibility multiplier of block (I, J, K) between the cells (I, J, K) and (I, J, K+1).

See *MULTZ - Multiply Cell Transmissibility in the +Z Direction* in the GRID section for a full description.

7.3.55 MULTZ- - MULTIPLY CELL TRANSMISSIBILITY IN THE -Z DIRECTION

MULTZ- multiples the transmissibility between two cell faces in the -Z direction for all the cells in the model via an array, that is the keyword sets the transmissibility multiplier of block (I, J, K) between the cells (I, J, K-1) and (I, J, K).

See *MULTZ- - Multiply Cell Transmissibility in the -Z Direction* in the GRID section for a full description.

7.3.56 NOECHO – DEACTIVATE ECHOING OF USER INPUT FILES TO THE PRINT FILE

Turns off echoing of all the input files to the print file. Note by default echoing of the inputs files is active. but can subsequently be switched off by the NOECHO activation keyword.

See *NOECHO – Deactivate Echoing of User Input Files to the Print File* in the GLOBAL section for a full description.

7.3.57 NOWARN – DEACTIVATE WARNING MESSAGES

Turns off warning messages to be printed to the print file; note that this keyword is deactivated by default and can subsequently be switched off by the WARN activation keyword. The warning messages may be turned on and off using keywords WARN and NOWARN.

See *NOWARN – Deactivate Warning Messages* in the GLOBAL section for a full description.

7.3.58 OPERATE – DEFINE MATHEMATICAL OPERATIONS ON ARRAYS

This keyword, OPERATE, defines mathematical operations on property arrays (NTG, PORO etc.) and optionally using another property array as input to the function. The keyword allows for various mathematical functions and their associated variables to be defined and applied to the selected array data. Input constants can be real or integer depending on the array type; however, the arrays that can be operated on is dependent on which section the keyword is being used.

See *OPERATE – Define Mathematical Operations on Arrays* in the GRID section for a full description.

7.3.59 OPERATER – DEFINE MATHEMATICAL OPERATIONS ON ARRAYS BY REGION

This keyword defines the mathematical operations on arrays for specific regions in the commercial simulator and is currently not supported by OPM Flow. However, similar functionality is provided by the ADD and MULTIPLY keywords.

See *OPERATER – Define Mathematical Operations on Arrays by Region* in the GRID section for a full description.

7.3.60 PORV - DEFINE THE PORE VOLUMES FOR ALL THE CELLS

Description

PORV defines the pore volumes for all the cells in the model via an array. The keyword can be used for all grid types, except for the Radial Grid geometry. The keyword effectively overwrites previously entered and calculated data. The area to be modified can be defined via the various grid selection keywords, ADD, BOX, EQUALS, etc., and areas that are not selected remain unchanged.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	PORV	PORV is an array of real positive numbers assigning a pore volume to each cell in the model. Only the values in the currently defined input BOX needed be entered. Repeat counts may be used, for example 20*100.0.			
		rb	rm^3	rcc	None

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) Values not reset by this keyword remain unaltered.
3) The keyword is terminated by a "/".

Table 7.6: PORV Keyword Description

Example

```
--
--       DEFINE INPUT BOX FOR EDITING INPUT ARRAYS
--
--       ---------- BOX ---------
--       I1  I2   J1  J2   K1  K2
BOX
         1* 100   1* 100   20  20                          / DEFINE BOX AREA
--
--       SET PORV FOR THE GRID BLOCKS
--
PORV
         1000*0.00                                        /
--
--       DEFINE END OF INPUT BOX EDITING OF INPUT ARRAYS
--
ENDBOX
```

Here the BOX statement is used to define the input grid for the PORV keyword, which overwrites the pore volume previously calculated with pore volume values of zero, resulting in a no-flow boundary in that part of the field between layers 19 and 21, since layer 20 is deactivated. The ENDBOX keyword resets the input box to the full grid.

7.3.61 PYEND – END THE DEFINITION OF A PYINPUT SECTION

The PYINPUT and PYEND keywords are a part of OPM Flow's Python scripting facility that processes standard Python commands that can be used to manipulate and define the simulators input parameters during processing of the input deck. The main purpose of the facility is to script the construction of the various keywords. A PYINPUT Definition Section is terminated by a PYEND keyword on a separate single line.

See *PYEND – End the Definition of a PYINPUT Section* in the GRID section for a full description.

7.3.62 PYINPUT – DEFINE THE START OF A PYINPUT SECTION

The PYINPUT and PYEND keywords are a part of OPM Flow's Python scripting facility that processes standard Python commands that can be used to manipulate and define the simulators input parameters during processing of the input deck. The main purpose of the facility is to script the construction of the various keywords used by the simulator. PYINPUT declares the start of a PYINPUT Definition Section on a single separate line, which is then followed by various standard Python commands, with one command per line.

See *PYINPUT – Define the Start of a PYINPUT Section* in the GRID section for a full description.

7.3.63 REFINE – START THE DEFINITION OF A LOCAL GRID REFINEMENT

The REFINE keyword defines the start of a Cartesian or radial local grid refinement ("LGR") definition that sets the properties of the selected LGR. The keyword is then followed by the property keywords associated with the section where the keyword is being invoked. For example, if the REFINE keyword is used in the GRID section then most of the keywords in that section can be used to set the grid properties for the LGR.

See *REFINE – Start the Definition of a Local Grid Refinement* in the GRID section for a full description.

7.3.64 SKIP – ACTIVATE SKIPPING OF ALL KEYWORDS AND INPUT DATA

The SKIP keyword turns on skipping of keywords until the ENDSKIP activation keyword is encountered. All keywords and between the SKIP and ENDSKIP keywords are ignored by OPM Flow.

See *SKIP – Activate Skipping of All Keywords and Input Data* in the GLOBAL section for a full description.

7.3.65 SKIP100 – ACTIVATE SKIPPING OF BLACK-OIL KEYWORDS AND INPUT DATA

This keyword turns on skipping of black-oil keywords until the ENDSKIP activation keyword is encountered. All black-oil keywords between the SKIP100 and ENDSKIP keywords are ignored by OPM Flow.

See *SKIP100 – Activate Skipping of Black-Oil Keywords and Input Data* in the GLOBAL section for a full description.

7.3.66 SKIP300 – ACTIVATE SKIPPING OF "COMPOSITIONAL" KEYWORDS AND INPUT DATA

Turns on skipping of "compositional" keywords until the ENDSKIP activation keyword is encountered. All "compositional" keywords between the SKIP300 and ENDSKIP keywords are ignored by OPM Flow.

See *Error: Reference source not found* in the GLOBAL section for a full description.

7.3.67 TRANR - DEFINE THE TRANSMISSIBILITY IN THE +R DIRECTION FOR ALL THE CELLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

TRANR defines the transmissibility in the +R direction for all the cells in the model via an array. The keyword can only be used with Radial Grid geometry grids. The keyword effectively overwrites previously entered and calculated data. The transmissibility overwritten is the +R face transmissibility of each grid block, that is for cell (I, J, K) the transmissibility between cells (I, J, K) and (I+1, J, K).

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	TRANR	TRANR is an array of real positive numbers assigning the transmissibility in the R direction to each cell in the model. Repeat counts may be used, for example 20*100.0.			
		cP.rb/day/psia	cP.rm^3/day/bars	cP.rcc/hr/atm	None

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) Values not reset by this keyword remain unaltered.
3) The keyword is terminated by a "/".

Table 7.7: TRANR Keyword Description

See also the TRANTHT and TRANYZ keywords to modify the transmissibilities in the other directions.

Example

```
--
--       DEFINE INPUT BOX FOR EDITING INPUT ARRAYS
--
--       ---------- BOX ---------
--       I1  I2   J1  J2   K1  K2
BOX
         1   1    10  10   1  120                        / DEFINE BOX AREA
--
--       SET TRANR+ TRANSMISSIBILITY
--
TRANR
         120*0.00                                       /
--
--       DEFINE END OF INPUT BOX EDITING OF INPUT ARRAYS
--
ENDBOX
```

Here the BOX statement is used to define the input grid for the TRANR keyword, which overwrites the transmissibility previously calculated with transmissibility values of zero, resulting in a no-flow boundary in that part of the grid. The ENDBOX keyword resets the input box to the full grid.

7.3.68 TRANTHT - DEFINE THE TRANSMISSIBILITY IN THE +THETA DIRECTION FOR ALL THE CELLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

TRANTHT defines the transmissibility in the +Theta direction for all the cells in the model via an array. The keyword can only be used with Radial Grid geometry grids. The keyword effectively overwrites previously entered and calculated data. The transmissibility overwritten is the +Theta face transmissibility of each grid block, that is for cell (I, J, K) the transmissibility between cells (I, J, K) and (I, J+1, K).

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	TRANTHT	TRANTHT is an array of real positive numbers assigning the transmissibility in the +Theta direction to each cell in the model. Repeat counts may be used, for example 20*100.0.			
		cP.rb/day/psia	cP.rm^3/day/bars	cP.rcc/hr/atm	None

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) Values not reset by this keyword remain unaltered.
3) The keyword is terminated by a "/".

Table 7.8: TRANR Keyword Description

See also the TRANR and TRANYZ keywords to modify the transmissibilities in the other directions.

Example

```
--
--       DEFINE INPUT BOX FOR EDITING INPUT ARRAYS
--
--       ---------- BOX ---------
--       I1  I2   J1  J2   K1  K2
BOX
         10  10   1   6    1   3                          / DEFINE BOX AREA
--
--       SET TRANTHT TRANSMISSIBILITY
--
TRANTHT
         18*0.00                                         /
--
--       DEFINE END OF INPUT BOX EDITING OF INPUT ARRAYS
--
ENDBOX
```

Here the BOX statement is used to define the input grid for the TRANTHT keyword, which overwrites the transmissibility previously calculated with transmissibility values of zero, resulting in a no-flow boundary in that part of the grid. The ENDBOX keyword resets the input box to the full grid.

7.3.69 TRANX - DEFINE THE TRANSMISSIBILITY IN THE X DIRECTION FOR ALL THE CELLS

Description

TRANX defines the transmissibility in the X direction for all the cells in the model via an array. The keyword can be used for all grid types, except for the Radial Grid geometry. The keyword effectively overwrites previously entered and calculated data. The transmissibility overwritten is the +X face transmissibility of each grid block, that is for cell (I, J, K) the transmissibility between cells (I, J, K) and (I+1, J, K).

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	TRANX	TRANX is an array of real positive numbers assigning the transmissibility in the X direction to each cell in the model. Repeat counts may be used, for example 20*100.0.			
		cP.rb/day/psia	cP.rm^3/day/bars	cP.rcc/hr/atm	None

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) Values not reset by this keyword remain unaltered.
3) The keyword is terminated by a "/".

Table 7.9: TRANX Keyword Description

See also the TRANY and TRANYZ keywords to modify the transmissibilities in the other directions.

Example

```
--
--        DEFINE INPUT BOX FOR EDITING INPUT ARRAYS
--
--        ---------- BOX ---------
--        I1  I2   J1  J2   K1  K2
BOX
          1   1    10  10   1  120                          / DEFINE BOX AREA
--
--        SET TRANX+ TRANSMISSIBILITY
--
TRANX
          120*0.00                                         /
--
--        DEFINE END OF INPUT BOX EDITING OF INPUT ARRAYS
--
ENDBOX
```

Here the BOX statement is used to define the input grid for the TRANX keyword, which overwrites the transmissibility previously calculated with transmissibility values of zero, resulting in a no-flow boundary in that part of the field. The ENDBOX keyword resets the input box to the full grid.

7.3.70 TRANY - DEFINE THE TRANSMISSIBILITY IN THE Y DIRECTION FOR ALL THE CELLS

Description

TRANY defines the transmissibility in the Y direction for all the cells in the model via an array. The keyword can be used for all grid types, except for the Radial Grid geometry. The keyword effectively overwrites previously entered and calculated data. The transmissibility overwritten is the +Y face transmissibility of each grid block, that is for cell (I, J, K) the transmissibility between cells (I, J, K) and (I, J+1, K).

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	TRANY	TRANY is an array of real positive numbers assigning the transmissibility in the Y direction to each cell in the model. Repeat counts may be used, for example 20*100.0.			
		cP.rb/day/psia	cP.rm^3/day/bars	cP.rcc/hr/atm	None

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) Values not reset by this keyword remain unaltered.
3) The keyword is terminated by a "/".

Table 7.10: TRANY Keyword Description

See also the TRANX and TRANZ keywords to modify the transmissibilities in the other directions.

Example

```
--
--        DEFINE INPUT BOX FOR EDITING INPUT ARRAYS
--
--        ---------- BOX ---------
--        I1  I2   J1  J2   K1  K2
BOX
          1   1    10  10   1  120                          / DEFINE BOX AREA
--
--        SET TRANY+ TRANSMISSIBILITY
--
TRANY
          120*0.00                                         /
--
--        DEFINE END OF INPUT BOX EDITING OF INPUT ARRAYS
--
ENDBOX
```

Here the BOX statement is used to define the input grid for the TRANY keyword, which overwrites the transmissibility previously calculated with transmissibility values of zero, resulting in a no-flow boundary in that part of the field. The ENDBOX keyword resets the input box to the full grid.

7.3.71 TRANZ - DEFINE THE TRANSMISSIBILITY IN THE Z DIRECTION FOR ALL THE CELLS

Description

TRANX defines the transmissibility in the z direction for all the cells in the model via an array. The keyword can be used for all grid types, except for the Radial Grid geometry. The keyword effectively overwrites previously entered and calculated data. The transmissibility overwritten is the +Z face transmissibility of each grid block, that is for cell (I, J, K) the transmissibility between cells (I, J, K) and (I, J, K+1).

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	TRANZ	TRANZ is an array of real positive numbers assigning the transmissibility in the Z direction to each cell in the model. Repeat counts may be used, for example 20*100.0.			
		cP.rb/day/psia	cP.rm^3/day/bars	cP.rcc/hr/atm	None

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) Values not reset by this keyword remain unaltered.
3) The keyword is terminated by a "/".

Table 7.11: TRANZ Keyword Description

See also the TRANX and TRANY keywords to modify the transmissibilities in the other directions.

Example

```
--
--        DEFINE INPUT BOX FOR EDITING INPUT ARRAYS
--
--        ---------- BOX ---------
--        I1  I2   J1  J2   K1  K2
BOX
          1* 100   1* 100   20  20                          / DEFINE BOX AREA
--
--        SET TRANZ+ TRANSMISSIBILITY
--
TRANZ
          1000*0.00                                        /
--
--        DEFINE END OF INPUT BOX EDITING OF INPUT ARRAYS
--
ENDBOX
```

Here the BOX statement is used to define the input grid for the TRANZ keyword, which overwrites the transmissibility previously calculated with transmissibility values of zero, resulting in a no-flow boundary in that part of the field between layers 20 and 21. The ENDBOX keyword resets the input box to the full grid.

7.3.72 WARN – ACTIVATE WARNING MESSAGES

Turns on warning messages to be printed to the print file (*.PRT); note that this keyword is activated by default and can subsequently be switched off by the NOWARN activation keyword. The warning messages may be turned on and off using keywords WARN and NOWARN. OPM Flow always prints error messages.

See *WARN – Activate Warning Messages* in the GLOBAL section for a full description.

CHAPTER 8: PROPS SECTION

8.1 INTRODUCTION

The PROPS section is the section that contains the fluid property keywords used to define the PVT behavior of the fluids in the model and therefore the data is fluid type dependent. For example, if the oil phase has been activated in the RUNSPEC section via the OIL keyword then one of the oil PVT keywords needs to be defined in this section. The other main type of data required for the PROPS section is the saturation tables that govern how the various fluids flow in the model relative to the other phases. Again, the saturation tables are fluid type dependent.

Various other fluid and rock property data are also entered in this section including rock compressibility via the ROCK keyword for the standard rock compressibility model, and rather unusually, the Carter-Tracy influence function entered via the AQUTAB keyword.

8.2 DATA REQUIREMENTS

For all phases activated in the model a complete description of the PVT behavior (PVT tables) and fluid flow behavior (saturations tables) is required. As the data is fluid type dependent, the next two sections attempt to define the appropriate keywords for the various activated fluid types. The first section outlines the fluid PVT property requirements and the second section summarizes the keywords associated with fluid flow property data, namely the relative permeability and capillary pressure data.

8.2.1 FLUID PROPERTIES

A phase is a portion of a system which is (1) homogeneous in composition, (2) bounded by a physical surface, and (3) mechanically separable from other phases which may be present. The most important phases which occur in petroleum production are a liquid phase (oil, distillate, condensate, and water) and a vapor or gas phase (natural gas) component. Separators are used to separate the gas, water and hydrocarbon liquid phases produced from wells. Solid phases also occur in petroleum production, mainly as:

(1) paraffin or other deposits within the producing formation, in the tubing, and in the surface equipment;

(2) gas hydrates which may freeze and clog gas-flow lines and equipment; and

(3) sand or reservoir rock[137].

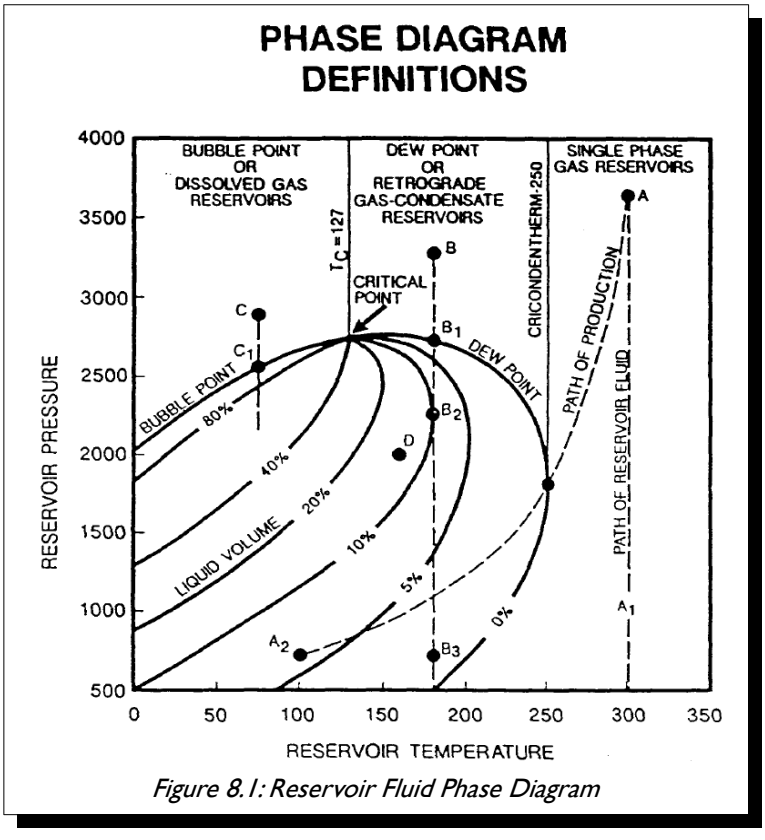


Figure 8.1: Reservoir Fluid Phase Diagram

The phase behavior of petroleum systems is represented by a Pressure-Volume-Temperature ("PVT") phase diagram, as illustrated in Figure 8.1, which shows a typical phase diagram after Clark[138].

[137] *Bradley, H. B.: "Petroleum Engineering Handbook", Society of Petroleum Engineers, Richardson, Texas, USA.*

[138] *Clark, Norman, "It Pays to Know Your Petroleum", World Oil, March and April, 1953.*

Several important definitions should be noted from Figure 8.1 that describes how a fluid behaves with respect to the composition, pressure and temperature, namely:

- **Bubble Point**: Pressure at which the first bubble of gas is formed.
- **Dew Point**: Pressure at which the last drop of liquid turns to gas.
- **Critical Point**: Point at which all properties of liquid and gas are identical, the point where the dew point and bubble point curves join.
- **Cricondentherm**: Temperature above which two phases cannot co-exist regardless of pressure.
- **Cricondenbar**: Pressure above which two phases cannot co-exist regardless of the temperature

Multi-component hydrocarbon systems are most accurately defined by their compositions, which are usually expressed as mole fractions of the components methane, ethane, propane, isobutane, n-butane, iso-pentane, n-pentane, hexanes, and heptanes and heavier (heptanes plus). The heptanes-plus fraction is usually defined by its average molecular weight, its specific gravity, and/or the fractions which boil off in selected temperature ranges. In addition to hydrocarbons, many naturally occurring petroleum systems also contain hydrogen sulphide, nitrogen, carbon dioxide, and helium in quantities which are sufficiently large to appreciably affect the phase behavior of the system (see GPA Standard 2145[139] and Standing (1952)[140] for additional information).

Gas Fluid Properties

The Ideal Gas Law is based on the above theory of gases, a mathematical equation called an Equation-of-State can be derived to express the relationship existing between pressure, volume, and temperature, for a given quantity of gas. This relationship is called the Ideal Gas Law, and is expresses mathematically as:

$$PV = nRT \tag{8.1}$$

Where,

P	=	pressure (psia or kPa).
T	=	temperature (°R = 459.67 + °F or K = 273.15 + °C).
V	=	volume of the gas (ft^3 or m^3).
n	=	number of lb moles of gas, where one lb mole is the molecular weight of the gas expressed in pounds (lb-mole or g-mol).
R	=	universal gas constant which has the value of 10.732 psia/ft^3/mole/°R or 8.314 kPa/m^3/g/mol/K.

However, the above equation of state is not applicable for **Real Gases** observed in petroleum reservoirs. In dealing with gases at a very low pressure, the ideal gas relationship (equation of state) is a convenient and accurate, within 2 - 3% at atmospheric conditions, while for pressures and temperatures found in petroleum reservoirs the physical properties calculated can lead to errors in excess of 500%. This is because real gases (natural gases) do not behavior as an ideal gas. Basically the magnitude of deviations of real gases from the conditions of the ideal gas law increases with increasing pressure and temperature, and varies widely with the composition of the gas. The reason for this is that the perfect gas law was derived under the assumption that the volume of molecules is insignificant and molecular attraction or repulsion takes place. This is not the case for real gases.

In order to express a more exact relationship between the variables P, V, and T, a correction factor called the gas compressibility factor (the gas deviation factor, or simply the Z-factor) must be introduced into the ideal gas law, that is:

[139] *Gas Processors Association, GPA, "Standard 2145 - Physical Constants of Paraffin Hydrocarbons and Other Components of Natural gas", 1986.*

[140] *Standing, M. B.:" Volumetric and Phase Behaviour of Oil Field Hydrocarbon Systems", Renihold Publishing Corp., New York City (1952).*

$$PV = ZnRT \tag{8.2}$$

Where,

P	=	pressure (psia or kPa)
T	=	temperature (°R = 459.67 + °F or K = 273.15 + °C)
V	=	volume of the gas (ft^3 or m^3)
n	=	number of lb moles of gas, where one lb mole is the molecular weight of the gas expressed in pounds (lb-mole or g-mol)
R	=	universal gas constant which has the value of 10.732 psia/ft^3/mole/°R or 8.314 kPa/m^3/g/mol/K.
Z	=	compressibility factor for the real gas

For an ideal gas, the gas compressibility factor is equal to one, while for a real gas, the Z-factor is greater or less than one depending on the pressure, temperature, and composition of the gas. The Z-factor can be calculated from the composition of the gas or from various correlations including Standing[141] and Dranchuk, P. M., Purvis, R. A., and Robinson[142].

Gas Density (ρg): Depends on composition, temperature, and pressure at which the gas is measured, it is defined as the ratio of mass (m) to volume (v), that is:

$$\rho_g = \frac{m}{V} = \frac{P \ MW}{ZRT} \tag{8.3}$$

Where MW is molecular weigh of the gas and ρg has units of lbm/ft3 and kg/m3 for field and metric units, respectively in equation (8.3). Gas density is often reported as Relative Density, that is relative to air, as per 0.65 (air-1).

Gas Formation Volume Factor (Gas Expansion Factor): Is used to relate the volume of gas, as measured at reservoir conditions, to the volume of gas as measured at standard conditions (60 oF and 14.7 psia, or 15 oC and 101.325 kPa). This gas property is then defined as the actual volume occupied by a certain amount of gas at a specified pressure and temperature, divided by the same amount of gas at standard conditions. Thus the gas formation volume factor can be expressed as:

$$E = \frac{V_{sc}}{V_i} \tag{8.4}$$

And substituting equation (8.3) into (8.4) we can derive the gas formation volume factor for a given reservoir pressure and temperature:

$$E = \left(\frac{P_i}{P_{sc}}\right)\left(\frac{T_{sc}}{T_i}\right)\left(\frac{1}{Z_i}\right) \tag{8.5}$$

Note

In oilfield units when we are dealing with oil reservoirs the gas formation volume factor is more commonly denoted as Bg and has the units of rb/scf. Whereas for gas reservoirs, the gas formation volume factor is Eg and has the units scf/rcf and is often described as the Gas Expansion Factor or GEF.

[141] *Standing, M.B. and Katz, D.L.: "Density of Natural Gases," Trans., AIME (1942) 146, 140-44.*

[142] *Dranchuk, P. M., Purvis, R. A., and Robinson, D. B., "Computer Calculations of Natural Gas Compressibility Factors Using the Standing and Katz Correlation", Institute of Petroleum Technical Series, No. IP 74-008, 1974.*

Gas Isothermal Compressibility: A knowledge of the variability of a fluid compressibility with pressure and temperature is essential in performing many reservoir engineering calculations. For a liquid phase, the compressibility is small and is usually assumed to be constant. Whereas for a gas phase, the compressibility is neither small nor constant. By definition the isothermal gas compressibility, c_g, of a substance is the change in volume per unit volume, for a unit change in pressure, i.e.

$$c_g = -\frac{1}{V}\left(\frac{\partial V}{\partial P}\right)_T \tag{8.6}$$

In general all simulators calculate the isothermal gas compressibility based on the entered PVT data, so it is not necessary to enter this variable directly. If needed the correlation of Meehan, D. N., and Lyons, W. K.[143] may be used to estimate the values.

Gas Viscosity (μg): The gas viscosity of a fluid is a measure of the internal fluid friction (resistance) to flow. If the friction between layers of the fluid is small, i.e., low viscosity, an applied shearing force will result in a large velocity gradient. As viscosity increases, each fluid layer exerts a larger frictional drag on the adjacent layers and velocity gradient increases. The viscosity of a fluid is generally defined as the ratio of the shear force per unit area to the local velocity gradient. Note that gas viscosity is not normally measured in the laboratory because it can be estimated precisely from empirical correlations. Gas viscosity increases with increasing temperature and pressure. Usually it is not measured but is obtained from the Carr, Kobayashi and Burrows[144] correlation, which include corrections for H_2S, CO_2 and N_2. For sour gases, this correlation is preferred to the Lee, Gonzalez and Eakin[145] formulation (which does not account for H_2S, CO_2 and N_2). Typically, gas viscosity is in the range of 0.015 to 0.03 cp.

Condensate-Gas Ratio **("CGR"):** Defines the number of standard cubic meters of hydrocarbon liquid that will dissolve in one stock tank meter cubed of gas at a certain pressure and temperature and has units of stb/MMscf and sm^3/sm^3 for field and metric units, respectively. Field units values range from low of 1or 2 stb/MMscf for dry gas reservoirs, up to 100 stb/MMscf for wet gas reservoirs. The empirical correlation of Meethan Vogel[146] can be used to estimate CGR.(Rv)

Water Vapor Content of Gas Water Gas Ratio ("WGR"): Gas in the reservoir is usually saturated with water vapor, but as the gas moves up the wellbore, the temperature and pressure decrease and water condenses out of the gas. Often, water that is produced at the surface is not formation water but is water of condensation. Confirmation of this is a low water-gas ratio (1 – 5 bbl/MMscf) and low salinity (fresh water less 100 ppm NaCl). The water vapor content of a gas depends on pressure, temperature, and composition of the gas. As pressure decreases, the water vapor content increases. Conversely, as temperature decreases, the water vapor content decreases. Units are stb/Mscf (although often quoted as stb/MMscf) and sm^3/sm^3 for field and metric units, respectively. Bubacek's[147] empirical correlation for sweet natural gas (methane) may be used for gases without H2S to calculate GWR or McKetta and Wehe's[148] correlation that takes into account water salinity and gas gravity. Finally Robinson et al[149] and Wichert and Wichert's[150] correlations may be used for sour natural gases (those containing H2S).

[143] *Meehan, D. N., and Lyons, W. K., "Programmable Calculations for Gas Compressibility", Oil and Gas Journal, Oct. 8, 1979.*

[144] *Carr, N.L., Kobayshi, R., and Burrows, D.B.; "Viscosity of Natural Cases Under Pressure", Trans. A.I.M.E. (1954), 201 pp. 264-272.*

[145] *Lee, A.L., Gonzalez M.H. and Eakin B.E.: "The Viscosity of Natural Gases", J.Pet. Tech. (1966) 18, pp. 997-1000.*

[146] *Meehan, D. N., and Vogel, E. L, HP-41 Reservoir Engineering Manual, Penn Well Books, 1982.*

[147] *Bukacek, R.F: "Equilibrium Moisture Content of Natural Gases," Bull., Inst. of Gas Technology Bulletin (1955).*

[148] *McKetta, J.J. and Wehe, A.H.: "Use This Chart for Water Vapor Content of Natural Gases," Petroleum Refiner (Aug. 1958) 153-54.*

[149] *J. N. Robinson et al., "Estimation of the Water Content of Sour Natural Gases", Paper Number SPE 6098, 51st Annual Fall Technical Conference, SPE, New Orleans, Oct. 3–6, 1976.*

[150] *Wichert, G. C. and Wichert, E., "Chart Estimates Water Content of Sour Natural Gas", O&GJ, March 29, 1993, pp. 61-64.*

Oil Fluid Properties

For oil and condensate fluids we have a similar set of properties; however, oil properties are nearly always measured experimentally in the laboratory, as although there are many correlations available, they generally have low predictability unless tuned with the laboratory measured data. Thus, the main oil properties are:

Oil Density: Like gas density, oil density depends on composition, temperature, and pressure at which the fluid is measured, it is defined as the ratio of mass (m) to volume (v), that is:

$$\rho = \frac{m}{V} \tag{8.7}$$

with units of lbm/ft^3, kg/m^3, and g/cm^3. Note that density is quoted in various units even within a given unit system. In oilfield (English) units it is quite common to quote density as a gradient that is psi/ft, this is because fluid gradients are a common form of in-situ measurement that can be easily related to density by:

$$\rho = 144 \; x \; fluid \;\; gradient \;\; (\text{psi/ft}) \tag{8.8}$$

For example, for an oil density measured as 0.338 psi/ft, the actual in-situ density would simply be:

$$\rho = 144 \; x \; 0.338 = 48.7 \; \frac{lbm}{ft^3} \tag{8.9}$$

Oil density can range as low as about 0.270 psi/ft (0.62 g/cc) in the Malay Basin (offshore Malaysia) to as high as 0.429 psi/ft (0.99 g/cc) in the heavy oil deposits in Cold Lake (Canada). However, most common oil densities range from around 0.78 - 0.85 g/cc (0.338 - 0.369 psi/ft). When using oil field units most engineers drop the m in the lbm units term, and therefore quote density as lb/ft^3 (or lb/cu. ft.), as lbm is implied.

Relative Density (γ_o) defines the ratio of the density of a known material to the density of reference material, at standard conditions of pressure and temperature. Standard conditions vary throughout the world, but for oil field units one normally uses14.7 psia and 60 °F, while for SI units some areas use 101.325 kPa and 15 °C . The reference material for liquids is always water, and thus ***Oil Relative Density*** is defined as:

$$\gamma_o = \frac{\rho_o}{\rho_{\rho w}} \tag{8.10}$$

And since the density of water at 14.7 psia and 60 °F is 62.4 lb/ft^3, then we have:

$$\gamma_o = \frac{\rho_o}{62.4} \tag{8.11}$$

For SI units, with density measured in kg/m^3, relative density is:

$$\gamma_o = \frac{\rho_o}{1000} \tag{8.12}$$

Commonly relative density is quoted with reference to the reference phase, for example, γ_o = 0.780 (water = 1).

The American Petroleum Institute ("API") classifies oils based on an API gravity (γ_{API}), or degrees API (°API), the relationship between relative density (γ_o) of oil and API gravity (γ_{API}) is given by:

$$\gamma_{API} = \frac{141.5}{\gamma_o} - 131.5 \quad (8.13)$$

Where,

γ_o	= oil gravity (water = 1.0)
ρ_o	= oil density (lb/ft^3 or kg/m^3)
API	= API gravity (°API)

API density is commonly reported at the well site.

Bubble Point (Saturation Pressure): The bubble point pressure, or saturation pressure (P_b or P_{sat}), is defined as the pressure at which the first bubbles of gas appear from a liquid as the pressure declines. The correlations given in the oil formation volume section (later on), also have the equivalent bubble point pressure correlations. It should be noted that the comments addressing the application of these correlations in the oil formation volume factor section, are equally applicable here.

Gas -Oil Ratio ("GOR"): The GOR, (R_s) is defined as the number of standard cubic meters of gas that will dissolve in one stock tank meter cubed of crude oil at a certain pressure and temperature. The solubility of a natural gas in a crude oil is a strong function of the pressure, the temperature, the API gravity, and gas gravity. For a particular gas and crude oil to exist at a constant temperature, the solubility increases with pressure until the saturation pressure is reached. At the saturation pressure (bubble point pressure) all the available gases are dissolved in the oil, and the gas solubility reaches its maximum value. Rather than measuring the amount of gas that will dissolve in given stock tank crude oil as the pressure is increased, it is customary to determine the amount of gas that will come out of the sample of reservoir crude oil.

Oil Formation Volume Factor (B_o): The oil formation factor, B_o, is defined as the ratio of the volume of oil (plus the gas in solution) at the prevailing reservoir pressure and temperature, to the volume of oil at standard conditions. Obviously the oil formation volume factor is always greater than or equal to one. Mathematically we can express the oil formation volume factor as:

$$B_o = \frac{V_{rc}}{V_{sc}} \quad (8.14)$$

Where:

B_o	= oil formation volume factor (rb/stb or m^3/Sm^3)
V_{rc}	= volume of oil at reservoir conditions (rb or m^3)
V_{sc}	= volume of at standard conditions (stb or Sm^3)

Values of oil FVF at reservoir temperature and various reservoir pressures can be obtained from a standard PVT analysis of a reservoir fluid sample. However, if this data is unavailable, the geologist/engineer must then resort to empirical correlations, **such as Standing[151], Vasquez and Beggs[152], or Glasø[153]. Generally Bo ranges from a low of 1.05 for heavy oils to a high of 2.5 rb/stb GOR volatile oils. Care should be exercised when applying any of the aforementioned correlations, as there is significant variation between the various correlations. In general, a given correlation may be more appropriate for a geological basin based on comparing actual measured PVT data from other fields with the correlation predicted results.**

Oil Isothermal Compressibility: The oil isothermal compressibility is defined as the rate of change in volume with pressure increase per unit volume of liquid, all variables other than pressure being constant. Thus, by definition, isothermal compressibility of a substance is defined mathematically by the following expression:

[151] *Standing, M. B.: "A Pressure-Volume-Temperature Correlation for Mixtures of California Oils and Gases", Drill, and Prod. Prac., API (1947).*

[152] *Vasquez M. and Beggs H. D.: "Correlations for Fluid Physical Property Prediction", J. Pet. Tech. (June 1980).*

[153] *Glasø, O.: "Generalized Pressure-Volume-Temperature Correlations", J.Pet.Tech. (May 1980).*

$$c_o = -\frac{1}{V}\left(\frac{\partial V}{\partial P}\right)_T \tag{8.15}$$

Where:

c_o	=	oil isothermal compressibility (1/kPa).
V	=	oil volume (m^3).
P	=	pressure (kPa).

or in terms of oil formation volume factor we can derive;

$$c_o = -\frac{1}{B_o}\frac{dB_o}{dP} \tag{8.16}$$

Where:

c_o	=	oil compressibility (1/kPa).
B_o	=	oil formation volume factor (m^3/Sm^3).

Generally, isothermal compressibility is determined from a laboratory PVT study, in cases where a study has not been performed the correlations of Vasquez and Beggs[154] or Ramey[155] may be used.

Oil Viscosity (μ_o): Crude oil viscosity is an important physical property that controls and influences the flow of oil through porous media and pipes. The viscosity, in general, is defined as the internal resistance of the fluid to flow, and oil viscosity is a strong function of the temperature, pressure, oil gravity, gas gravity, and gas solubility. Oil viscosity can be classified into three main groups, based on the nature or state of the crude with respect to pressure, these are:

- Dead Oil Viscosity — The dead oil viscosity is defined as the viscosity of the oil at atmospheric pressure and temperature conditions (no gas in solution).
- Saturated Oil Viscosity — The saturated (bubble point) oil viscosity is defined as the viscosity of oil at the bubble point pressure and reservoir temperature.
- Undersaturated Oil Viscosity — The undersaturated oil viscosity is defined as the viscosity of the oil at a pressure above the bubble point and at reservoir temperature.

Whenever possible, oil viscosity should be determined in the laboratory at reservoir conditions, and is normally included in a standard PVT analysis of the crude. However, if no data is available, the correlations of Beggs and Robinson[156] and Andrade-Guzman-Reynolds[157] may be employed.

[154] *Vasquez M. and Beggs H. D.: "Correlations for Fluid Physical Property Prediction", J. Pet. Tech. (June 1980).*

[155] *Ramey, H.J.: "Rapid Methods for Estimating Reservoir Compressibilities", J.Pet. Tech. (April 1964).*

[156] *Beggs, H. D. and Robinson J. R.: "Estimating the Viscosity of Crude Oil Systems", J. Pet. Tech. (Sept., 1975)*

[157] *Andrade, E. N. da C., Nature, 125 (1930), 309.*

Water Fluid Properties

Water fluid properties are similar to the oil fluid properties, and include:

Water Density (ρw): Water density as the same definition as oil density, equation (8.7), with units of lbm/ft^3, kg/m^3, and g/cm$_3$. Fresh water density at 14.7 psia and 60 °F is 62.4 lb/ft^3, and for SI units at 101. 3 kpa and 15 °C, the density is 1000 kg/m^3. Note that density is quoted in various units even within a given unit system. In oilfield (English) units it is quite common to quote density as a gradient that is psi/ft. when laboratory data or actual water samples are unavailable. The density of formation water at reservoir conditions can be estimated roughly (usually to within +/−10%) from correlations such as McCain's[158] and [159] correlations.

Water Formation Volume Factor (Bw): The water formation volume factor has the same definition as for the oil formation volume factor. Normally this property is estimated from correlations based on salt content, for example the Numbere et al[160] correlation.

Water Viscosity (μw): Again, the definition is the same as for oil and is normally estimated from correlations based on salt content, for example by the Meehan[161]

8.2.2 CLASSIFICATION OF RESERVOIRS BY FLUID TYPE

Reservoir classification by fluid type includes the following reservoir fluids:

- Low Shrinkage Oil Reservoirs.
- High Shrinkage Oil Reservoirs.
- Volatile Oil Reservoirs.
- Dry Gas Reservoirs.
- Wet Gas Reservoirs.
- Retrograde Condensate Gas Reservoirs.

The following sections briefly outline the various reservoir types based on their fluid properties.

[158] *McCain, W.D. Jr.: McCain, W.D. Jr. 1990. The Properties of Petroleum Fluids, second edition. Tulsa, Oklahoma: PennWell Books.*

[159] *Cain Jr., W.D. 1991. Reservoir-Fluid Property Correlations-State of the Art (includes associated papers 23583 and 23594). SPE Res Eng 6 (2): 266-272. SPE-18571-PA. http://dx.doi.org/10.2118/18571-PA.*

[160] *Numbere, D., Brigham, W. E., and Standing, M. B., Correlations for Physical Properties of Petroleum Reservoir Brines, Petroleum Research Institute, Stanford University, November, 1977).*

[161] *Meehan, D. N., "Estimating Water Viscosity at Reservoir Conditions", Petroleum Engineer, July 1980.*

Low Shrinkage Oil Reservoirs

The phase diagram for a typical low shrinkage oil is shown in Figure 8.2 (after Clark[162]), in this case the two-phase region covers a wide range of pressure and temperature (the critical temperature is above the reservoir temperature). The line marked 1 and 2 shows the effect of pressure reduction, with constant reservoir temperature, as the reservoir is produced. The dashed line shows the conditions encountered as the fluid travels up the tubing string and into the surface facilities.

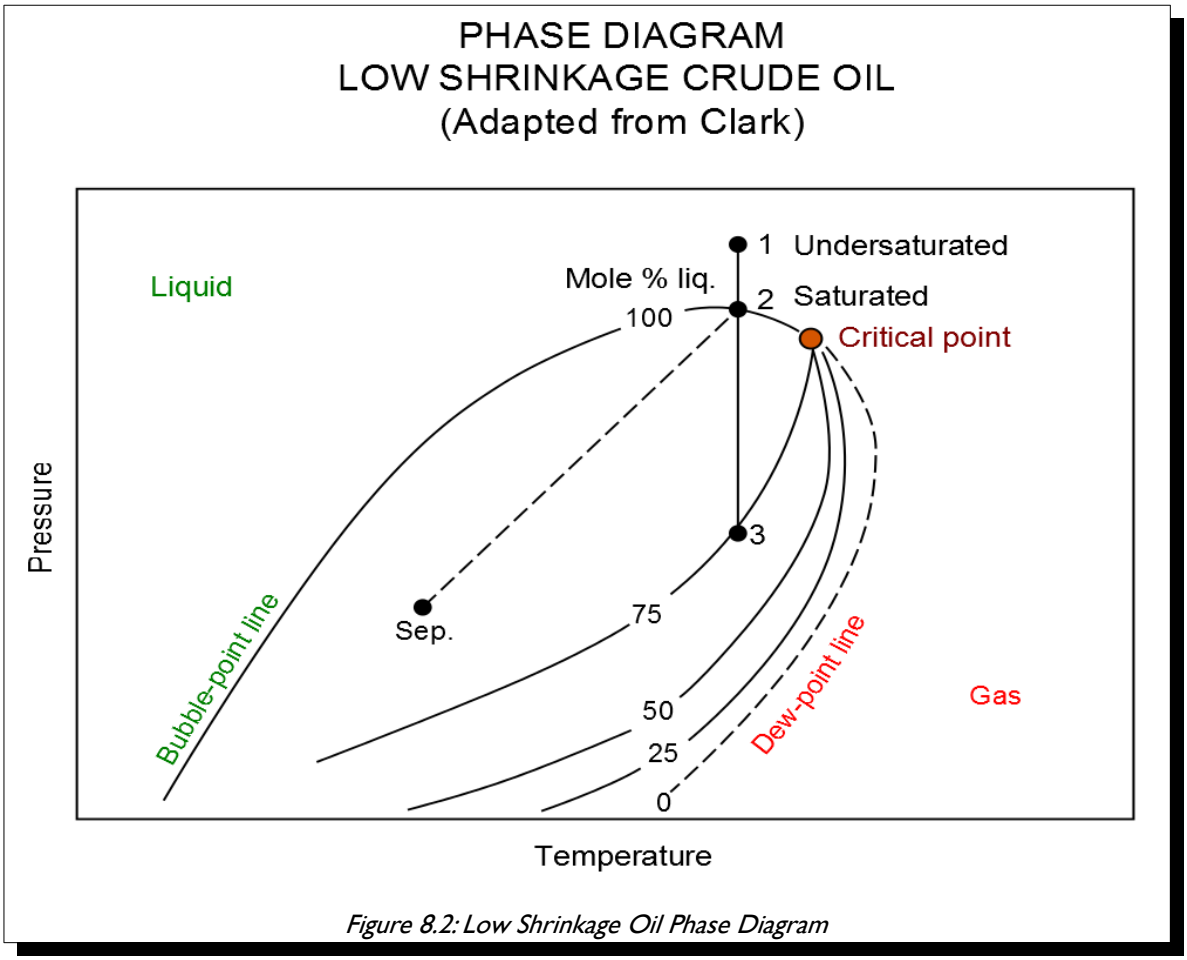


Figure 8.2: Low Shrinkage Oil Phase Diagram

If the reservoir pressure and temperature is at point 2 on the phase diagram, then the oil is at bubble point pressure and is said to be saturated, i.e., the oil contains the maximum amount of dissolved gas at the given conditions. However, if the reservoir conditions are at point 1 on the phase diagram, then the oil is said to be undersaturated. That is the oil is able to dissolve more gas at the current reservoir pressure and temperature.

The term low shrinkage oil, is derived from the fact that oil shrinkage is small, i.e., with an oil formation volume factors less than 1.5. This oil can be described as having a broad based phase envelope, high percentage of liquid, high proportion of heavier hydrocarbons, GOR's less than 500 scf/stb or 100 Sm^3/m^3, oil gravity 30 °API or heavier, and the stock tank liquid being black or a deep color. If the oil has a very low GOR and therefore has an oil formation volumes factor close to one, for as observed in heavy oil reservoirs, then it common to model this type of reservoir neglecting the gas phase. That is these type of reservoir fluids are typical modeled as two-phase oil-water system in black-oil formulations, which results in greater computational efficient due to having only two phases as oppose to three phases in an oil-gas-water system.

[162] *Clark, Norman, "It Pays to Know Your Petroleum", World Oil, March and April, 1953.*

High Shrinkage Oil Reservoirs

A high shrinkage oil reservoir fluid's phase diagram is shown in Figure 8.3.

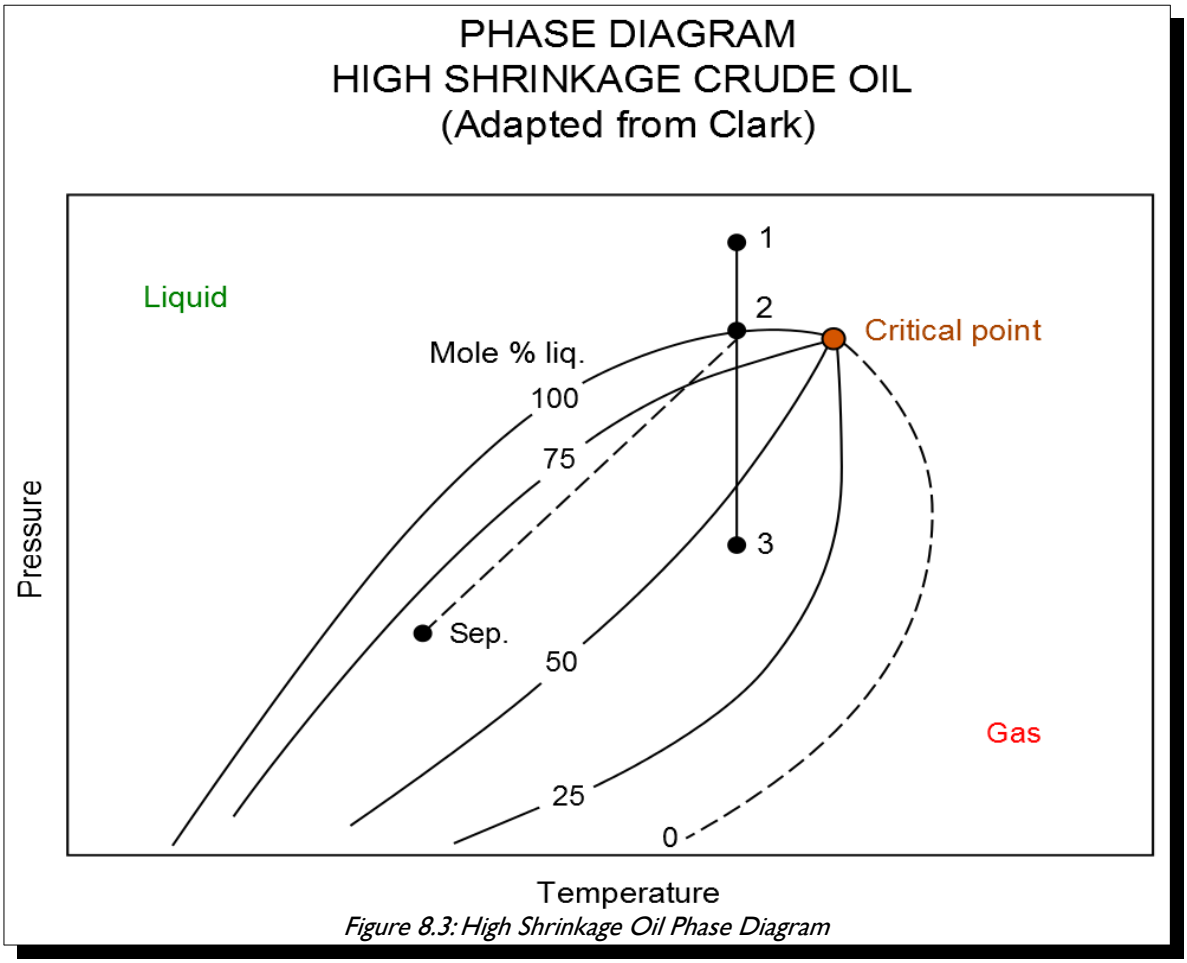


Figure 8.3: High Shrinkage Oil Phase Diagram

This type of reservoir fluid is characterized by not so broad a phase envelope, fewer heavier hydrocarbons, deep colored stock tank fluid, gravity less than 50 °API, GOR less than 8000 scf/stb or 1500 Sm^3/m^3, and oil formation volume factors greater than 1.5 Sm^3/m^3.

Dry Gas Reservoirs

Natural gas that occurs in the absence of condensate or liquid hydrocarbons, or gas that had condensable hydrocarbons removed, is called dry gas. It is primarily methane with some intermediates. The hydrocarbon mixture is solely gas in the reservoir and there is no liquid (condensate surface liquid) formed either in the reservoir or at surface.

The phase diagram for a dry gas reservoir is illustrated in Figure 8.4.

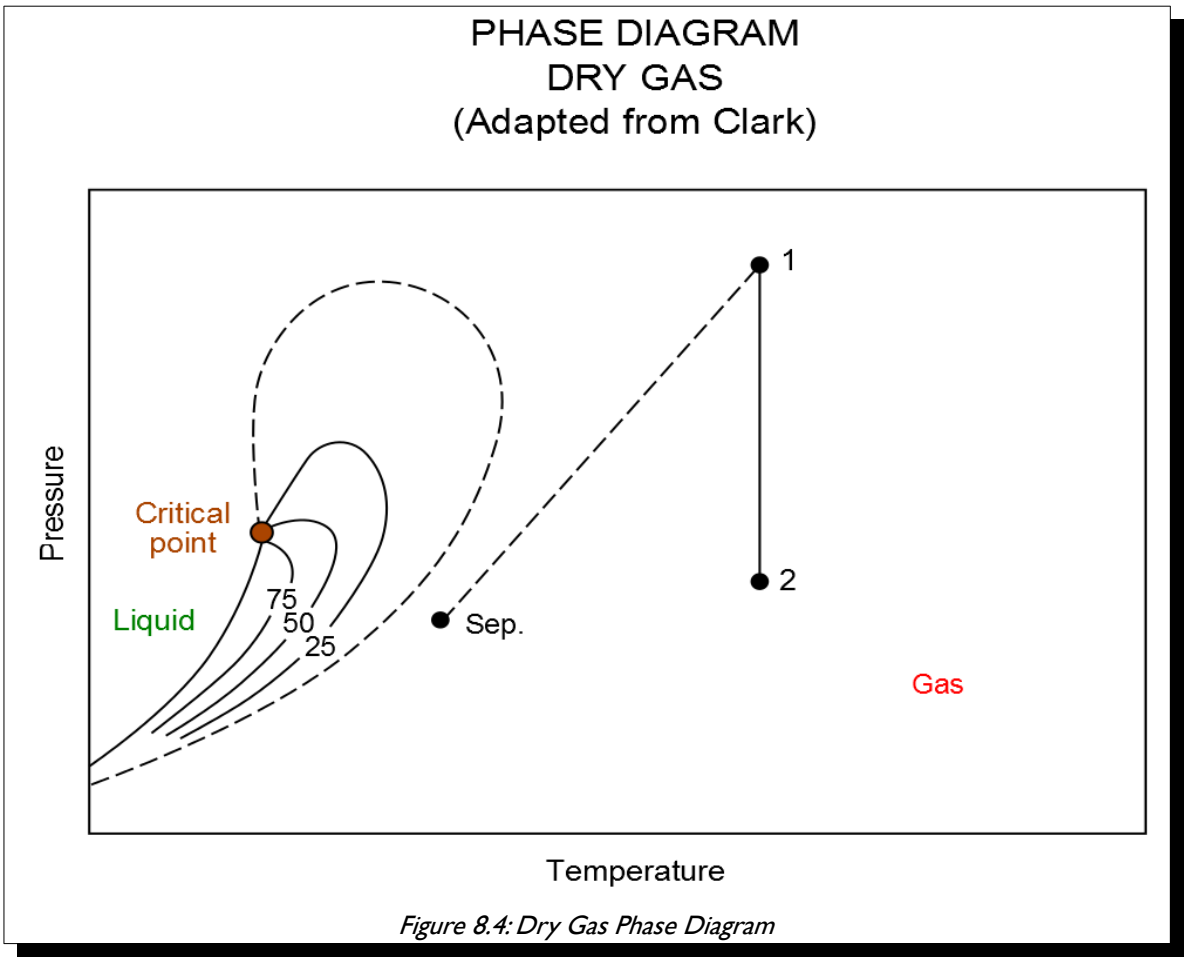


Figure 8.4: Dry Gas Phase Diagram

Notice that the separator conditions lie outside the phase envelope in the gas region. The term dry indicates that the gas does not contain heavier hydrocarbons to form liquids at the surface conditions. Dry gas typically has GOR's greater than 100,000 scf/stb or 18,000 Sm^3/m^3.

These type of reservoir fluids are typically modeled as two-phase gas-water systems in black-oil formulations, which results in greater computational efficient due to having only two phases as oppose to three phases in an oil-gas-water system. Also it is not uncommon to model dry gas reservoirs using analytical models, for example material balance, as in general the reservoirs behave like "tanks" either with or without aquifer influx.

Wet Gas Reservoirs

Natural gas that contains significant heavy hydrocarbons such as propane, butane and other liquid hydrocarbons is known as wet gas or rich gas. The general rule of thumb is if the gas contains less methane (typically less than 85% methane) and more ethane, and other more complex hydrocarbons, it is labelled as wet gas.

As can be seen from the figure below, wet gas exists in the gas phase in the reservoir throughout depletion of the reservoir pressure. However, the separator conditions lie in the two-phase region, resulting in liquid production (condensate).

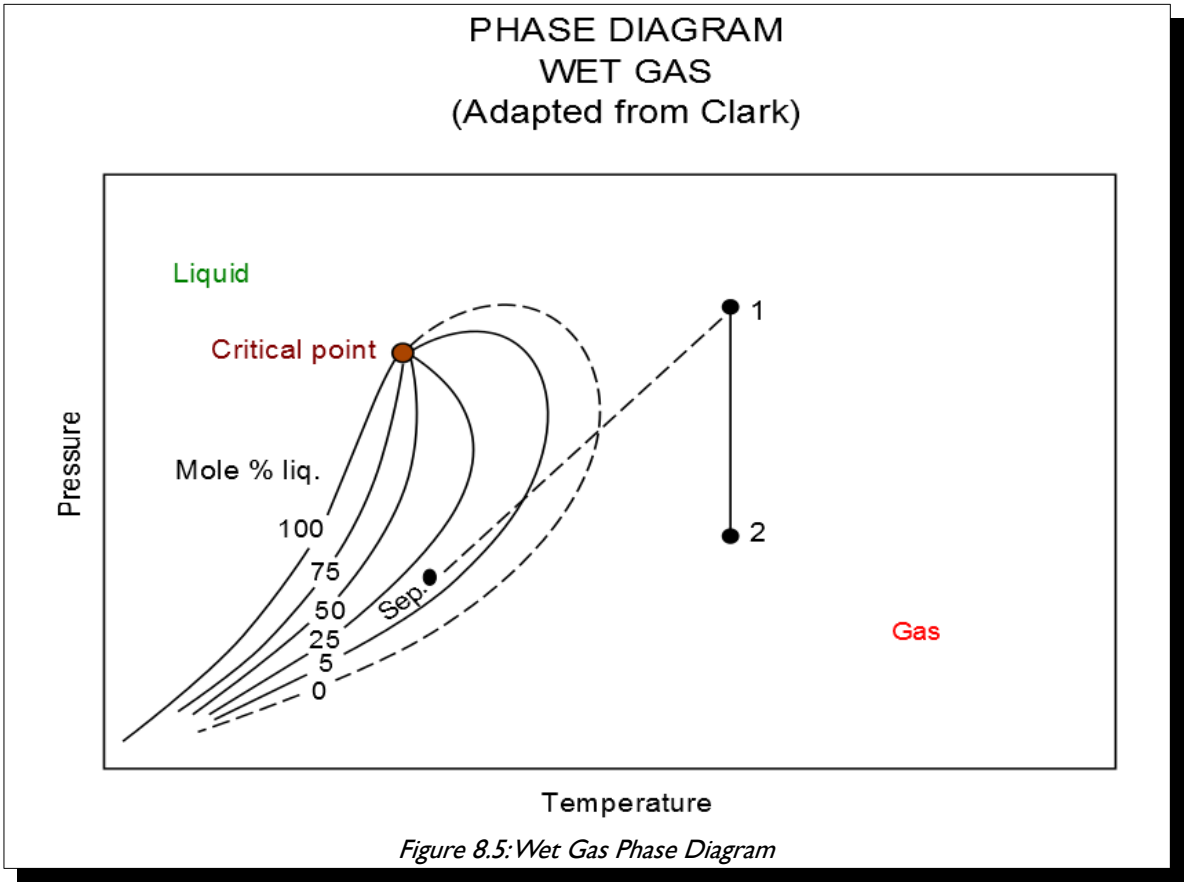


Figure 8.5: Wet Gas Phase Diagram

Wet gas normally has GOR's less than 100,000 scf/stb or 18,000 Sm^3/m^3, with the condensate having a gravity greater than 50 °API.

Retrograde Condensate Gas Reservoirs

If the reservoir temperature lies between the critical point and the cricondentherm a retrograde reservoir exists. As the pressure declines to point 2, which is the dew point, liquids begin to form in the reservoir. As the pressure further declines to point 3, the liquid yield increases. Further reduction in the pressure causes the liquid to vaporize back into the gas phase.

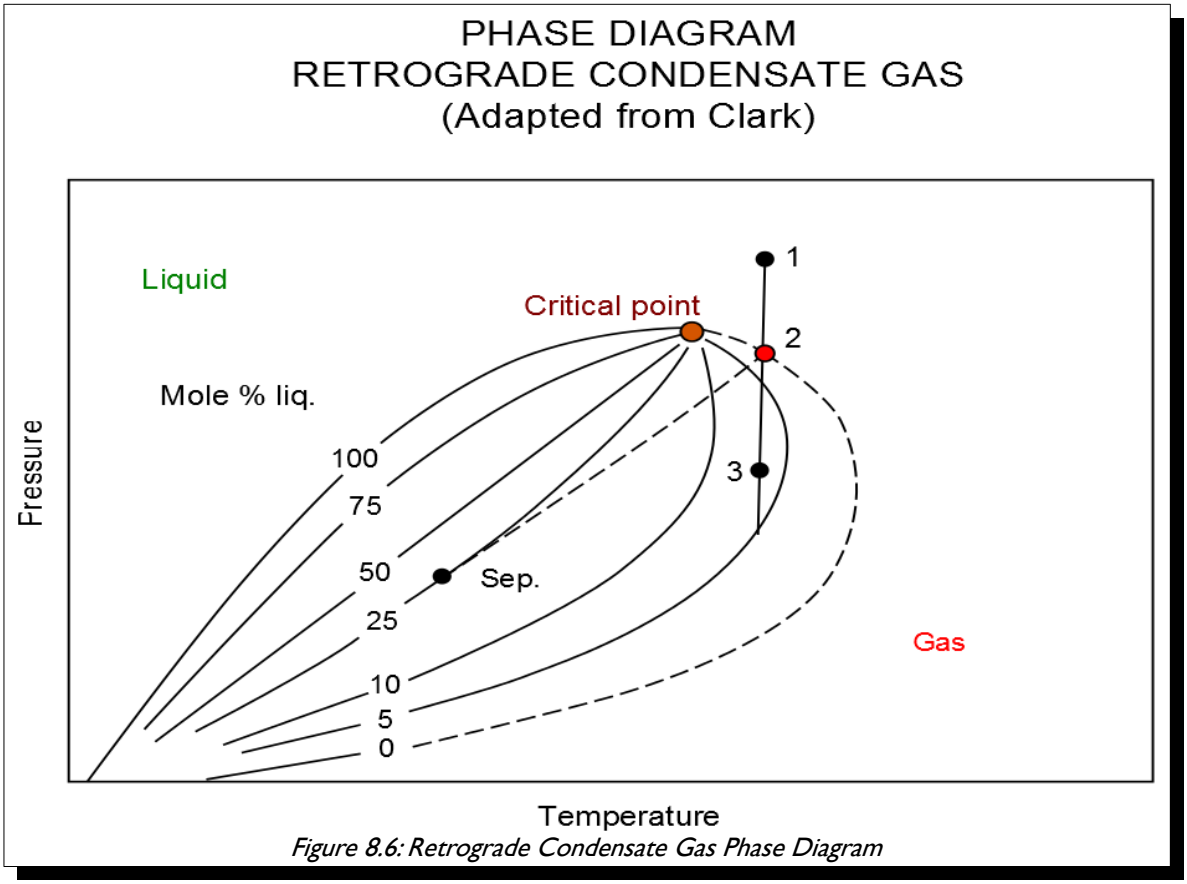


Figure 8.6: Retrograde Condensate Gas Phase Diagram

The liquid drop out in the reservoir can cause a well's productivity to be reduced, due to the system moving from a two-phase relative permeability effect, to a three phase effect. This is normally diagnosed by the well having a high skin factor.

This type of fluid has more lighter hydrocarbons than high shrinkage oils, and fewer heavier hydrocarbons than high shrinkage oils. The gravity can be as high as 60 °API, with GOR's up to 70,000 scf/stb or 13,000 Sm^3/m^3. Generally the stock tank liquid is water-white, or slightly colored.

Reservoir Classification

The following tables outline some general parameters used to classify reservoir fluids. Table 8.1 outlines some general producing characteristics used to define reservoir fluids.

Item	Crude Oil	Volatile Oil	Gas Condensate	Gas
Initial Separator Gas-Liquid Ratio, SCF/STB	<2000	2000 - 6000	6000 - 100,000 Generally 7,000 – 15,000	> 100,000 Usually below 3000,00
Color of Produced Liquid	Black to Light Green	Dark Straw Colored	Colorless to Light Straw Colored	Colorless
Usual API Gravity Range of Produced Fluid	10-45+	40-50	45-65	--
Composition of Produced Fluid	C_{7+}: usually > 40%	C_{7+}: 10-40%	C_{7+}: 2-10%	Primarily Methane
Oil Formation Volume Factor, RB/STB	< 2.0 frequently < 1.5	>2.0 often > 2.5		

Table 8.1: General Reservoir Fluid Classification

In terms of of fluid composition reservoir fluids can be indicated by the values shown in Table 8.2 below.

Item	Gas	Gas Condensate	Volatile Oil	Crude Oil
Methane (mole%)	96	87	64	49
Ethane (mole %0	3	4	8	3
Heptanes Plus (mole %)	1	3	15	42
Solution GOR (scf/stb)	105,000	18,200	2,000	625
Stock Tank Oil °API	68	61	50	34

Table 8.2: Reservoir Fluids Composition Indicators

in addition reservoir oil is further divided into Light, Medium, Heavy, Extra Heavy and Bituman (Wilmon[163]) as shown in Table 8.3 below:

Temperatue	STOCK TANK OIL °API	In Situ Viscosity (cp)
Light	>31.1	
Medium	22.3 – 31.1	
Heavy	10.0 – 22.3	100 to 10,000
Extra Heavy	<10.0	<10,000
Bitumen	<10.0	>10,000

Table 8.3: Oil Sub-Classification (after Wilmon)

[163] *Wilmon, G. J. : "Economic Outlook for Extra-heavy Oils, Natural Bitumens and Shale Oils", World Petroleum Congress (1987).*

8.2.3 FLUID PROPERTY TABLES

Table 8.4 outlines the oil, gas and water fluid types that can be active in the model, together with the related RUNSPEC section keywords that activate the phases, versus the PVT keywords that can be used to define the PVT behavior. The table also includes several water types, the standard water phase used in most simulations models, the Brine water phase used in the Brine Tracking model to track the flow of brine through the simulation grid and the effect of brine on reservoir performance, and finally OPM Flow's Vaporized Water phase that is used in the simulator's Salt Precipitation model. The latter water phase is not available in the commercial simulator.

Fluid Property Keywords Versus Oil, Gas & Water Fluid Types								
Item	**Oil**		**Gas**				**Water**	
Fluid Type	**Dead Oil**	**Live Oil**	**Dry Gas**	**Wet Gas**	**Dry / Wet Gas with Vaporized Water**	**CO_2 (EOR)**	**Water**	**Brine**
RUNSPEC Keywords	OIL	OIL	GAS	GAS	GAS	GAS	WATER	WATER
		DISGAS		VAPOIL	None/ VAPOIL			
					VAPWAT5			
					WATER			
					BRINE			BRINE
Pressure Dependent PVT	PVCDO	PVCO	PVDG	PVTG	PVTWSALT	PVTSOL	PVTW	PVTWSALT
	PVDO	PVTO	PVZG		PVTGW5	PVTDS		
					PVTGWO5			
Miscellaneous	RSCONST		RVCONST		RWGSALT5			
	RSCONSTT		RVCONSTT					
Surface Density	DENSITY				BDENSITY		DENSITY	BDENSITY
	GRAVITY						GRAVITY	

Notes:

1) Cells colored in gray with no mnemonic indicate that there is no keyword for this combination.
2) Cells colored orange show keywords that have not been implemented in OPM Flow for the given fluid type.
3) When two or more keywords are stated for the RUNSPEC keyword for a given fluid type, then all are required to define the given phase.
4) When two keywords are stated for the Pressure Dependent PVT and Miscellaneous data for a given fluid type, then either one can be used to define the PVT behavior for the given phase.
5) OPM Flow specific keywords.

Table 8.4: Fluid Property Keywords versus Oil, Gas and Water Fluid Type

For the Dead Oil phases the RSCONST and RSCONSTT keywords are used to set a constant gas-oil ratio (Rs). In this case the Rs is independent of the reservoir pressure and Rs is also negligible, as in for example heavy oil type fluids.

Similarly for the Dry Gas phase, where the RVCONST and RVCONSTT keywords are used to set a condensate-gas ratio (Rv) which is independent of the reservoir pressure and is also negligible, as in for example dry gas type fluids.

For the Vaporized Water phase and model, both dry and wet gas can be incorporated. Note that the PVTGW, PVTGWO, RWGSALT, and VAPWAT keywords are OPM Flow specific keywords.

In addition for the Brine phase, then either the SALT or SALTVD keywords in the SOLUTION section should be used to define the initial equilibration salt concentration for the model.

CO_2 can either be used as Enhanced Oil Recovery ("EOR") fluid by injecting the CO2 into an oil reservoir, or for modeling CO_2 storage. For the former, CO_2 is declared via the GAS keyword in the RUNSPEC and the PVT data is entered via the standard gas fluid properties for the hydrocarbon gas and either the PVTDS or the PVTSOL keywords are used to describe the interaction of the in situ oil and the injected CO_2.

In addition to the above the ROCK keyword should be used to define the rock compressibility. 7

Similarly, Table 8.5 outlines the fluid property data keywords for the CO2 storage, foam, polymer and solvent phases. Note that for these phases multiple keywords can be used to define the desired property behavior.

Fluid Property Keywords Versus CO2 (Storage), Foam, Polymer, Solvent, And MICP Fluid Types					
Fluid Type	**CO_2 (Storage)**	**Foam**	**Polymer**	**Solvent**	**MICP[7]**
RUNSPEC Keywords	CO2STORE	FOAM	POLYMER	SOLVENT	MICP
	GAS				WATER
	WATER				
Pressure Dependent PVT	Not required, calculated via correlations			PVDS	PVTW
				PVTSOL6	
Surface Density				SDENSITY	DENSITY
Miscellaneous	SALINITY5	FOAMADS	PLMIXPAR		MICPPARA
		FOAMDCYO	PLYADS		
		FOAMDCYW	PLYADSS		
		FOAMFCN	PLYATEMP		
		FOAMFRM	PLYCAMAX		
		FOAMFSC	PLYDHFLF		
		FOAMFSO	PLYESAL		
		FOAMFST	PLYKRRF		
		FOAMFSW	PLYMAX		
		FOAMMOB	PLYMWINJ8		
		FOAMMOBP	PLYRMDEN		
		FOAMMOBS	PLYROCK		
		FOAMOPTS	PLYSHEAR		

Fluid Property Keywords Versus CO2 (Storage), Foam, Polymer, Solvent, And MICP Fluid Types					
Fluid Type	**CO_2 (Storage)**	**Foam**	**Polymer**	**Solvent**	**MICP[7]**
		FOAMROCK	PLYSHLOG		
			PLYTRRF		
			PLYTRRFA		
			PLYVISC		
			PLYVISCS		
			PLYVISCT		
			PLYVMH8		
			PLYVSCST		
			SKPRPOLY8		
Miscellaneous			SKPRWAT8		

Notes:

1) Cells colored in gray with no mnemonic indicate that there is no keyword for this combination.
2) Cells colored orange show keywords that have not been implemented in OPM Flow.
3) When two or more keywords are stated for the RUNSPEC keyword for a given fluid type, then all are required to define the given phase.
4) When multiple keywords are stated for the Miscellaneous data for a given fluid type, then several keywords can be used to defined the fluid property behavior.
5) SALINITY is OPM Flow specific keyword, that defines the salinity for all cells in the model.
6) PVTSOL is used to model CO_2 interaction with in situ oil only and is an OPM Flow specific keyword.
7) OPM Flow's implementation of the Microbial Induced Calcite Precipitation ("MICP") model used to investigate CO_2 leakage remediation. All the keywords are specific to OPM Flow. This model also requires the ROCK keyword to define the rock compressibility.
8) OPM Flow has an additional formulation to the standard polymer flooding model, compared to the commercial simulator, known as the Polymer Molecular Weight Transport option, that uses the polymer molecular weight in calculating the polymer viscosity. This model is activated via the POLYMER and POLYMW keywords in the RUNSPEC section. The model does not account for non-Newtonian flow; the apparent viscosity is simply set equal to the zero-shear viscosity. Secondly, the standard polymer property data keywords: PLYROCK, PLYADS, PLYMAX, etc., are still required to fully describe the polymer fluid.

Table 8.5: Fluid Property Keywords versus CO2 (Storage), Foam, Polymer, Solvent, and MICP Fluid Types

As mentioned previously, CO2 can either be used as an EOR fluid by injecting the gas into an oil reservoir, or for modeling CO2 storage. For the latter, CO2 is declared via the GAS and CO2STORE keywords in the RUNSPEC section, and, except for the SALINITY keyword, all the remaining PVT data is calculated automatically via correlations by the simulator. A full description of the underlying PVT models is described by Sandve at al.[164]

Typical live oil and dry gas PVT data from the Volve[165] field is shown in Figure 8.7 and Figure 8.8, respectively. To fully define the live oil, dry gas, and water PVT properties for Volve the DENSITY, PVDG, PVTO, and PVTW keywords are employed.

[164] *Tor Harald Sandve1, Sarah E. Gasda, Atgeirr Rasmussen, and Alf Birger Rustad. Convective dissolution in field scale CO_2 storage simulation using the OPM Flow simulator. Submitted to TCCS 11 – Trondheim Conference on CO_2 Capture, Transport and Storage Trondheim, Norway – June 21-23, 2021.*

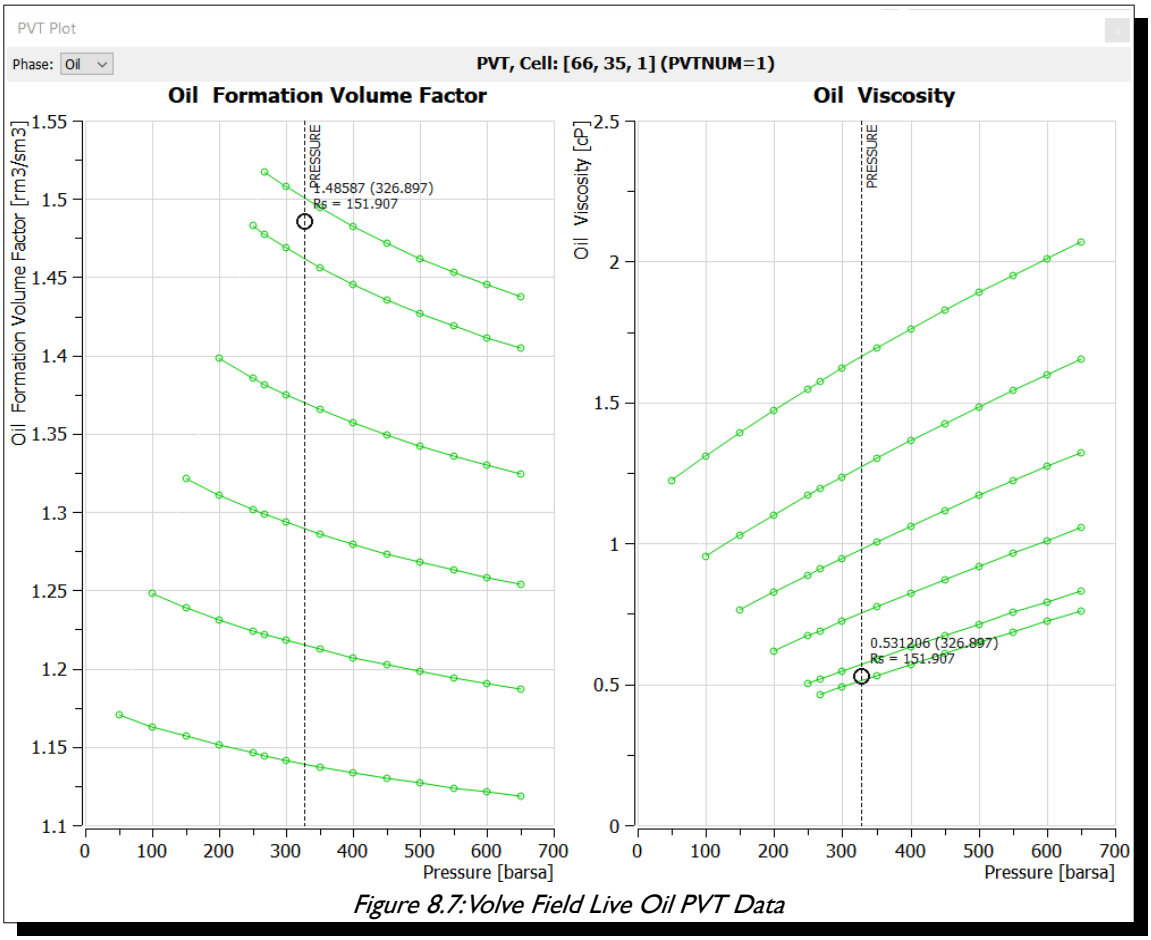


Figure 8.7: Volve Field Live Oil PVT Data

[165] *The Volve Data was approved for data sharing in 2018 by the initiative of the last Operating company, Equinor and approved by the license partners ExxonMobil E&P Norway AS and Bayerngas Norge AS in the end of 2017.*

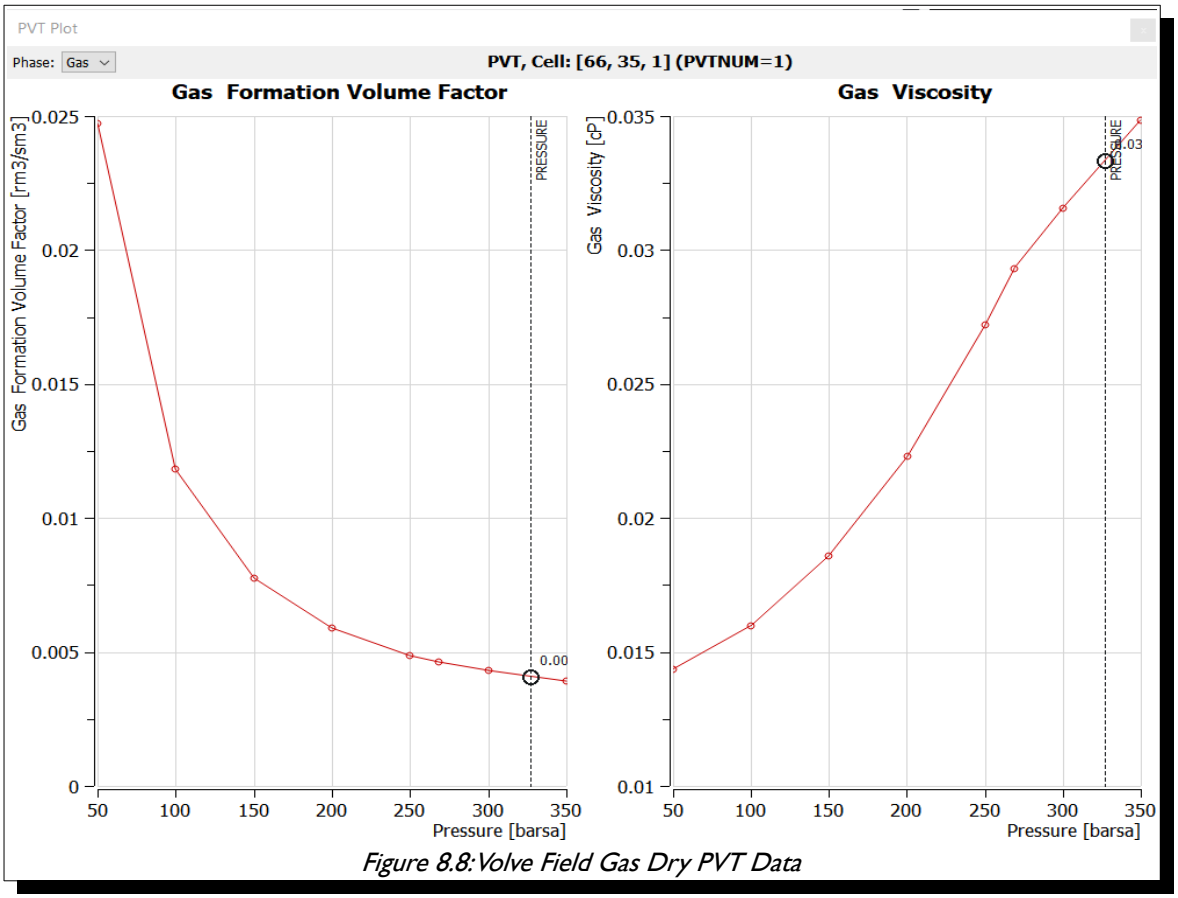


Figure 8.8: Volve Field Gas Dry PVT Data

8.2.4 SATURATION TABLES (RELATIVE PERMEABILITY AND CAPILLARY PRESSURE TABLES)

Saturation tables contain the relative permeability and capillary pressure data as a function of fluid saturation and are used to both initialize the model and to describe multi-phase flow in the reservoir. Multiple saturation tables can be entered and allocated to various areas in the model, based on rock typing. Alternatively, a limited number of saturation tables may be entered and allocated by region and combined with end-point scaling option to enable a more robust reservoir rock characterization.

Note

Confusion abounds with the terms: critical water saturation, connate water saturation, and irreducible water saturation definitions, as the terms are extensively used interchangeably to define the maximum water saturation at which the water phase will remain immobile. For reference:

Connate Water Saturation: Water trapped in the interstices of the sediments at the time of deposition, as opposed to water that migrated into the formations after deposition[166]. Normally the nomenclature of S_{wc} is used to describe this type of water saturation.

Irreducible Water Saturation: The fraction of the pore volume occupied by water in a reservoir at maximum hydrocarbon saturation. It represents water that has not been displaced by hydrocarbons because it is trapped by adhering to rock surfaces, trapped in small pore spaces and narrow interstices, etc. Irreducible water saturation is an equilibrium situation. It differs from residual water saturation, the value measured by core analysis, because of filtrate invasion and the gas expansion that occurs when a core is removed from the bottom of the hole to the surface. Also called immovable water[167]. Here the nomenclature for irreducible water saturation is $S_{wirr.}$

Critical Water Saturation: Defines the largest water saturation for which the water relative permeability is zero.

In addition to the above definitions, the simulator uses the term SWL, which is lowest water saturation in a relative permeability table. Thus, SWL can represent either the connate water saturation or the irreducible water saturation in the relative permeability tables and functions.

Secondly, the term irreducible water saturation is replaced by term critical water saturation in the relative permeability tables and function, as both terms are equivalent with regards to the mobility of the water phase. This terminology is also consistent with the critical values of the oil and gas phases, which are dependent on the displacing phase.

Thirdly, it is not uncommon for SWL to be set equal to the critical water saturation; however, care should be taken in this instance if end-point scaling is being used to ensure that the cell scaled relative permeability curves are as one might expect.

A typical oil-water relative permeability set of curves is shown in Figure 8.9 indicating the oil end-point data (KRO, KRORW and (1 – SOWCR)) and the water end-point data (KRWR, KRW. SWL and SWCR).

[166] *https://wiki.seg.org/wiki/Dictionary:Connate_water*

[167] *https://wiki.seg.org/wiki/Dictionary:Irreducible_water_saturation*

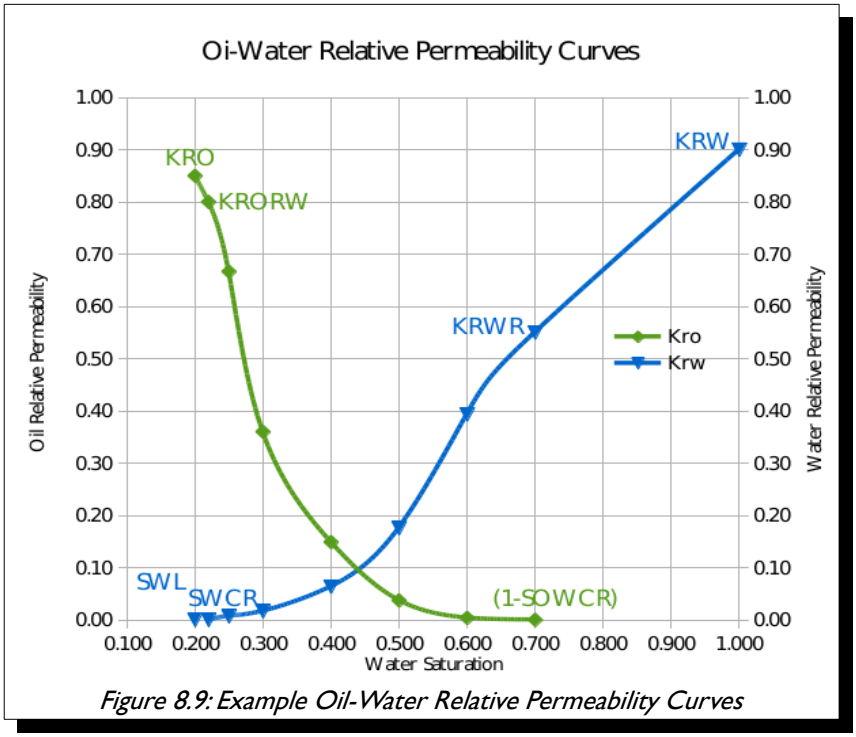


Figure 8.9: Example Oil-Water Relative Permeability Curves

The associated oil-water end-point definitions are outlined in the following table:

Type	End-Point Keyword	Oil-Water End-Point Definitions
Saturation	SOWCR	Critical oil-in-water saturation, that is the largest oil saturation for which the oil relative permeability is zero in an oil-water system.
	SWL	Connate water saturation, that is the smallest water saturation in a water saturation function table.
	SWCR	Critical water saturation, that is the largest water saturation for which the water relative permeability is zero.
	SWU	Maximum water saturation in a water saturation table. This is commonly set equal to one, unless there is a residual oil saturation below the defined OWC.
Relative Permeability	KRO	Relative permeability of oil at the maximum oil saturation.
	KRORW	Relative permeability of oil at the critical water saturation
	KRW	Relative permeability of water at the maximum water saturation (normally the maximum water saturation is one).
	KRWR	Relative permeability of water at the residual oil saturation or the residual gas saturation in a gas-water run.
Capillary Pressure	SWLPC	Capillary pressure connate water saturation, that is the smallest water saturation in a water saturation function table to be scaled independently from the SWL relative permeability end-point value.

Table 8.6: Oil-Water Relative Permeability End-Point Data Definitions

Similarly for gas-oil systems, Figure 8.10 illustrates a typical gas-oil relative permeability set of curves indicating the oil end-point data (KRORG and (1 – SOGCR)) and the gas end-point data (KRGR, KRG and SGCR).

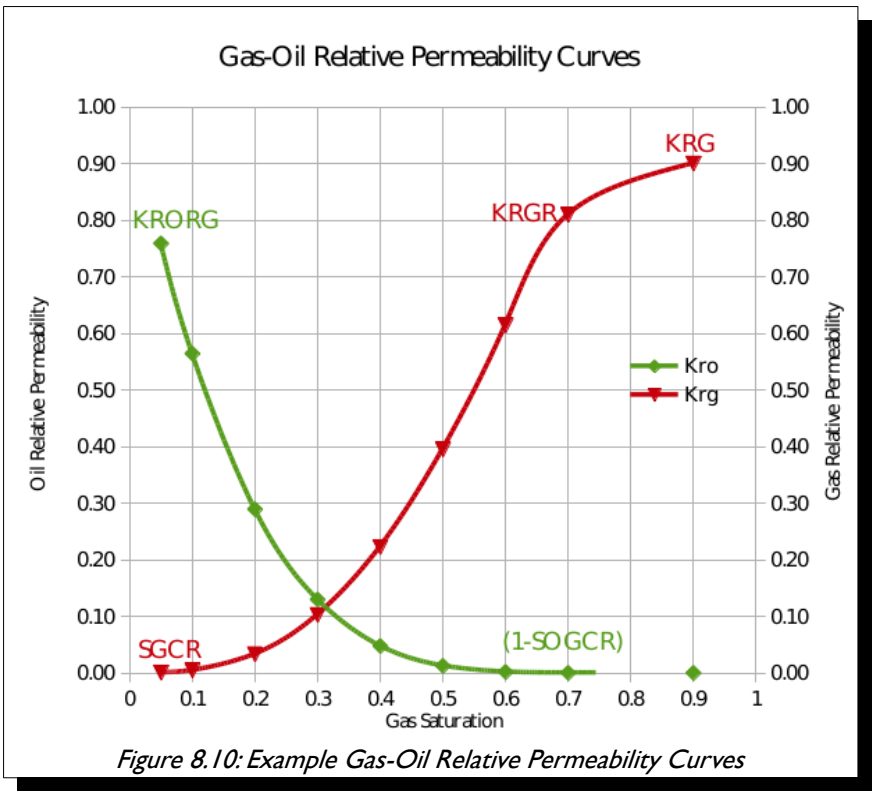


Figure 8.10: Example Gas-Oil Relative Permeability Curves

The gas-oil end-point definitions are outlined in the following table:

Type	End-Point Keyword	Gas-Oil End-Point Definitions
Saturation	SGCR	Critical gas saturation, that is the largest gas saturation for which the gas relative permeability is zero.
	SGL	Connate gas saturation, that is the smallest gas saturation in a gas saturation function table.
	SOGCR	Critical oil-in-gas saturation, that is the largest oil saturation for which the oil relative permeability is zero in an oil-gas-connate water system.
Relative Permeability	KRG	Relative permeability of gas at the maximum gas saturation.
	KRGR	Relative permeability of gas at the residual oil saturation or the critical water saturation in a gas-water run.
	KRORG	Relative permeability of oil at the critical gas saturation.
Capillary Pressure	SGLPC	Capillary pressure connate gas saturation, that is the smallest gas saturation in a gas saturation function table to be scaled independently from the SGL relative permeability end-point value.

Table 8.7: Gas-Oil Relative Permeability End-Point Data Definitions

Finally, for two phase gas-water systems, Figure 8.11 illustrates a typical gas-water relative permeability set of curves indicating the gas end-point data (KRG, KRGR and (1 – SGWCR)) and the water end-point data (KRW, KRW, SWL and SWCR).

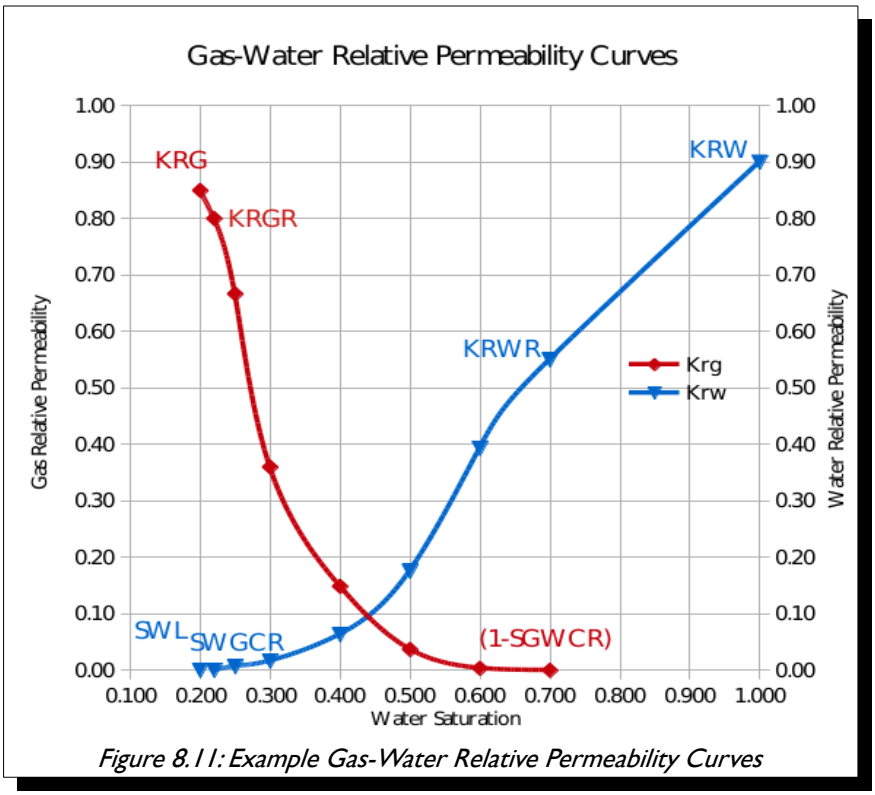


Figure 8.11: Example Gas-Water Relative Permeability Curves

The gas-oil end-point definitions are outlined in the following table:

Type	End-Point Keyword	Gas-Water End-Point Definitions
Saturation	SGL	Connate gas saturation, that is the smallest gas saturation in a water saturation function table.
	SGU	Maximum gas saturation in a gas saturation table.
	SGWCR	Critical gas-in-water saturation, that is the largest gas saturation for which the gas relative permeability is zero in a gas-water system.
	SWL	Connate water saturation, that is the smallest water saturation in a water saturation function table.
	SWCR	Critical water saturation, that is the largest water saturation for which the water relative permeability is zero.
	SWU	Maximum water saturation in a water saturation table.
Relative Permeability	KRG	Relative permeability of gas at the maximum gas saturation.
	KRGR	Relative permeability of gas at the residual oil saturation or the critical water saturation in a gas-water run.
	KRWR	Relative permeability of water at the residual oil saturation or the residual gas

Type	End-Point Keyword	Gas-Water End-Point Definitions
		saturation in a gas-water run.
Capillary Pressure	SWLPC	Capillary pressure connate water saturation, that is the smallest water saturation in a water saturation function table to be scaled independently from the SWL relative permeability end-point value.

Table 8.8: Gas-Water Relative Permeability End-Point Data Definitions

End-point scaling is activated in the RUNSPEC section with the ENDSCALE keyword and the data used to apply end-point scaling is entered in the PROPS section using the end-point keywords defined in Table 8.6 and Table 8.7 to define each grid block's end-point data. There is also direction dependent versions of the keywords for when directional end-point scaling has been activated. For example for critical water saturation, SWCR is used with non-direction end-point scaling and the SWCRX± , SWCRX± and SWCRX± series of keyword is used for when directional end-point scaling has been activated. In addition, there is also the facility to incorporate end-point scaling based on the drainage and / or imbibition process which again can be either non-directional or directional.

Note

If the hysteresis model option has been activated on the SATOPTS keyword in the RUNSPEC section, then the equivalent imbibition arrays suffixed with the letter I, for example IKRORW, can be used to define the KRORW for the relative permeability imbibition tables.

Consequently, if the hysteresis model option has not been activated, then the standard keywords are used to define the end-point scaling parameters, that is KRORW should be used and not IKRORW, even though this is usually considered an imbibition process with the wetting phase (normally water) increasing.

Saturation functions can be entered via several keywords consisting of two format types as depicted in the following table:

Format Type One				Format Type Two			
Keyword	**Oil**	**Gas**	**Water**	**Keyword**	**Oil**	**Gas**	**Water**
SGOF	Pcog			SGFN3		Pcog	
SLGOF	Pcog			SGWFN		Pcgw	
SWOF	Pcwo		Pcwo	SOF24	No Pc		
				SOF35	No Pc		
				SOF32D	No Pc		
				SWFN			Pcwo
OPM Flow LET[6]				**Format Type Three7**			
Keyword	**Oil**	**Gas**	**Water**	**Keyword**	**Oil**	**Gas**	**Water**
SGOFLET	Pcog			GSF		Pcgw	
SWGFLET		Pcgw		WSF			No Pc

OPM Flow LET[6]				Format Type Three7			
Keyword	**Oil**	**Gas**	**Water**	**Keyword**	**Oil**	**Gas**	**Water**
SWOFLET	Pcwo		Pcwo				

Notes:

1) Cells colored in gray with no mnemonic indicate that there is no keyword for this combination.
2) Cells colored orange show keywords that have not been implemented in OPM Flow.
3) In gas-water systems, the gas-water Pcgw data should be entered on the SWFN keyword and the Pcog on the SGFN keyword should be set to zero.
4) The SOF2 keyword defines the relative permeability in oil-gas and oil-water runs only, and the miscible hydrocarbon in SOLVENT runs. This keyword should not be used to define the oil relative permeability when oil, gas and water are present.
5) Defines oil in relative with respect to water and oil relative permeability with respect gas.
6) OPM Flow specific keywords implement the LET family of saturation function, see Lomeland et al.[168].
7) Format Type Three keywords are available in the commercial compositional simulator only, and will cause an error if used in the commercial black-oil simulator.

Table 8.9: Saturation Table Formats and Phases

Note that only format type can be used in a run, that is one must either use format type one relative permeability keywords to define the required saturation functions, or format two. One cannot combine the keywords from the different format types in the same input deck.

Figure 8.12 shows an unscaled oil-water relative permeability curve from the Volve model, and Figure 8.13 displays the corresponding scaled relative permeability curves after applying the end-point scaling parameters. Similarly for the gas-oil curves, Figure 8.14 illustrates the unscaled curves and Figure 8.15 the scaled curves. The plots indicate that the end-point scaling has been applied correctly.

[168] *Lomeland F., Ebeltoft E. and Thomas W.H., 2005. A New Versatile Relative Permeability Correlation. Paper SCA2005-32 presented at the International Symposium of the Society of Core Analysts held in Toronto, Canada, 21-25 August, 2005.*

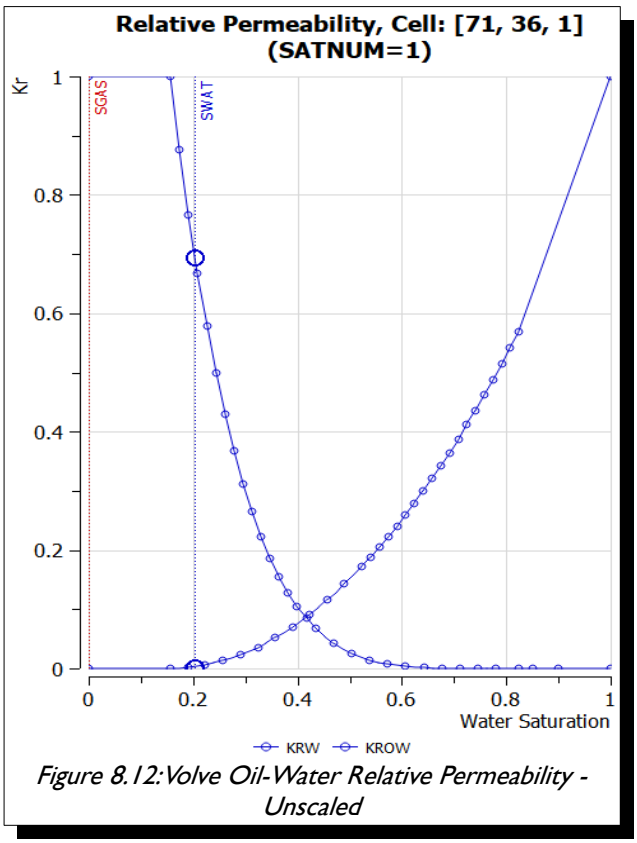


Figure 8.12: Volve Oil-Water Relative Permeability - Unscaled

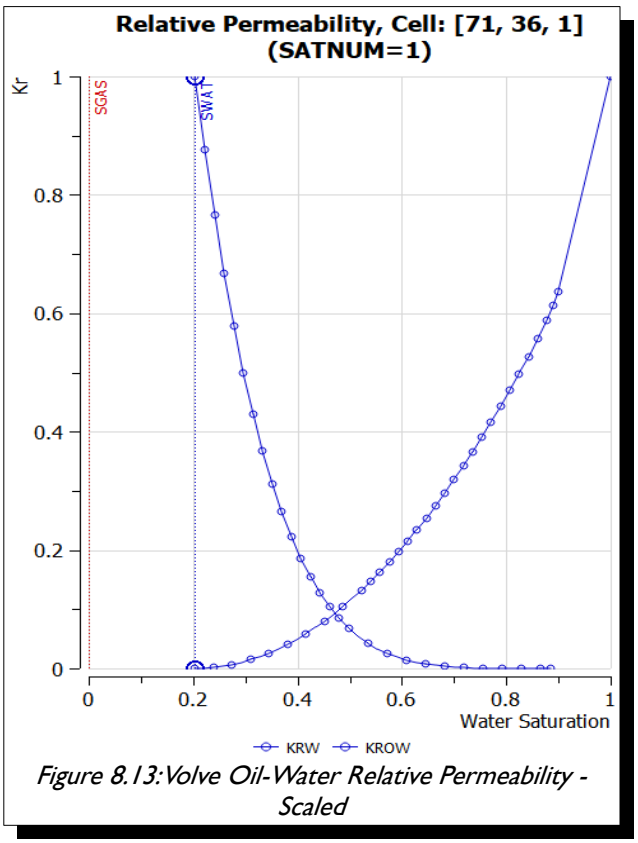


Figure 8.13: Volve Oil-Water Relative Permeability - Scaled

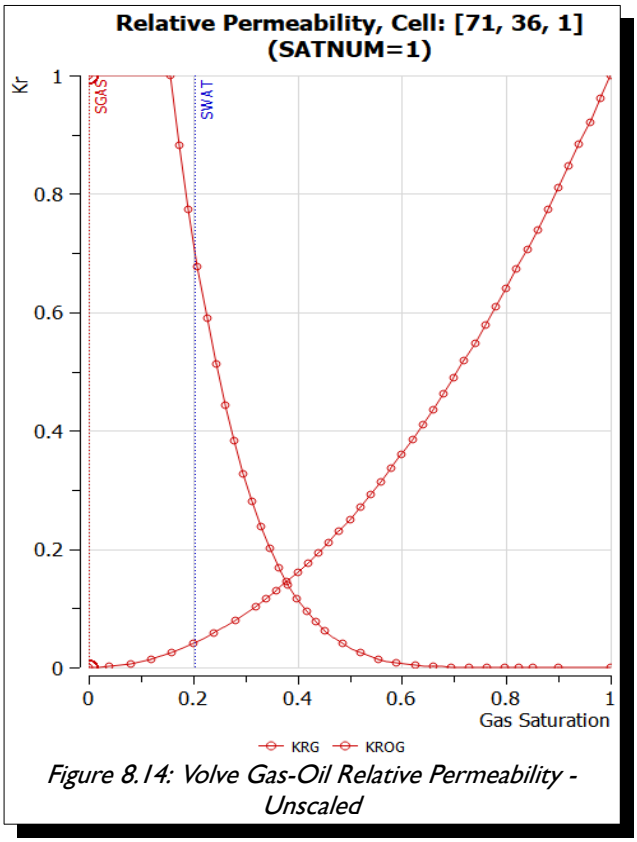


Figure 8.14: Volve Gas-Oil Relative Permeability - Unscaled

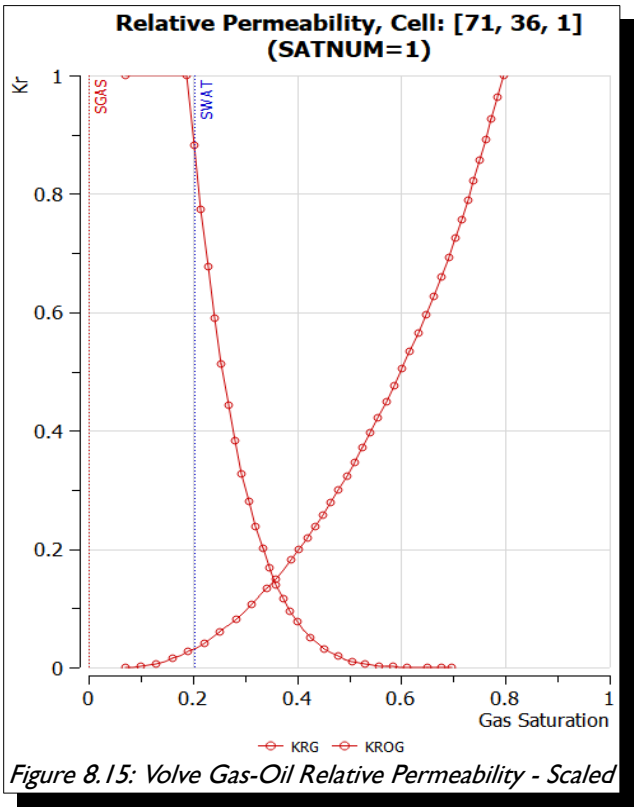


Figure 8.15: Volve Gas-Oil Relative Permeability - Scaled

8.2.5 SATURATION TABLE GENERATION - COREY CURVES

As mentioned in the *2.4.1 Data Quality and Assurance* section, Corey relative permeability curves are often used to "smooth" the intended relative permeability curves to be used in the simulator in order to improve convergence.

Corey[169] combined the work of Purcell[170] and Burdine[171] that was widely accepted for its simplicity. His original equations were developed for the drainage cycle in water-wet sandstones, but have also been used in carbonate formations. Corey's original water-oil equations were as follows:

$$k_{ro}\left(S_w\right) = \left[\frac{1 - S_w}{1 - S_{wcr}}\right]^{n_o} \tag{8.17}$$

$$k_{rw}\left(S_w\right) = \left[\frac{S_w - S_{wcr}}{1 - S_{wcr}}\right]^{n_w} \tag{8.18}$$

Where:

$k_{ro}(S_w)$	= relative permeability to oil,
$k_{rw}(S_w)$	= relative permeability to water,
n_o	= Corey oil exponent, set to four in the original paper,
n_w	= Corey water exponent, set to four in the original paper,
S_w	= water saturation, and
S_{wc}	= critical water saturation.

The denominator in equations (8.17) and (8.18) scales the water saturation to the mobile water phase. There are several forms of these equations, with the most common normalizing the saturation over the mobile hydrocarbon phase, as depicted in equations (8.19) and (8.20).

$$k_{ro}\left(S_w\right) = k_{row}\left[\frac{1 - S_w - S_{orw}}{1 - S_{orw} - S_{wcr}}\right]^{n_o} \tag{8.19}$$

$$k_{rw}\left(S_w\right) = k_{rww}\left[\frac{S_w - S_{wc}}{1 - S_{orw} - S_{wcr}}\right]^{n_w} \tag{8.20}$$

Where,

k_{row}	= maximum oil relative permeability at Swc,
k_{rw}	= maximum water relative permeability st Sorw,
S_{wc}	= critical water saturation, and
S_{orw}	= residual oil saturation under a water flood (SOWCR).

Similar equations exist for gas-oil and water-oil systems. As mentioned above, the denominator in the equations, normalizes the saturation to the mobile phase, as a consequence, it is still necessary to extend the resulting Corey water curve to 100% water saturation in order to correctly model the water leg. In terms of the values for the various Corey exponents, Table 8.10 offers some guidelines based on the rock's wettability.

[169] *Corey, A. T.: "The Interrelation Between gas and Oil Relative Permeabilities", Production Mon., 19. 38. (1954).*

[170] *Purcell, W. R., "Capillary Pressures- Their Measurement Using Mercury and the Calculation of Permeability Therefrom", Transactions AIME, 186, 39 (1949).*

[171] *Burdine, N. T., "Relative Permeability Calculations from Pore Size Distribution Data", , Transactions AIME, 198, 71 (1953).*

Corey Guide (Water-Oil)			Corey Guide (Gas-Oil)		
Wettability	**Kro**	**Krw**	**Wettability**	**Kro**	**Krg**
Water wet	2 - 4	5 - 8	Water wet		
Intermediate	3 - 6	3 - 5	Intermediate		
Oil wet	6 - 8	2 - 3	Oil wet	4 - 8	2 - 4

Table 8.10: Corey Exponent Values

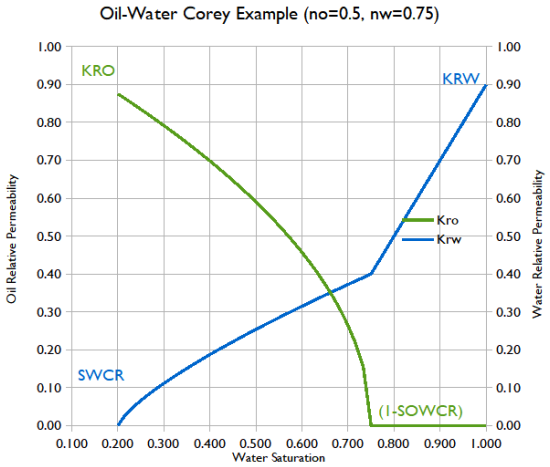


Figure 8.16: Oil-Water Corey Example (no=3.0, nw=4.0)

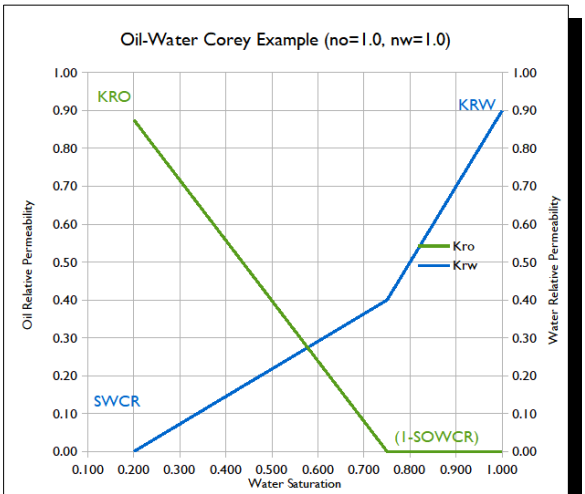


Figure 8.17: Oil-Water Corey Example (no=1.0, nw=1.0)

As can be seen in Figure 8.16 to Figure 8.18, the Corey exponents less than one result in a concave curve, and values great than one result in a convex curve, where as a value of one gives a straight line.

Unfortunately, neither OPM Flow nor the commercial simulator, unlike some other simulators, support the direct entry of Corey type curves, and thus Corey curves have to generated outside of the simulator. Fortunately, OPM Flow does support the more advance and flexible LET family of models instead, which are described in the next section.

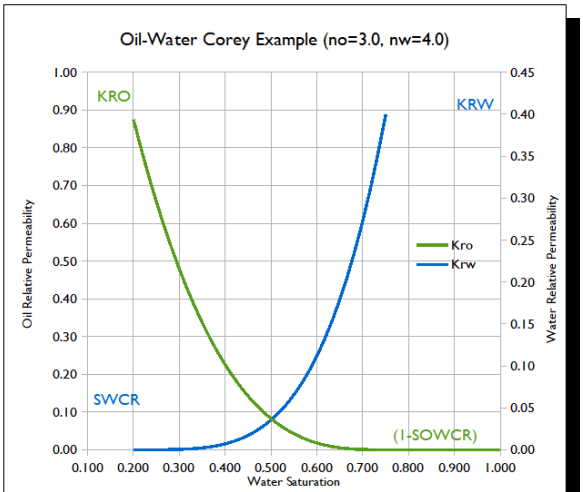


Figure 8.18:: Oil-Water Corey Example (no=3.0, nw=4.0)

8.2.6 SATURATION TABLE GENERATION - LET FUNCTIONS

OPM Flow has implemented the LET family of saturation functions, models, based on the work of Lomeland et al.[172,173,174 and 175,,] via the *SGOFLET– Gas-Oil LET Relative Permeability Functions*, *SGWFLET – Gas-Water LET Relative Permeability Functions*, and *SWOFLET – Water-Oil LET Relative Permeability Functions* keywords in this section. The keywords are used as replacements for the SGOF and SWOF keywords for three-phase oil-gas-water systems, and the SGWFN keyword for gas-water systems. Note there a two versions of the LET functions, LET[172] for two-phase flowing conditions and LETx[176] for three-phase flowing conditions. All three keywords implement the LET version for two phase systems.

The functions are dependent on the drainage and imbibition cycle of the wetting phase as well as drainage and inhibition cycle number, since a reservoir may undergo several flooding events. To account for this the system defines the flooding event using the three saturations: Sw, So, and Sg together with the state of the three saturations during the flooding event. The saturation state can be Increasing, Decreasing, or Constant, for a given flooding event cycle number (n). Thus, Sw(D), So(I), Sg(C)1or DIC1, means the water phase is decreasing, the oil phase is increasing and the gas phase is constant for the primary or first cycle (n equals one). This is case for when oil is migrating into the reservoir rock and displacing the initial water contained with the reservoir. The various flooding processes are outlined in Table 8.11

Modeling Of Flooding Processes				
Oil Field		Gas Field		Comment
Oil-Water Oil Leg	Oil-Gas Oil-Leg	Gas-Water Gas Cap	Oil-Gas Gas Cap	Nomenclature LET(Sw, So, Sg, n)
DIC1	CDI2	DCI1	CDI2	Primary starting point.
IDC2	CID3	ICD2	CID3	Second event
DIC3	CID4	DCI3	CID4	Third event

Table 8.11: LET Modeling of Flooding Processes

Thus, DIC1 is the well-known primary (first inflow number) drainage for an oil/water system, and IDC2 is the well-known (second general inflow number) imbibition for an oil/water system.

The following section describe the various flooding scenarios and the standard LET equations used in the literature.

[172] *Lomeland F., Ebeltoft E. and Thomas W.H., 2005. A New Versatile Relative Permeability Correlation. Paper SCA2005-32 presented at the International Symposium of the Society of Core Analysts held in Toronto, Canada, 21-25 August, 2005.*

[173] *Lomeland F. and Ebeltoft E., 2008. A New Versatile Capillary Pressure Correlation. Paper SCA2008-08 presented at the International Symposium of the Society of Core Analysts held in Abu Dhabi, UAE, 29 Oct. – 2 Nov., 2008.*

[174] *Lomeland F., Hasanov B., Ebeltoft E. and Berge M., 2012. A Versatile Representation of Up-scaled Relative Permeability for Field Applications. Paper SPE 154487-MS presented at the EAGE Annual Conference & Exhibition incorporating SPE Europec held in Copenhagen, Denmark, 4-7 June 2012.*

[175] *Lomeland F., 2018.. Overview Of The Let Family Of Versatile Correlations For Flow Functions. Paper SCA2018-056 presented at the International Symposium of the Society of Core Analysts held in Trondheim, Norway, 27-30 August 2018.*

[176] *Lomeland F. and Ebeltoft E., 2013. Versatile Three-phase Correlations for Relative Permeability and Capillary Pressure. Paper SCA2013-034 presented at the International Symposium of the Society of Core Analysts held in Napa Valley, California, USA, 16-19 September, 2013.*

Oil Displacing Water: Primary Drainage Cycle (DIC1)

For the Primary Drainage, assuming water is the wetting phase, then, Sw(D), So(I), Sg(C)1or DIC1, means the water phase is decreasing, the oil phase is increasing and the gas phase is constant for the primary or first cycle (n equals one). This is case for when oil is migrating into the reservoir rock and displacing the initial water contained within the reservoir. Under these circumstance the LET normalize water saturation and relative permeability functions are defined as:

$$K_{row} = \frac{K_{rot}\left(1 - S_{wp}\right)^{L_o}}{\left(1 - S_{wp}\right)^{L_o} + E_o S_{wp}^{T_o}} \tag{8.21}$$

$$K_{rw} = \frac{K_{rwt} S_{wp}^{L_w}}{S_{wp}^{L_w} + E_w\left(1 - S_{wp}\right)^{T_w}} \tag{8.22}$$

$$S_{wp} = \frac{S_w - S_{wirr}}{1 - S_{wirr}} \tag{8.23}$$

Where:

E_α	=	empirical E parameter for the α phase,
K_{row}	=	oil relative permeability with respect to water saturation,
K_{rot}	=	base or top oil relative permeability, used to scale the maximum oil permeability,
K_{rw}	=	water relative permeability,
K_{rwt}	=	base or top water relative permeability, used to scale the maximum water permeability,
L_α	=	empirical L parameter for the α phase,
T_α	=	empirical T parameter for the α phase,
S_w	=	water saturation,
S_{wirr}	=	irreducible water saturation, and
S_{wp}	=	normalize water saturation.

The LET empirical parameters: E, L and T, influence the shape of the relative permeability curves, as depicted in Figure 8.19

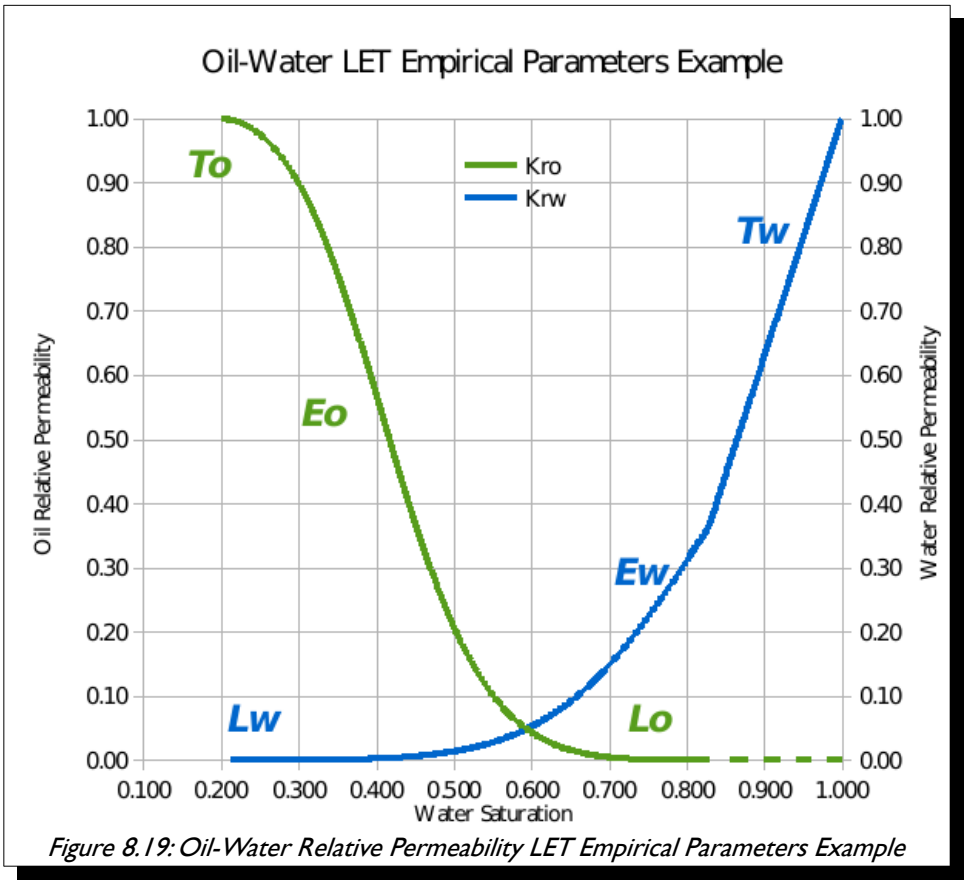


Figure 8.19: Oil-Water Relative Permeability LET Empirical Parameters Example

For the for the primary drainage or first cycle, DIC1, with the water phase decreasing, the oil phase increasing and the gas phase constant, the LET capillary pressure functions are:

$$Y = \frac{P_{cow} - P_{ct}}{P_{cir} - P_{ct}} \quad and \quad Y = F\left(S_{wp}\right) \quad where \quad S_{wirr} \leqslant S_w \leqslant 1.0 \tag{8.24}$$

$$P_{cow}\left(S_w = S_{wirr}\right) = P_{cir} \quad and \quad P_{cow}\left(S_w = 1.0\right) = P_{cit} \tag{8.25}$$

$$F = \frac{\left(1 - S_{wp}\right)^L}{\left(1 - S_{wp}\right)^L + E\,S_{wp}^T} \quad and \quad S_{wp} = \frac{S_w - S_{wirr}}{1 - S_{wirr}} \tag{8.26}$$

Where:

E	=	empirical E parameter,
L	=	empirical L parameter,
P_{cir}	=	capillary pressure at irreducible water saturation,
P_{ct}	=	capillary threshold / entry pressure,
T	=	empirical T parameter,
S_w	=	water saturation,
S_{wirr}	=	irreducible water saturation, and
S_{wp}	=	normalize water saturation.

The capillary pressure LET functions with either L or T equal to one are called a semi-simple LET functions, and LET functions with both L and T equal to one are called a simple LET function. The simple LET function is equal to the Pc correlation of Honapour et al.[177] except for the arrangement of the empirical coefficients.

The L parameter describes the lower part of the curve, whereas the T parameter describes the top part of the curve in a similar to how the L parameter describes the lower part of the curve. Finally, the E parameter describes the position of the slope of the curve, that is the elevation. An E value of one is neutral, and the position of the slope is governed by the L and T parameters in this case. Whereas, increasing the value of the E moves the slope towards the high end of the curve, and decreasing the value of the E moves the slope towards the lower end of the curve. Lombard et al.173 note that experience using the LET correlation indicate that L, E, and T values should greater than or equal to 0.01, with no upper limit.

Figure 8.20 shows a typical LET derived drainage oil-water capillary pressure curve, and Figure 8.21 shows a typical imbibition curves, the later is discussed in the following section.

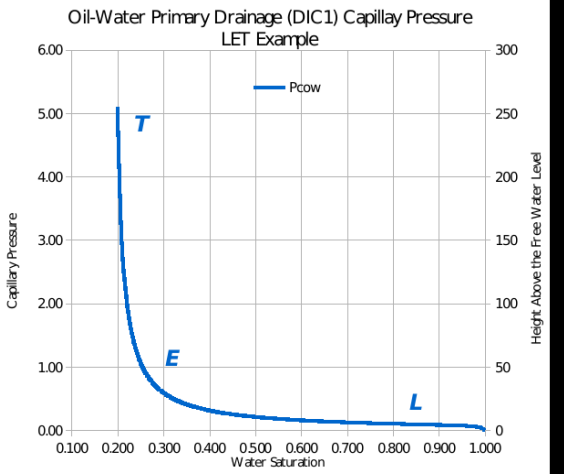


Figure 8.20: Oil-Water Primary Drainage (DIC1) Capillary Pressure LET Example

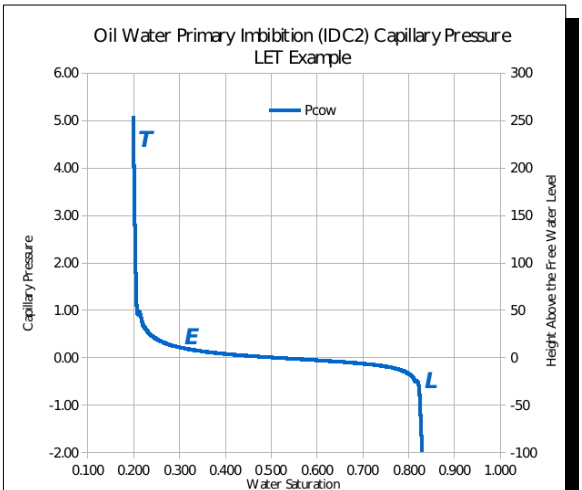


Figure 8.21: Oil Water Primary Imbibition (IDC2) Capillary Pressure LET Example

OPM has implemented a more restrictive form of the general LET functions, more akin to the original paper[172], as depicted in Figure 8.20. Table 8.12 relates the general LET equations presented previously with the parameters on the SWOFLET keyword.

LET Function (DIC1) versus SWOFLET Keyword					
SWOFLET	**LET Function**	**SWOFLET**	**LET Function**	**SWOFLET**	**LET Function**
SWL	S_{wirr}	SORW		PCIR	P_{cir}
SWCR	S_{wirr}	SOWCR		PCIT	P_{ct}
LWAT	L_w	LOIL	L_o	LPC	L
EWAT	E_w	EOIL	E_o	EPC	E
TWAT	T_w	TOIL	T_o	TPC	T

[177] *Honarpour M.M., Djabbarah N.F. and Kralik J.G., 2004. Expert-Based Methodology for Primary Drainage Capillary Pressure Measurements and Modeling. Paper SPE-88709 presented at the 11th Abu Dhabi International Petroleum Exhibition and Conference held in Abu Dhabi, UAE, 10-13 Oct., 2004.*

LET Function (DIC1) versus SWOFLET Keyword					
SWOFLET	**LET Function**	**SWOFLET**	**LET Function**	**SWOFLET**	**LET Function**
KRTWAT	K_{rwt}	KRTOIL	K_{rot}		

Notes:

1) The current implementation of the LET functions is restricted to a single branch capillary pressure curves as shown in Figure 8.20, and not the multi-branch curve shown in Figure 8.21.

Table 8.12: LET Functions (DIC1) versus SWOFLET Keyword

Water Displacing Oil: Primary Imbibition Cycle (IDC2)

Similarly for the oil producing phase, were the water phase is increasing either by replacing the displaced oil or via water injection, that is Sw(I), So(D), Sg(C), for the second flooding event, IDC2. Under this scenario the LET equations are:

$$K_{row} = \frac{K_{rot}\left(1 - S_{wn}\right)^{L_o}}{\left(1 - S_{wn}\right)^{L_o} + E_o S_{wn}^{T_o}} \tag{8.27}$$

$$K_{rw} = \frac{K_{rwr} S_{wn}^{L_w}}{S_{wn}^{L_w} + E_w\left(1 - S_{wn}\right)^{T_w}} \tag{8.28}$$

$$S_{wn} = \frac{S_w - S_{wirr}}{1 - S_{orw} - S_{wirr}} \tag{8.29}$$

Where:

E_α = empirical E parameter for the α phase,
K_{row} = oil relative permeability with respect to water saturation,
K_{rot} = base or top oil relative permeability, used to scale the maximum oil permeability,
K_{rw} = water relative permeability,
K_{rwr} = base or top water relative permeability, used to scale the maximum water permeability,
L_α = empirical L parameter for the α phase,
T_α = empirical T parameter for the α phase,
S_{orw} = residual oil saturation under water displacement,
S_w = water saturation,
S_{wirr} = irreducible water saturation, and
S_{wn} = normalize water saturation.

For the for the primary imbibition or second cycle, IDC2, with the water phase increasing, the oil phase decreasing and the gas phase constant, the LET capillary pressure functions are:

$$P_{cow} = \left(P_{cir} - P_{ct}\right)G_n + \left(P_{cor} - P_{ct}\right)F_n + P_{ct} \tag{8.30}$$

$$where\ S_{wir} \le S_w \le \left(1 - S_{orw}\right)\ and\ P_{cow}\left(S_w = S_{wzo}\right) = 0 \Rightarrow E_{sn}$$

$$G_n = \frac{\left(1 - S_{wn}\right)^{L_{sn}}}{\left(1 - S_{wn}\right)^{L_{sn}} + E_{sn} S_{wn}^{T_{sn}}} \qquad F_n = \frac{S_{wn}^{L_{fn}}}{S_{wn}^{L_{fn}} + E_{fn}\left(1 - S_{wn}\right)^{T_{fn}}} \tag{8.31}$$

$$S_{wn} = \frac{S_w - S_{wirr}}{1 - S_{orw} - S_{wirr}} \tag{8.32}$$

Where:

E_{fn} = forced imbibition empirical E parameter,
E_{sn} = spontaneous imbibition empirical E parameter,
L_{fn} = forced imbibition empirical L parameter,
L_{sn} = spontaneous imbibition empirical L parameter,
P_{cir} = capillary pressure at irreducible water saturation,
P_{ct} = capillary threshold / entry pressure,
T_{fn} = forced imbibition empirical T parameter,
T_{sn} = spontaneous imbibition empirical T parameter,
S_{orw} = residual oil saturation under water displacement,
S_w = water saturation,
S_{wirr} = irreducible water saturation, and
S_{wn} = normalize water saturation.

Spontaneous imbibition, refers to the wetting phase displacing the non-wetting phase, resulting in capillary pressure values greater than zero. For example, in a water wet oil reservoir under water drive, the water will easily imbibe into the pore space and displace the non-wetting oil phase. Forced imbibition is when the non-wetting phase displaces the wetting phase, and results in a negative capillary pressure values necessary to overcome the capillary force holding the wetting phase in-place. Thus, for a water wet oil reservoir, with the oil zone expanding into the water zone, an additional force is required to displace the water.

Table 8.13 relates the general LET equations presented previously with the parameters on the SWOFLET keyword.

LET Function (IDC2) versus SWOFLET Keyword					
SWOFLET	**LET Function**	**SWOFLET**	**LET Function**	**SWOFLET**	**LET Function**
SWL	S_{wirr}	SORW	S_{orw}	PCIR	P_{cir}
SWCR	S_{wirr}	SOWCR	S_{orw}	PCIT	P_{ct}
LWAT	L_w	LOIL	L_o	LPC	L_{sn}
EWAT	E_w	EOIL	E_o	EPC	E_{sn}
TWAT	T_w	TOIL	T_o	TPC	T_{sn}
KRTWAT	K_{rwr}	KRTOIL	K_{rot}		

Notes:

1) The current implementation of the LET functions is restricted to a single branch capillary pressure curves as shown in Figure 8.20, and not the multi-branch curve shown in Figure 8.21.

Table 8.13: LET Functions (IDC2) versus SWOFLET Keyword

Gas Displacing Oil: Primary Imbibition (CDI2)

For the oil producing phase, were the gas phase is increasing either by replacing the displaced oil or via gas injection, CDI2, the LET equations are:

$$K_{rog} = \frac{K_{rot}\left(1 - S_{gn}\right)^{L_o}}{\left(1 - S_{gn}\right)^{L_o} + E_o S_{gn}^{T_o}} \tag{8.33}$$

$$K_{rg} = \frac{K_{rgt} S_{gn}^{L_g}}{S_{gn}^{L_g} + E_g\left(1 - S_{gn}\right)^{T_g}} \tag{8.34}$$

$$S_{gn} = \frac{S_g}{1 - S_{org} - S_{wirr}} \tag{8.35}$$

Where:

E_α	=	empirical E parameter for the α phase,
K_{rog}	=	oil relative permeability with respect to gas saturation,
K_{rot}	=	base or top oil relative permeability, used to scale the maximum oil permeability,
K_{rg}	=	gas relative permeability,
K_{rgt}	=	base or top gas relative permeability, used to scale the maximum gas permeability,
L_α	=	empirical L parameter for the α phase,
T_α	=	empirical T parameter for the α phase,
S_g	=	gas saturation,
S_{gn}	=	normalize gas saturation,
S_{org}	=	residual oil saturation under gas displacement, and
S_{wirr}	=	irreducible water saturation.

Similarly for the for the capillary pressure CDI2, with the water phase constant, the oil phase decreasing and the gas phase increasing, the LET capillary pressure functions are:

$$P_{cog} = \left(P_{cir} - P_{ct}\right) G_n + \left(P_{cor} - P_{ct}\right) F_n + P_{ct} \tag{8.36}$$

$$where\ S_{wir} \le S_w \le \left(1 - S_{org} - S_{wirr}\right)\ and\ P_{cog}\left(S_w = S_{wzo}\right) = 0 \Rightarrow E_{sn}$$

$$G_n = \frac{\left(1 - S_{gn}\right)^{L_{sn}}}{\left(1 - S_{gn}\right)^{L_{sn}} + E_{sn} S_{gn}^{T_{sn}}} \qquad F_n = \frac{S_{gn}^{L_{fn}}}{S_{gn}^{L_{fn}} + E_{fn}\left(1 - S_{gn}\right)^{T_{fn}}} \tag{8.37}$$

$$S_{gn} = \frac{S_g}{1 - S_{org} - S_{wirr}} \tag{8.38}$$

Where:

E_{fn}	=	forced imbibition empirical E parameter,
E_{sn}	=	spontaneous imbibition empirical E parameter,

L_{fn} = forced imbibition empirical L parameter,
L_{sn} = spontaneous imbibition empirical L parameter,
P_{cir} = capillary pressure at irreducible water saturation,
P_{ct} = capillary threshold / entry pressure,
T_{fn} = forced imbibition empirical T parameter,
T_{sn} = spontaneous imbibition empirical T parameter,
S_g = gas saturation,
S_{gn} = normalize gas saturation,
S_{org} = residual oil saturation under gas displacement, and
S_{wirr} = irreducible water saturation.

Table 8.14 relates the general LET equations presented previously with the parameters on the SGOFLET keyword.

LET Function (CDI2) versus SGOFLET Keyword					
SGOFLET	**LET Function**	**SGOFLET**	**LET Function**	**SGOFLET**	**LET Function**
SGL		SORG	S_{org}	PCIR	P_{cir}
SGCR		SOGCR	S_{org}	PCIT	P_{ct}
LGAS	L_g	LOIL	L_o	LPC	L_{sn}
EGAS	E_g	EOIL	E_o	EPC	E_{sn}
TGAS	T_g	TOIL	T_o	TPC	T_{sn}
KRTGAS	K_{rgt}	KRTOIL	K_{rot}		

Notes:

1) The current implementation of the LET functions is restricted to a single branch capillary pressure curves as shown in Figure 8.20, and not the multi-branch curve shown in Figure 8.21.
2) The S_{wirr} parameter value is taken from the SWOFLET keyword.

Table 8.14: LET Functions (CDI2) versus SGOFLET Keyword

Gas Displacing Water: Primary Drainage Cycle (DCI1)

For the Primary Drainage, assuming water is the wetting phase, then, Sw(D), So(C), Sg(I)1or DCI1, means the water phase is decreasing, the oil phase is constant and the gas phase is increasing for the primary or first cycle (n equals one). This is case for when gas is migrating into the reservoir rock and displacing the initial water contained within the reservoir. Under these circumstance the LET normalize water saturation and relative permeability functions are defined as:

$$K_{rgw} = \frac{K_{rgt}\left(1 - S_{wp}\right)^{L_g}}{\left(1 - S_{wp}\right)^{L_g} + E_g S_{wp}{}^{T_g}} \tag{8.39}$$

$$K_{rw} = \frac{K_{rwt} S_{wp}^{L_w}}{S_{wp}^{L_w} + E_w\left(1 - S_{wp}\right)^{T_w}} \tag{8.40}$$

$$S_{wp} = \frac{S_w - S_{wirr}}{1 - S_{wirr}} \tag{8.41}$$

Where:

E_α = empirical E parameter for the α phase,
K_{rgw} = gas relative permeability with respect to water saturation,
K_{rgt} = base or top gas relative permeability, used to scale the maximum gas permeability,
K_{rw} = water relative permeability,
K_{rwt} = base or top water relative permeability, used to scale the maximum water permeability,
L_α = empirical L parameter for the α phase,
T_α = empirical T parameter for the α phase,
S_w = water saturation,
S_{wirr} = irreducible water saturation, and
S_{wp} = normalize water saturation.

For the for the primary drainage or first cycle, DCI1, with the water phase decreasing, the oil phase constant and the gas phase increasing, the LET capillary pressure functions are:

$$Y = \frac{P_{cgw} - P_{ct}}{P_{cir} - P_{ct}} \quad and \quad Y = F\left(S_{wp}\right) \quad where \quad S_{wirr} \leqslant S_w \leqslant 1.0 \tag{8.42}$$

$$P_{cgw}\left(S_w = S_{wirr}\right) = P_{cir} \quad and \quad P_{cgw}\left(S_w = 1.0\right) = P_{cit} \tag{8.43}$$

$$F = \frac{\left(1 - S_{wp}\right)^L}{\left(1 - S_{wp}\right)^L + E\,S_{wp}^T} \quad and \quad S_{wp} = \frac{S_w - S_{wirr}}{1 - S_{wirr}} \tag{8.44}$$

Where:

E = empirical E parameter,
L = empirical L parameter,
P_{cir} = capillary pressure at irreducible water saturation,
P_{ct} = capillary threshold / entry pressure,
T = empirical T parameter,
S_w = water saturation,
S_{wirr} = irreducible water saturation, and
S_{wp} = normalize water saturation.

Table 8.15 relates the general LET equations presented previously with the parameters on the SGOFLET keyword.

LET Function (DCI1) versus SGWFLET Keyword					
SGWFLET	**LET Function**	**SGWFLET**	**LET Function**	**SGWFLET**	**LET Function**
SGL		SWL	S_{wirr}	PCIR	P_{cir}

LET Function (DCI1) versus SGWFLET Keyword					
SGWFLET	**LET Function**	**SGWFLET**	**LET Function**	**SGWFLET**	**LET Function**
SGCR		SWCR	S_{wirr}	PCIT	P_{ct}
LGAS	L_g	LWAT	L_w	LPC	L
EGAS	E_g	EWAT	E_w	EPC	E
TGAS	T_g	TWAT	T_w	TPC	T
KRTGAS	K_{rgt}	KRTWAT	K_{rwr}		

Notes:

1) The current implementation of the LET functions is restricted to a single branch capillary pressure curves as shown in Figure 8.20, and not the multi-branch curve shown in Figure 8.21.

Table 8.15: LET Functions (DCI1) versus SGOFLET Keyword

Note

The SGWFLET keyword has not been implemented, as of this current release.

Water Displacing Gas: Primary Imbibition Cycle (ICD2)

Similarly for the gas producing phase, were the water phase is increasing either by replacing the displaced gas or via water influx from the aquifer, that is Sw(I), So(C), Sg(D), for the second flooding event, IDC2. Under this scenario the LET equations are:

$$K_{rgw} = \frac{K_{rgt}\left(1 - S_{wn}\right)^{L_o}}{\left(1 - S_{wn}\right)^{L_o} + E_o S_{wn}^{T_o}} \tag{8.45}$$

$$K_{rw} = \frac{K_{rwr} S_{wn}^{L_w}}{S_{wn}^{L_w} + E_w\left(1 - S_{wn}\right)^{T_w}} \tag{8.46}$$

$$S_{wn} = \frac{S_w - S_{wirr}}{1 - S_{grw} - S_{wirr}} \tag{8.47}$$

Where:

E_α = empirical E parameter for the α phase,
K_{rgw} = gas relative permeability with respect to water saturation,
K_{rot} = base or top gas relative permeability, used to scale the maximum gas permeability,
K_{rw} = water relative permeability,
K_{rwr} = base or top water relative permeability, used to scale the maximum water permeability,
L_α = empirical L parameter for the α phase,
T_α = empirical T parameter for the α phase,

S_{ogw} = residual gas saturation under water displacement,
S_w = water saturation,
S_{wirr} = irreducible water saturation, and
S_{wn} = normalize water saturation.

For the for the primary imbibition or second cycle, IDC2, with the water phase increasing, the oil phase decreasing and the gas phase constant, the LET capillary pressure functions are:

$$P_{cgw} = (P_{cir} - P_{ct}) G_n + (P_{cgr} - P_{ct}) F_n + P_{ct} \quad (8.48)$$

$$where\ S_{wir} \le S_w \le (1 - S_{gorw})\ and\ P_{cgw}(S_w = S_{wzo}) = 0 \Rightarrow E_{sn}$$

$$G_n = \frac{(1 - S_{wn})^{L_{sn}}}{(1 - S_{wn})^{L_{sn}} + E_{sn} S_{wn}^{T_{sn}}} \qquad F_n = \frac{S_{wn}^{L_{fn}}}{S_{wn}^{L_{fn}} + E_{fn}(1 - S_{wn})^{T_{fn}}} \quad (8.49)$$

$$S_{wn} = \frac{S_w - S_{wirr}}{1 - S_{grw} - S_{wirr}} \quad (8.50)$$

Where:

E_{fn} = forced imbibition empirical E parameter,
E_{sn} = spontaneous imbibition empirical E parameter,
L_{fn} = forced imbibition empirical L parameter,
L_{sn} = spontaneous imbibition empirical L parameter,
P_{cir} = capillary pressure at irreducible water saturation,
P_{ct} = capillary threshold / entry pressure,
T_{fn} = forced imbibition empirical T parameter,
T_{sn} = spontaneous imbibition empirical T parameter,
S_{ogw} = residual gas saturation under water displacement,
S_w = water saturation,
S_{wirr} = irreducible water saturation, and
S_{wn} = normalize water saturation.

Table 8.16 relates the general LET equations presented previously with the parameters on the SGOFLET keyword.

LET Function (DCI1) versus SGWFLET Keyword					
SGWFLET	**LET Function**	**SGWFLET**	**LET Function**	**SGWFLET**	**LET Function**
SGL		SWL	S_{wirr}	PCIR	P_{cir}
SGCR		SWCR	S_{wirr}	PCIT	P_{ct}
LGAS	L_g	LWAT	L_w	LPC	L_{sn}
EGAS	E_g	EWAT	E_w	EPC	E_{sn}
TGAS	T_g	TWAT	T_w	TPC	T_{sn}
KRTGAS	K_{rgt}	KRTWAT	K_{rwr}		

LET Function (DCI1) versus SGWFLET Keyword					
SGWFLET	**LET Function**	**SGWFLET**	**LET Function**	**SGWFLET**	**LET Function**

Notes:

1) The current implementation of the LET functions is restricted to a single branch capillary pressure curves as shown in Figure 8.20, and not the multi-branch curve shown in Figure 8.21.

Table 8.16: LET Functions (DCI1) versus SGWFLET Keyword

Note

The SGWFLET keyword has not been implemented, as of this current release.

8.2.7 SATURATION TABLE ALLOCATION

Saturation tables are allocated to individual cells using the SATNUM keyword in the REGIONS section for when the hysteresis model option has not been activated on the SATOPTS keyword in the RUNSPEC section. If the hysteresis model option has been activated then the SATNUM keyword allocates the "drainage" saturation tables instead.

If the Directional Dependent Saturation Function option has been activated by the DIRECT parameter on the SATOPTS keyword in the RUNSPEC section, then the KRNUM series of keywords in the REGIONS section may be used to allocate the saturation tables for each grid block face. Here the KRNUMX, KRNUMY and KRNUMZ keywords are used instead of KRNUM. There is also the facility to make the directional end-point scaling reversible or non-reversible and if the non-reversible option is selected the non-reversible versions of the aforementioned arrays should be used, that is KRNUMX, KRNUMX-, KRNUMY, KRNUMY-, KRNUMZ and KRNUMZ-, instead of the KRNUM keyword. Again, if the hysteresis model option has been activated then the KRNUM series of keywords allocate the "drainage" saturation tables instead.

Consequently, if the hysteresis model option has been activated, then the equivalent imbibition keyword, IMBNUM, is used to allocate the imbibition tables. In addition, imbibition saturation table assignment may be directional dependent, in which case the directional dependent versions of the aforementioned IMBNUM array should be used, that is IMBNUMX, IMBNUMY and IMBNUMZ instead of IMBNUM. There is also the facility to make the directional end-point scaling reversible or non-reversible and if the non-reversible option is selected the non-reversible versions of the aforementioned arrays should be used, that is IMBNUMX, IMBNUMX-, IMBNUMY, IMBNUMY-, IMBNUMZ and IMBNUMZ-, instead of the IMBNUM keyword.

The above discussion is based on the assumption that the model is constructed using a Cartesian grid; however, if the model is based on a radial grid then the equivalent radial saturation table allocation keywords should be used. Thus, the equivalent directional KRNUM keywords would be KRNUMR, KRNUMR-, KRNUMT, KRNUMT-, KRNUMZ and KRNUMZ-. Note that the KRNUMZ and KRNUMZ- keywords are used for both Cartesian and radial grids.

For the IMBNUM imbibition keyword for a radial grid, one would use IMBNUMR, IMBNUMR-, IMBNUMT, IMBNUMT-, IMBNUMZ and IMBNUMZ-, and again the IMBNUMZ and IMBNUMZ- keywords are used for both Cartesian and radial grids.

8.3 KEYWORD DEFINITIONS

8.3.1 ADD – ADD A CONSTANT TO A SPECIFIED ARRAY

The ADD keyword adds a constant to a specified array or part of an array. The constant can be real or integer depending on the array type; however, the arrays that can be operated on is dependent on which section the ADD keyword is being applied.

See *ADD – Add a Constant to a Specified Array* in the GRID section for a full description.

8.3.2 ADDREG – ADD A CONSTANT TO AN ARRAY BASED ON A REGION NUMBER

The ADDREG keyword adds a constant to a specified array or part of an array based on cells with a specific region number. The region number array can be FLUXNUM, MULTNUM or OPERNUM and these arrays must be defined and be available before the ADDREG keyword is read by the simulator. The constant can be real or integer depending on the property array type; however, the property arrays that can be operated on is dependent on which section the ADDREG keyword is being used.

See *ADDREG – Add a Constant to an Array based on a Region Number* in the GRID section for a full description.

8.3.3 ADSALNOD – SALT CONCENTRATION BASED ON SATNUM ARRAY

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

ADSALNOD defines the salt concentration value based on a cells SATNUM number. The ADSALNOD property is used in the calculation of a polymer viscosity when the polymer and the salt options has been activated by the POLYMER and BRINE keywords in the RUNSPEC section. In the RUNSPEC section the number of SATNUM functions is declared by the NTSFUN variable on the TABDIMS keyword and allocated to individual cells by the SATNUM property array in the REGIONS section. NSSFUN on the TABDIMS keyword in the RUNSPEC section defines the maximum number of rows (or saturation values) in the relative permeability saturation tables and also sets the maximum number of entries for each ADSALNOD data set. The number of values for each data set must correspond to the number of polymer solution adsorption entries on the PLYADSS keyword. For example, if there are three sets of relative permeability tables and four values on the PLYADSS keyword, then three ADSALNOD data sets with four values of salt concentrations need to be entered.

The salt concentrations within each data set should be positive and monotonically increasing and each ADSALNOD data set is delimited by "/" including the last data set.

No.	Name	Description			Default
1	SALTCON	**Field**	**Metric**	**Laboratory**	
		A real positive columnar vector that sets the salt concentrations for the given relative permeability saturation tables.			
		lb/stb	kg/sm3	gm/scc	None
Notes: 1) Each data set must be terminated by a "/" including the last data set.					

Table 8.17:ADSALNOD Keyword Description

An alternative manner of entering the salt concentrations is by utilizing the PVTNUM region array by using the SALTNODE keyword in the PROPS section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

Example

Given three sets of relative permeability tables and four values on the PLYADSS keyword, then the data salt concentration should be entered as follows:

```
--
--    SETS SALT CONCENTRATION FOR POLYMER SOLUTION ADSORPTION
--    VIA SATNUM ARRAY ALLOCATION
--
--    SALT
--
ADSALNOD
      1.0
      5.0
      10.5
      25.0      / SATNUM TABLE NO. 01
      1.0
      3.0
      7.5
      15.0      / SATNUM TABLE NO. 02
      1.0
      7.5
      20.5
      35.0      / SATNUM TABLE NO. 03
```

See also the SALTNODE keyword.

8.3.4 ADSORP – DEFINE GENERALIZED LANGMUIR ADSORPTION FUNCTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The ADSORP keyword defines the parameters for the generalized Langmuir Adsorption[178] function for when the polymer, surfact, alkaline, foam and tracers phases have been activated in the RUNSPEC section by the POLYMER, SURFACT, ALKALINE, FOAM and TRACER keywords.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

[178] *Langmuir, Irving (June 1918). "The Adsorption of Gases on Plane Surface of Glass, Mica and Platinum". The Research Laboratory of the General Electric Company. 40 (9): 1361–1402. doi:10.1021/ja02242a004*

8.3.5 ALKADS – DEFINE ALKALINE ADSORPTION FUNCTIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

ALKADS defines the alkaline adsorption functions for when the alkaline model has been activated via the ALKALINE keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.6 ALKROCK – DEFINE ROCK ALKALINE PROPERTIES

Description

The ALKROCK keyword defines the rock alkaline properties for when the alkaline model has been activated via the ALKALINE keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.7 ALPOLADS – POLYMER ADSORPTION VERSUS ALKALINE CONCENTRATION MULTIPLIERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

ALPOLDS defines the polymer adsorption versus alkaline concentration multipliers for when the alkaline model has been activated via the ALKALINE keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.8 ALSURFAD – SURFACTANT ADSORPTION VERSUS ALKALINE CONCENTRATION MULTIPLIERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

ALSURAD defines the surfactant adsorption versus alkaline concentration multipliers for when the alkaline model has been activated via the ALKALINE keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.9 ALSURFST – WATER-OIL SURFACE TENSION VERSUS ALKALINE CONCENTRATION MULTIPLIERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The ALSURFST keyword defines the water-oil surface tension versus alkaline concentration multipliers for when the alkaline model has been activated via the ALKALINE keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.10 APIGROUP – DEFINE API TRACKING NUMBER OF GROUPED OIL PVT TABLES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

The APIGROUP keyword defines the maximum number of groups of oil PVT tables when the API tracking option has been activated via the API keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.11 AQUCT – DEFINE CARTER-TRACY ANALYTICAL AQUIFERS

The AQUCT keyword defines a Carter Tracy aquifer, the properties of the aquifer, including the Carter-Tracy aquifer influence function associated with the aquifer, defined by the AQUTAB keyword in the PROPS section.

See *AQUCT – Define Carter-Tracy Analytical Aquifers* in the GRID section for a full description

8.3.12 AQUTAB – DEFINE CARTER-TRACY AQUIFER INFLUENCE FUNCTIONS

Description

The AQUTAB keyword defines additional Carter-Tracy[179] aquifer functions to be used in the model. Carter-Tracy representation of the aquifer influx is via a qw term in the non-linear aquifer influence function Q(t). It allows the water influx from the aquifer to be represented in the simulator by assuming that there is a constant water influx rate over finite time periods. It is derived from the superposition methods of van Everdingen and Hurst[180], whose superposition methods are not suitable for implementation in reservoir simulation software, although they are very useful in interpreting aquifer response. The storage requirements and calculation complexity of handling the resulting superposition formulae can be largely eliminated by use of the Carter-Tracy approximate water influx method.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	TD	Dimensionless Time			
		dimensionless	dimensionless	dimensionless	None
2	PD	Dimensionless Pressure			
		dimensionless	dimensionless	dimensionless	None

Notes:

1) The keyword is followed by NIFTBL tables as declared on the AQUDIMS keyword in the RUNSPEC section.
2) Each table must contain at least two complete rows with a maximum of NRIFTB rows as declared on the AQUDIMS keyword in the RUNSPEC section. Note that NRIFTB must not be less than 36 in order to accommodate the default infinite acting Carter-Tracy aquifer influence function.
3) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.18:AQUTAB Keyword Description

Note

OPM Flow includes the infinite acting Carter-Tracy aquifer influence table as a default for table number one; thus data entered on this keyword starts from table number two.

Table 8.19 to Table 8.22 outline the standard finite tables derived by van Everdingen and Hurst that are used for the Carter-Tracy analytical aquifers and are taken from Bradely[181] table 38-6 on page 38-12. In the tables rD is defined as the ratio of the aquifer external radius divided by hydrocarbon radius, that is: $r_D = \frac{r_e}{r_o}$.

[179] *Carter, R. D., and Tracy, G. W. "An Improved Method for Calculating Water Influx." Transactions of AIME, Vol. 219 (1060), pp 415-417.*

[180] *Van Everdingen, A. F., and Hurst, W. "The Application of the Laplace Transform to Flow Problems in Reservoirs." Transactions of AIME, Vol. 186 (1949), pp. 305-324.*

[181] *Bradley Howard B., et al,, Petroleum Engineering Handbook, Society of Petroleum Engineers (June 1989), ISBN:9781555630102.*

Carter-Tracy Aquifer Influence Functions								
No.	R_D = 1.5 Dimensionless		R_D = 2.0 Dimensionless		R_D = 2.5 Dimensionless		R_D = 3.0 Dimensionless	
	t_D	p_D	t_D	p_D	t_D	p_D	t_D	p_D
1	0.0600	0.2510	0.2200	0.4430	0.4000	0.5650	0.5200	0.6270
2	0.0800	0.2880	0.2400	0.4590	0.4200	0.5760	0.5400	0.6360
3	0.1000	0.3220	0.2600	0.4760	0.4400	0.5870	0.5600	0.6450
4	0.1200	0.3550	0.2800	0.4920	0.4600	0.5980	0.6000	0.6620
5	0.1400	0.3870	0.3000	0.5070	0.4800	0.6080	0.6500	0.6830
6	0.1600	0.4200	0.3200	0.5220	0.5000	0.6180	0.7000	0.7030
7	0.1800	0.4520	0.3400	0.5360	0.5200	0.6280	0.7500	0.7210
8	0.2000	0.4840	0.3600	0.5510	0.5400	0.6380	0.8000	0.7400
9	0.2200	0.5160	0.3800	0.5650	0.5600	0.6470	0.8500	0.7580
10	0.2400	0.5480	0.4000	0.5790	0.5800	0.6570	0.9000	0.7760
11	0.2600	0.5800	0.4200	0.5930	0.6000	0.6660	0.9500	0.7910
12	0.2800	0.6120	0.4400	0.6070	0.6500	0.6880	1.0000	0.8060
13	0.3000	0.6440	0.4600	0.6210	0.7000	0.7100	1.2000	0.8650
14	0.3500	0.7240	0.4800	0.6340	0.7500	0.7310	1.4000	0.9200
15	0.4000	0.8040	0.5000	0.6480	0.8000	0.7520	1.6000	0.9730
16	0.4500	0.8840	0.6000	0.7150	0.8500	0.7720	2.0000	1.0760
17	0.5000	0.9640	0.7000	0.7820	0.9000	0.7920	3.0000	1.3280
18	0.5500	1.0440	0.8000	0.8490	0.9500	0.8120	4.0000	1.5780
19	0.6000	1.1240	0.9000	0.9150	1.0000	0.8320	5.0000	1.8280
20			1.0000	0.9820	2.0000	1.2150		
21			2.0000	1.6490	3.0000	1.5960		
22			3.0000	2.3160	4.0000	1.9770		
23			5.0000	3.6490	5.0000	2.3580		

Table 8.19: Carter-Tracy Aquifer Influence Functions (RD =1.5, 2.0, 2.5 and 3.0)

Carter-Tracy Aquifer Influence Functions								
No.	R_D = 3.5 Dimensionless		R_D = 4.0 Dimensionless		R_D = 4.5 Dimensionless		R_D = 5.0 Dimensionless	
	t_D	p_D	t_D	p_D	t_D	p_D	t_D	p_D
1	1.0000	0.8020	1.5000	0.9270	2.0000	1.0230	3.0000	1.1670
2	1.1000	0.8300	1.6000	0.9480	2.1000	1.0400	3.1000	1.1800
3	1.2000	0.8570	1.7000	0.9680	2.2000	1.0560	3.2000	1.1920
4	1.3000	0.8820	1.8000	0.9880	2.3000	1.0720	3.3000	1.2040
5	1.4000	0.9060	1.9000	1.0070	2.4000	1.0870	3.4000	1.2150
6	1.5000	0.9290	2.0000	1.0250	2.5000	1.1020	3.5000	1.2270
7	1.6000	0.9510	2.2000	1.0590	2.6000	1.1160	3.6000	1.2380
8	1.7000	0.9730	2.4000	1.0920	2.7000	1.1300	3.7000	1.2490
9	1.8000	0.9940	2.6000	1.1230	2.8000	1.1440	3.8000	1.2590
10	1.9000	1.0140	2.8000	1.1540	2.9000	1.1580	3.9000	1.2700
11	2.0000	1.0340	3.0000	1.1840	3.0000	1.1710	4.0000	1.2810
12	2.2500	1.0830	3.5000	1.2550	3.2000	1.1970	4.2000	1.3010
13	2.5000	1.1300	4.0000	1.3240	3.4000	1.2220	4.4000	1.3210
14	2.7500	1.1760	4.5000	1.3920	3.6000	1.2460	4.6000	1.3400
15	3.0000	1.2210	5.0000	1.4600	3.8000	1.2690	4.8000	1.3600
16	4.0000	1.4010	5.5000	1.5270	4.0000	1.2920	5.0000	1.3780
17	5.0000	1.5790	6.0000	1.5940	4.5000	1.3490	5.5000	1.4240
18	6.0000	1.7570	6.5000	1.6600	5.0000	1.4030	6.0000	1.4690
19			7.0000	1.7270	5.5000	1.4570	6.5000	1.5130
20			8.0000	1.8610	6.0000	1.5100	7.0000	1.5560
21			9.0000	1.9940	7.0000	1.6150	7.5000	1.5980
22			10.0000	2.1270	8.0000	1.7190	8.0000	1.6410
23					9.0000	1.8230	9.0000	1.7250
24					10.0000	1.9270	10.0000	1.8080
25					11.0000	2.0310	11.0000	1.8920
26					12.0000	2.1350	12.0000	1.9750
27					13.0000	2.2390	13.0000	2.0590
28					14.0000	2.3430	14.0000	2.1420
29					15.0000	2.4470	15.0000	2.2250

Table 8.20: Carter-Tracy Aquifer Influence Functions (RD =3.5, 4.0, 4.5 and 5.0)

Carter-Tracy Aquifer Influence Functions								
No.	R_D = 6.0 Dimensionless		R_D = 7.0 Dimensionless		R_D = 8.0 Dimensionless		R_D = 9.0 Dimensionless	
	t_D	p_D	t_D	p_D	t_D	p_D	t_D	p_D
1	4.0000	1.2750	6.0000	1.4360	8.0000	1.5560	10.0000	1.6510
2	4.5000	1.3220	6.5000	1.4700	8.5000	1.5820	10.5000	1.6730
3	5.0000	1.3640	7.0000	1.5010	9.0000	1.6070	11.0000	1.6930
4	5.5000	1.4040	7.5000	1.5310	9.5000	1.6310	11.5000	1.7130
5	6.0000	1.4410	8.0000	1.5590	10.0000	1.6530	12.0000	1.7320
6	6.5000	1.4770	8.5000	1.5860	10.5000	1.6750	12.5000	1.7500
7	7.0000	1.5110	9.0000	1.6130	11.0000	1.6970	13.0000	1.7680
8	7.5000	1.5440	9.5000	1.6380	11.5000	1.7170	13.5000	1.7860
9	8.0000	1.5760	10.0000	1.6630	12.0000	1.7370	14.0000	1.8030
10	8.5000	1.6070	11.0000	1.7110	12.5000	1.7570	14.5000	1.8190
11	9.0000	1.6380	12.0000	1.7570	13.0000	1.7760	15.0000	1.8350
12	9.5000	1.6680	13.0000	1.8010	13.5000	1.7950	15.5000	1.8510
13	10.0000	1.6980	14.0000	1.8450	14.0000	1.8130	16.0000	1.8670
14	11.0000	1.7570	15.0000	1.8880	14.5000	1.8310	17.0000	1.8970
15	12.0000	1.8150	16.0000	1.9310	15.0000	1.8490	18.0000	1.9260
16	13.0000	1.8730	17.0000	1.9740	17.0000	1.9190	19.0000	1.9550
17	14.0000	1.9310	18.0000	2.0160	19.0000	1.9860	20.0000	1.9830
18	15.0000	1.9880	19.0000	2.0580	21.0000	2.0510	22.0000	2.0370
19	16.0000	2.0450	20.0000	2.1000	23.0000	2.1160	24.0000	2.0900
20	17.0000	2.1030	22.0000	2.1840	25.0000	2.1800	26.0000	2.1420
21	18.0000	2.1600	24.0000	2.2670	30.0000	2.3400	28.0000	2.1930
22	19.0000	2.2170	26.0000	2.3510	35.0000	2.4990	30.0000	2.2440
23	20.0000	2.2740	28.0000	2.4340	40.0000	2.6580	34.0000	2.3450
24	25.0000	2.5600	30.0000	2.5170	45.0000	2.8170	38.0000	2.4460
25	30.0000	2.8460					40.0000	2.4960
26							45.0000	2.6210
27							50.0000	2.7460

Table 8.21: Carter-Tracy Aquifer Influence Functions (RD =6.0, 7.0, 8.0 and 9.0)

Carter-Tracy Aquifer Influence Functions				
No.	R_D = 10.0 Dimensionless		Finite Linear Dimensionless	
	t_D	p_D	t_D	p_D
1	12.0000	1.7320	0.0050	0.0798
2	12.5000	1.7500	0.0100	0.1130
3	13.0000	1.7680	0.0200	0.1596
4	13.5000	1.7840	0.0600	0.2764
5	14.0000	1.8010	0.0800	0.3192
6	14.5000	1.8170	0.1000	0.3568
7	15.0000	1.8320	0.1200	0.3909
8	15.5000	1.8470	0.1600	0.4515
9	16.0000	1.8620	0.2000	0.5052
10	17.0000	1.8900	0.2400	0.5544
11	18.0000	1.9170	0.3000	0.6228
12	19.0000	1.9430	0.4000	0.7294
13	20.0000	1.9680	0.6000	0.9328
14	22.0000	2.0170	0.8000	1.1333
15	24.0000	2.0630	1.0000	1.3333
16	26.0000	2.1080		
17	28.0000	2.1510		
18	30.0000	2.1940		
19	32.0000	2.2360		
20	34.0000	2.2780		
21	36.0000	2.3190		
22	38.0000	2.3600		
23	40.0000	2.4010		
24	50.0000	2.6040		
25	60.0000	2.8060		
26	70.0000	3.0080		

Table 8.22: Carter-Tracy Aquifer Influence Functions (RD =10 and Finite Linear)

For the finite linear Carter-Tracy influence function in Table 8.22 set the inner radius of the aquifer to the length of linear aquifer and the angle of influence to $= \frac{360 \times Width}{(2 \times \pi \times Length)}$ on the AQUCT keyword in the grid section.

For reference, Table 8.23 outlines the content terminal rate case for an infinite aquifer derived by van Everdingen and Hurst, that is the default table number one used for the Carter-Tracy analytical aquifers, as implemented in OPM Flow.

Carter-Tracy Infinite Radial Aquifer Influence Function (Default)					
No.	Infinite Dimensionless		No.	Infinite Dimensionless	
	t_D	p_D		t	p_D
1	1.0 x 10^{-2}	0.1120	27	20,00	1.9600
2	5.0 x 10^{-2}	0.2290	28	25,00	2.0670
3	1.0 x 10^{-1}	0.3150	29	30,00	2.1470
4	1.5 x 10^{-1}	0.3760	30	40,00	2.2820
5	2.0 x 10^{-1}	0.4240	31	50,00	2.3880
6	2.5 x 10^{-1}	0.4690	32	60,00	2.4760
7	3.0 x 10^{-1}	0.5030	33	70,00	2.5500
8	4.0 x 10^{-1}	0.5640	34	80,00	2.6150
9	5.0 x 10^{-1}	0.6160	35	90,00	2.6720
10	6.0 x 10^{-1}	0.6590	36	100,00	2.7230
11	7.0 x 10^{-1}	0.7020	37	150,00	2.9210
12	8.0 x 10^{-1}	0.7350	38	200,00	3.0640
13	9.0 x 10^{-1}	0.7720	39	250,00	3.1730
14	1.0	0.8020	40	300,00	3.2630
15	1.5	0.9270	41	400,00	3.4060
16	2.0	1.0200	42	500,00	3.5160
17	2.5	1.1010	43	600,00	3.6080
18	3.0	1.1690	44	700,00	3.6840
19	4.0	1.2750	45	800,00	3.7500
20	5.0	1.3620	46	900,00	3.8090
21	6.0	1.4360	47	1000,00	3.8600
22	7.0	1.5000	48	2000,00	4.1801
23	8.0	1.5560	49	5000,00	4.6350

Carter-Tracy Infinite Radial Aquifer Influence Function (Default)					
No.	Infinite Dimensionless		No.	Infinite Dimensionless	
24	9.0	1.6040	50	8000,00	4.8732
25	10.0	1.6510	51	10000,00	4.9815
26	15.0	1.8290			

Table 8.23: Carter-Tracy Infinite Radial Aquifer Influence Function (Default)

For an overview of analytical aquifers see Dake[182].

Example

```
--
--       CARTER-TRACY AQUIFER INFLUENCE TABLES
--       (STARTS FROM TABLE NO. 2, AS DEFAULT IS TABLE NO. 1)
--
AQUTAB
--
--         TD          PD
--       -------    ---------
          0.06        0.251
          0.08        0.288
          0.10        0.322
          0.12        0.355
          0.14        0.387
          0.16        0.420
          0.18        0.452
          0.20        0.484
          0.22        0.516
          0.24        0.548
          0.26        0.580
          0.28        0.612
          0.30        0.644
          0.35        0.724
          0.40        0.804
          0.45        0.884
          0.50        0.964
          0.55        1.044
          0.60        1.124                          / RD=1.5 TABLE NO. 02
--
--         TD          PD
--       -------    ---------
          0.22        0.443
          0.24        0.459
          0.26        0.476
          0.28        0.492
          0.30        0.507
          0.32        0.522
          0.34        0.536
          0.36        0.551
          0.38        0.565
          0.40        0.579
          0.42        0.593
          0.44        0.607
```

[182] *Dake, L.P. Fundamentals of Reservoir Engineering, Amsterdam, The Netherlands, Elsevier Science BV (1978) Chapter 9.*

```
0.46        0.621
0.48        0.634
0.50        0.648
0.6         0.715
0.7         0.782
0.8         0.849
0.9         0.915
1.0         0.982
2.0         1.649
3.0         2.316
5.0         3.649                                   / RD=2.0 TABLE NO. 03
```

The above example defines tables two and three Carter-Tracy aquifer influence tables.

8.3.13 BDENSITY – DEFINE THE SURFACE BRINE DENSITY FOR THE FLUID

Description

BDENSITY defines the brine surface density for when the brine phase has been activated in the model by the BRINE keyword in the RUNSPEC section. The number of BDENSITY vector data sets is defined by the NTPVT parameter on the TABDIMS keyword in the RUNSPEC section. Each record consists of a maximum of NPPVT values, as declared on the TABDIMS keyword in the RUNSPEC section, with each value representing a brine surface density.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

The keyword is used in conjunction with the PVTWSALT keyword in the PROPS section, with each brine density value matching with the salt concentration values in column 1 of each table in the PVTWSALT keyword. Note that the BDENSITY keyword is optional, and if absent from the input file, then the brine surface densities will be set to the water density values declared via the DENSITY keyword in the PROPS section. In this case there is no variation in brine surface density with respect to salt concentration.

No.	Name	Description			Default
1	WATDEN	**Field**	**Metric**	**Laboratory**	
		WATDEN is a real monotonically increasing positive row vector that defines the brine density at surface conditions for the given salt concentrations on the corresponding PVTWSALT keyword in the PROPS section. There should be one row element for each salt concentration columnar element (SALTCON) on the PVTWSALT keyword.			
		lb/ft^3	kg/m	gm/cc	None
Notes: 1) Each row vector data set must be terminated by a "/" including the last data set.					

Table 8.24: BDENSITY Keyword Description

Note

In OPM Flow the tracer equations are solved decoupled from the reservoir equations at the end of a time step. For each tracer an implicit system is solved, however, the tracer equations are linear, resulting in converge in two iterations. However, the Brine phase is solved fully implicitly and is fully coupled with the other flow equations.

This is different to the commercial simulator, where the tracer equations are solved explicitly after the flow equations have converged at the end of a time step. This can lead to numerical instabilities if there are large variations in brine densities.

Example

The following shows the BDENSITY and PVTWSALT keywords for when NTPVT on the TABDIMS keyword in the RUNSPEC section is set equal to two and NPPVT is set to greater than four on the TABDIMS keyword.

```
--
--       BRINE WATER DENSITY DATA FOR PVTWSALT KEYWORD
--
--       SALTCON  SALTCON  SALTCON  SALTCON  SALTCON
--       DENSITY  DENSITY  DENSITY  DENSITY  DENSITY
--       -------  -------  -------  -------  -------
BDENSITY
         62.20    63.50    64.75    65.90              / FOR PVTWSALT TABLE 1
         64.00    65.50    67.00                       / FOR PVTWSALT TABLE 2
--
--       WATER SALT PVT TABLE
--
PVTWSALT
--       REF PRES  REF SALT
--       PSIA      LB/STB
--       --------  --------
         4500.0    0.000                               / TABLE NO. REF. DATA
--
--       SALTCONC  BW         CW        VISC     VISC
--       LB/STB    RB/STB     1/PSIA    CPOISE   GRAD
--       --------  --------   -------   ------   ------
            0.0    1.020      2.7E-6    0.370    0.0
            2.0    1.010      2.7E-6    0.370    0.0
            4.0    1.000      2.7E-6    0.370    0.0
           10.0    0.950      2.7E-6    0.370    0.0     / TABLE NO. 01 SALT DATA
--
--       REF PRES  REF SALT
--       PSIA      LB/STB
--       --------  --------
         4000.0    0.000                               / TABLE NO. 02 REF. DATA
--
--       SALTCONC  BW         CW        VISC     VISC
--       LB/STB    RB/STB     1/PSIA    CPOISE   GRAD
--       --------  --------   -------   ------   ------
            0.0    1.005      2.5E-6    0.320    0.0
            6.0    0.985      2.5E-6    0.320    0.0
           12.0    0.930      2.5E-6    0.320    0.0     / TABLE NO. 02 SALT DATA
```

8.3.14 BGGI - DEFINE GI GAS FORMATION VOLUME FACTOR PRESSURE TABLES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The BGGI keyword defines Gi gas formation volume factor as a function of Gi and pressure for when the Gi option has been invoked via the GIMODEL keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

The accuracy of gas condensate and volatile oil modeling using a black-oil reservoir simulator is now firmly accepted in the industry; both depletion and gas cycling above the saturation point can be modeled and yield an acceptable match compared with compositional simulators. The main deficiency with the existing black-oil formulation is the treatment of gas injection below the saturation point, where the compositional effect of the stripping of liquid components is inversely proportional to their molecular weights, is not considered. This is because the black-oil model assumes that the saturated hydrocarbon fluid properties are only functions of pressure. Thus, when dry gas is injected into a condensate reservoir below the saturation pressure the injected gas continues to re-vaporize liquid at a rate governed only by the cell pressures. Compositional modeling indicate that this not the case.

The GI option attempts to overcome the limitation of the standard black-oil approach by extending the black-oil model using the method of Cook et al.[183] to take into account the fluid property changes occurring during gas injection. This is done by extending the fluid property treatment so that the saturated fluid properties depend on pressure (as per the standard black-oil formulation) and also on an additional parameter which characterizes the compositional changes in the reservoir liquid and vapor phases at constant pressure.

See also the PVTG and GINODE keywords in the PROPS section.

[183] *Cook, R. E., Jacoby, R. H., and Ramesh, A. B. "A Beta-type Reservoir Simulator for Approximating Compositional Effects During Gas Injection" paper SPE 4272, Society of Petroleum Engineers Journal (1974) 14, No. 5, 471-481.*

8.3.15 BOGI - DEFINE GI OIL FORMATION VOLUME FACTOR PRESSURE TABLES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The BOGI keyword defines Gi oil formation volume factor as a function of Gi and pressure for when the Gi option has been invoked via the GIMODEL keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

The accuracy of gas condensate and volatile oil modeling using a black-oil reservoir simulator is now firmly accepted in the industry; both depletion and gas cycling above the saturation point can be modeled and yield an acceptable match compared with compositional simulators. The main deficiency with the existing black-oil formulation is the treatment of gas injection below the saturation point, where the compositional effect of the stripping of liquid components is inversely proportional to their molecular weights, is not considered. This is because the black-oil model assumes that the saturated hydrocarbon fluid properties are only functions of pressure. Thus, when dry gas is injected into a condensate reservoir below the saturation pressure the injected gas continues to re-vaporize liquid at a rate governed only by the cell pressures. Compositional modeling indicate that this not the case.

The GI option attempts to overcome the limitation of the standard black-oil approach by extending the black-oil model using the method of Cook et al.[184] to take into account the fluid property changes occurring during gas injection. This is done by extending the fluid property treatment so that the saturated fluid properties depend on pressure (as per the standard black-oil formulation) and also on an additional parameter which characterizes the compositional changes in the reservoir liquid and vapor phases at constant pressure.

See also the PVTG and GINODE keywords in the PROPS section.

[184] *Cook, R. E., Jacoby, R. H., and Ramesh, A. B. "A Beta-type Reservoir Simulator for Approximating Compositional Effects During Gas Injection" paper SPE 4272, Society of Petroleum Engineers Journal (1974) 14, No. 5, 471-481.*

8.3.16 BOX - DEFINE A RANGE OF GRID BLOCKS TO ENTER PROPERTY DATA

BOX defines a range of grid blocks for which subsequent data is assigned for all the cells in the defined BOX. Note that the BOX grid is reset by the keyword ENDBOX by resetting the current defined BOX to be the whole grid. The keyword can be used for any array and for all grid types.

See *BOX - Define a Range of Grid Blocks to Enter Property Data* in the GRID section for a full description.

8.3.17 CNAMES – DEFINE COMPOSITIONAL COMPONENT NAMES

Description

The CNAMES keyword defines the names of the compositional components in the model, as such it should have the same number of entries as that declared via the COMPS keyword in the RUNSPEC section, and the NCOMPS keyword in the PROPS section. The keyword should only be used if the CO2STORE keyword and either the GASWAT or the GAS and WATER keywords in the RUNSPEC section, have also be activated for the gas-water two component model.

Note

This is an OPM Flow keyword used with OPM Flow's CO2STORE and GASWAT keywords in the RUNSPEC section, and should not be confused with the more general version of the CNAMES keyword used in the commercial compositional simulator.

Secondly, although OPM Flow parses the keyword, the simulator currently ignores the data for this keyword.

No.	Name	Description	Default
1	CNAMES	A series of NCOMPS character strings, enclosed in quotes, that define the names of the of the compositional components active in the model. Each character string can be up to eight characters in length. Note that the number of component names should be the same as that enter via the NCOMPS keyword in the PROPS section, and the COMPS keyword in the RUNSPEC section. Secondly, only the default value of two components are currently supported by OPM Flow.	None

Notes:

1) The keyword is terminated by a "/".

Table 8.25: CNAMES Keyword Description

Example

The following example defines how to confirm a two component formulation, together with defining the names of the composition components, to be used with the CO2STORE and GASWAT options.

```
--
--       CONFIRM NUMBER OF COMPOSITIONAL COMPONENTS (OPM FLOW KEYWORD)
--
NCOMPS
        2                                                                 /
--
--       DEFINE COMPOSITIONAL COMPONENTS NAMES (OPM FLOW KEYWORD)
--
CNAMES
        'CO2'
        'H2O'                                                             /
```

8.3.18 COALADS – DEFINE GAS AND SOLVENT RELATIVE ADSORPTION TABLES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The COALADS keyword defines the gas and solvent relative adsorption tables for when the coal phase has been activated via the COAL keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.19 COALPP – DEFINE GAS AND SOLVENT PARTIAL PRESSURE ADSORPTION TABLES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The COALPP keyword defines the gas and solvent partial pressure adsorption tables for when the coal phase has been activated via the COAL keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.20 COLUMNS – DEFINE INPUT FILE COLUMN MARGINS

The COLUMNS keyword defines the input file column margins; characters outside the margins are ignored by the input parser.

See *COLUMNS – Define Input File Column Margins* in the GLOBAL section for a full description.

8.3.21 COPY – COPY ARRAY DATA TO ANOTHER ARRAY

The COPY keyword copies an array (or part of an array) to another array or part of an array. The arrays can be real or integer depending on the array type; however, the arrays that can be operated on is dependent on which section the COPY keyword is being used.

See *COPY – Copy Array Data to Another Array* in the GRID section for a full description.

8.3.22 COPYBOX – COPY ARRAY DATA DEFINED BY A BOX

The COPYBOX keyword copies an array (or part of an array) to part of the same array. The array can be real or integer depending on the array type; however, the arrays that can be operated on is dependent on which section the COPYBOX keyword is being used.

See *COPYBOX – Copy Array Data Defined by a Box* in the GRID section for a full description.

8.3.23 COPYREG – COPY AN ARRAY TO ANOTHER ARRAY BASED ON A REGION NUMBER

The COPYREG keyword copies a specified array or part of an array based on cells with a specific region number to another array. The region number array can be FLUXNUM, MULTNUM or OPERNUM and these arrays must be defined and be available before the COPYREG keyword is read by the simulator. The property arrays can be real or integer depending on the property array type; however, the property arrays that can be operated on is dependent on which section the COPYREG keyword is being used.

See *COPYREG – Copy an Array to Another Array based on a Region Number* in the GRID section for a full description.

8.3.24 DEBUG – DEFINE THE DEBUG DATA TO BE PRINTED TO FILE

This keyword defines the debug data to be written to the debug file (*.DBG). This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

See *DEBUG – Define the Debug Data to be Printed to File* in the GLOBAL section for a full description.

8.3.25 DENSITY – DEFINE THE SURFACE OIL, WATER GAS DENSITIES FOR THE FLUIDS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

DENSITY defines the oil, water and gas surface densities for the fluids for various regions in the model. The number of DENSITY vector data sets is defined by the NTPVT parameter on the TABDIMS keyword in the RUNSPEC section and the allocation of the DENSITY data sets to different grid blocks in the model is done via the PVTNUM keyword in the REGIONS section. One data set consists of one record or line which is terminated by a "/". The surface density or gravity must be entered using either the DENSITY or GRAVITY keywords irrespective of which phases are active in the model.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	OILDEN	OILDEN is a real number defining the density of the oil phase at surface conditions.			
		lb/ft^3 37.457	kg/m^3 600	gm/cc 0.6	Defined
2	WATDEN	WATDEN is a real number defining the density of the water phase at surface conditions.			
		lb/ft^3 62.366	kg/m^3 999.014	gm/cc 0.999014	Defined
3	GASDEN	GASDEN is a real number defining the density of the gas phase at surface conditions.			
		lb/ft^3 0.062428	kg/m^3 1.000	gm/cc 0.001	Defined

Notes:

1) The keyword is followed by NTPVT tables as declared on the TABDIMS keyword in the RUNSPEC section.
2) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.26: DENSITY Keyword Description

Note that provided the first table has been entered, subsequent tables may be defaulted, in this case the prior table is copied to the current table. See the third example for an illustration on how to use this feature.

According to the SPE SI standard[185], ***Relative Density*** (γ) replaces ***Specific Gravity*** as the term used to define the ratio of the density of a known material to the density of reference material, at standard conditions of pressure and temperature. Standard conditions vary throughout the world, but for oil field units one normally uses14.7 psia and 60 °F, whereas for SI units some areas use 101.325 kPa and 15 °C.

See also the GRAVITY keyword in the PROPS section, that can be used to enter the relative density values instead of the density numbers.

[185] *The SI Metric System of Units and SPE Metric Standard, Adopted for Use as a Voluntary Standard by the SPE Board of Directors, June 1983, Society of Petroleum Engineers.*

Examples

The following shows the DENSITY keyword for when NTPVT on the TABDIMS keyword in the RUNSPEC section is set to one.

```
--
--        OIL      WAT        GAS
--        DENSITY  DENSITY    DENSITY
--        -------  -------    -------
DENSITY
          39.0     62.37      0.04520                        / PVT DATA REGION 1
```

The next example shows the DENSITY keyword for when NTPVT on the TABDIMS keyword in the RUNSPEC section is set to three.

```
--
--        OIL      WAT        GAS
--        DENSITY  DENSITY    DENSITY
--        -------  -------    -------
DENSITY
          38.0     62.30      0.04500                        / PVT DATA REGION 1
          39.0     62.37      0.04520                        / PVT DATA REGION 2
          40.0     62.40      0.04800                        / PVT DATA REGION 3
```

The third, and final, example shows the DENSITY keyword for when NTPVT on the TABDIMS keyword in the RUNSPEC section is set to four. Here table two defaults to table one, and table four defaults to table three.

```
--
--        OIL      WAT        GAS
--        DENSITY  DENSITY    DENSITY
--        -------  -------    -------
DENSITY
          38.0     62.30      0.04500                        / PVT DATA REGION 1
                                                             / PVT DATA REGION 2
          39.0     62.37      0.04520                        / PVT DATA REGION 3
                                                             / PVT DATA REGION 3
```

Again, note that there is no terminating "/" for this keyword.

8.3.26 DEPTHTAB – RIVER TIME AND DEPTH TABLES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, DEPTHTAB, defines the river time and depth tables, for when the River option has been activated via the RIVRDIMS keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.27 DIAGDISP – ACTIVATE ALTERNATE FORM OF TRACER DISPERSION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, DIAGDISP, activates the alternate form of tracer dispersion matrix for when the Tracer facility has been activated by the TRACERS keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.28 DIFFC – DEFINE PVT REGION MOLECULAR DIFFUSION TABLES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The DIFFC keyword defines the molecular weight of the fluids and diffusion coefficients between phases for each PVT region, for when the Molecular Diffusion option has been activated by the DIFFUSE keyword in the RUNSPEC section. This keyword is optional as OPM Flow will automatically calculate the coefficients, as described by Sandve et al.[186], if the DIFFC keyword is absent from the input deck. The keyword thus allows one to overwrite the automatically calculated values.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	OILMW	OILMW is a real positive number that specifies the molecular weight of the oil in the given PVT region.			None
		lb/lb-M	kg/kg-M	gm/gm-M	
2	GASMW	GASMW is a real positive number that defines the molecular weight of the gas in the given PVT region.			None
		lb/lb-M	kg/kg-M	gm/gm-M	
3	GASGASDF	A real positive number that defines the gas in gas diffusion coefficient in the given PVT region.			None
		ft^2/day	m^2/day	cm^2/hour	
4	OILGASDF	A real positive number that declares the oil in gas diffusion coefficient in the given PVT region.			None
		ft^2/day	m^2/day	cm^2/hour	
5	GASOILDF	A real positive number that specifies the gas in oil diffusion coefficient in the given PVT region.			None
		ft^2/day	m^2/day	cm^2/hour	
6	OILOILDF	A real positive number that defines the oil in oil diffusion coefficient in the given PVT region.			None
		ft^2/day	m^2/day	cm^2/hour	
7	GASOILCD	A real positive number that defines the gas in oil cross phase diffusion coefficient in the given PVT region. This parameter is ignored by OPM Flow and should be defaulted or set equal to zero.			0.0
		ft^2/day	m^2/day	cm^2/hour	

[186] *Tor Harald Sandve1, Sarah E. Gasda, Atgeirr Rasmussen, and Alf Birger Rustad. Convective dissolution in field scale CO_2 storage simulation using the OPM Flow simulator. Submitted to TCCS 11 – Trondheim Conference on CO_2 Capture, Transport and Storage Trondheim, Norway – June 21-23, 2021.*

No.	Name	Description			Default
		Field	Metric	Laboratory	
8	OILOILCD	A real positive number that defines the oil in oil cross phase diffusion coefficient in the given PVT region. This parameter is ignored by OPM Flow and should be defaulted or set equal to zero.			0.0
		ft^2/day	m^2/day	cm^2/hour	

Notes:

1) The keyword is followed by NTPVT tables as declared on the TABDIMS keyword in the RUNSPEC section.
2) Each record is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.27: DIFFC Keyword Description

Note

The option has been tested in combination with the CO2STORE keyword, but not for the general case at this point.

See also the DIFFUSE keyword in the RUNSPEC section to activate the Molecular Diffusion option.

Example

The example below is based on field units, with NTPVT equal to three on the TABDIMS keyword.

```
--
--       PVT REGION MOLECULAR DIFFUSION TABLES
--
DIFFC
--       OIL     GAS    GAS IN   OIL IN   GAS IN   OIL IN   GAS IN   OIL IN
--       MW      MW     GAS DF   GAS DF   OIL DF   OIL DF   OIL CD   OIL CD
--       ------  -----  -------  -------  -------  -------  -------  ------
         103.20  1.120  1.35E-6  1.05E-7  4.50E-7  1.05E-8                /TAB-1
         102.00  1.130  1.25E-6  1.25E-7  4.80E-7  1.05E-8                /TAB-2
         100.00  1.250  1.22E-6                                          /TAB-3
```

Here the third PVT region has no values for the various oil related diffusion coefficients.

8.3.29 DIFFCGAS – DEFINE PVT REGION GAS COMPONENT DIFFUSION COEFFICIENTS

Description

The DIFFCGAS keyword defines the gas diffusion coefficients for each compositional component in the model and for each PVT region, for when the Molecular Diffusion option has been activated by the DIFFUSE keyword in the RUNSPEC section. This keyword is optional as OPM Flow will automatically calculate the coefficients, as described by Sandve et al.[187], if the DIFFCGAS keyword is absent from the input deck. The keyword thus allows one to overwrite the automatically calculated values.

The keyword should only be used if the CO2STORE keyword and either the GASWAT or the GAS and WATER keywords in the RUNSPEC section, have also be activated for the gas-water two component model.

Note

This is an OPM Flow keyword used with OPM Flow's CO2STORE and GASWAT keywords in the RUNSPEC section, and should not be confused with the more general version of the DIFFCGAS keyword used in the commercial compositional simulator.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	CO2DIFF	A real positive number that declares the CO2 in gas diffusion coefficient in the given PVT region.			None
		ft^2/day	m^2/day	cm^2/hour	
2	WATDIFF	A real positive number that specifies the water in gas diffusion coefficient in the given PVT region.			None
		ft^2/day	m^2/day	cm^2/hour	

Notes:

1) The keyword is followed by NTPVT tables as declared on the TABDIMS keyword in the RUNSPEC section.
2) Each record is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.28: DIFFCGAS Keyword Description

Note

The option has been tested in combination with the CO2STORE keyword, but not for the general case at this point.

See also the DIFFUSE keyword in the RUNSPEC section to activate the Molecular Diffusion option and the DIFFCWAT keyword in the PROPS section that define the water diffusion coefficients for each compositional component in the model and for each PVT region. Finally, for gas-oil systems the DIFFC keyword in the PROPS section should be used.

[187] *Tor Harald Sandve1, Sarah E. Gasda, Atgeirr Rasmussen, and Alf Birger Rustad. Convective dissolution in field scale CO_2 storage simulation using the OPM Flow simulator. Submitted to TCCS 11 – Trondheim Conference on CO_2 Capture, Transport and Storage Trondheim, Norway – June 21-23, 2021.*

Normally diffusion coefficients are measured in laboratory units, that is cm^2/s, for easy of use, Table 8.29 outlines the conversion factors for converting the laboratory measured diffusion coefficients to those used by the simulator.

Diffusivity Conversion Factors		
Laboratory Measured Units	**Conversion Factor**	**Simulator Units**
1 cm^2/s	92.9979 ft^2/day	Field
	8.64 m^2/day	Metric
	3600 cm2/hour	Laboratory

Table 8.29: Diffusivity Conversion Factors

Example

The example below is based on field units, with NTPVT equal to three on the TABDIMS keyword.

```
--
--       PVT REGION GAS COMPONENT DIFFUSION COEFFICIENTS (OPM FLOW KEYWORD)
--
DIFFCGAS
--       CO2 IN   WAT IN
--       GAS DF   GAS DF
--       -------  -------
         0.160    0.150               / PVT REGION NO. 01
         0.165    0.155               / PVT REGION NO. 02
                                      / PVT REGION NO. 03
```

Here the third PVT region has no values for the two component diffusion coefficients, and therefore the simulator will use correlations to define the diffusivity coefficients for this PVT region.

8.3.30 DIFFCOAL – DEFINE COAL BED METHANE GAS DIFFUSION DATA

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The DIFF keyword defines the coal bed methane diffusion data for when the Coal Bed Methane option has been activated via the COAL keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.31 DIFFCWAT – DEFINE PVT REGION WATER COMPONENT DIFFUSION COEFFICIENTS

Description

The DIFFCWAT keyword defines the water diffusion coefficients for each compositional component in the model and for each PVT region, for when the Molecular Diffusion option has been activated by the DIFFUSE keyword in the RUNSPEC section. This keyword is optional as OPM Flow will automatically calculate the coefficients, as described by Sandve et al.[188], if the DIFFCWAT keyword is absent from the input deck. The keyword thus allows one to overwrite the automatically calculated values.

The keyword should only be used if the CO2STORE keyword and either the GASWAT or the GAS and WATER keywords in the RUNSPEC section, have also be activated for the gas-water two component model.

Note

This is an OPM Flow keyword used with OPM Flow's CO2STORE and GASWAT keywords in the RUNSPEC section, and should not be confused with the more general version of the DIFFCWAT keyword used in the commercial compositional simulator.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	CO2DIFF	A real positive number that declares the CO2 in water diffusion coefficient in the given PVT region.			None
		ft^2/day	m^2/day	cm^2/hour	
2	WATDIFF	A real positive number that specifies the water in water diffusion coefficient in the given PVT region.			None
		ft^2/day	m^2/day	cm^2/hour	

Notes:

1) The keyword is followed by NTPVT tables as declared on the TABDIMS keyword in the RUNSPEC section.
2) Each record is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.30: DIFFCWAT Keyword Description

Note

The option has been tested in combination with the CO2STORE keyword, but not for the general case at this point.

See also the DIFFUSE keyword in the RUNSPEC section to activate the Molecular Diffusion option and the DIFFCGAS keyword in the PROPS section that define the gas diffusion coefficients for each compositional component in the model and for each PVT region. Finally, for gas-oil systems the DIFFC keyword in the PROPS section should be used.

[188] *Tor Harald Sandve1, Sarah E. Gasda, Atgeirr Rasmussen, and Alf Birger Rustad. Convective dissolution in field scale CO_2 storage simulation using the OPM Flow simulator. Submitted to TCCS 11 – Trondheim Conference on CO_2 Capture, Transport and Storage Trondheim, Norway – June 21-23, 2021.*

Normally diffusion coefficients are measured in laboratory units, that is cm^2/s, for easy of use Table 8.31 outlines the conversion factors for converting the laboratory measured diffusion coefficients to those used by the simulator.

Diffusivity Conversion Factors		
Laboratory Measured Units	**Conversion Factor**	**Simulator Units**
1 cm^2/s	92.9979 ft^2/day	Field
	8.64 m^2/day	Metric
	3600 cm2/hour	Laboratory

Table 8.31: Diffusivity Conversion Factors

Example

The example below is based on field units, with NTPVT equal to three on the TABDIMS keyword.

```
--
--       PVT REGION WATER COMPONENT DIFFUSION COEFFICIENTS (OPM FLOW KEYWORD)
--
DIFFCWAT
--       CO2 IN   WAT IN
--       WAT DF   WAT DF
--       -------  -------
         0.160    0.150               / PVT REGION NO. 01
         0.165    0.155               / PVT REGION NO. 02
                                      / PVT REGION NO. 03
```

Here the third PVT region has no values for the two component diffusion coefficients, and therefore the simulator will use correlations to define the diffusivity coefficients for this PVT region.

8.3.32 DIFFDP – ACTIVATE DUAL POROSITY MOLECULAR DIFFUSION FOR MATRIX-FRACTURE FLOW ONLY

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, DIFFDP, activates the dual porosity molecular diffusion for matrix-fracture flow only option for when the Dual Porosity option has be activated by either the DUALPORO or DUALPERM keywords, and the Diffusivity option has been activated by the DIFFUSE keywords; three keywords are in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.33 DIFFMMF – DEFINE DIFFUSIVITY MULTIPLIERS FOR MATRIX-FRACTURES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, DIFFMMF, defines the diffusivity multipliers for matrix-fractures for when the Dual Porosity option has be activated by either the DUALPORO or DUALPERM keywords, or the Coal Bed Methane option is selected by the COAL keyword, and the Diffusivity option has been activated by the DIFFUSE keywords; all four keywords are in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.34 DPKRMOD – MODIFY MATRIX OIL RELATIVE PERMEABILITY DATA

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The DPKRMOD keyword can be used to modify the matrix oil relative permeability data (oil-water, oil-gas) and the scaling of the fracture to matrix relative permeabilities, for dual porosity runs for when a Dual Porosity model has been activated by either the DUALPORO or DUALPERM keywords in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.35 DSPDEINT – ACTIVATE BRINE TRACER DISPERSION INTERPOLATION BY WATER DENSITY

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, DSPDEINT, activates the brine tracer dispersion interpolation by water density option for when the Brine phase is activated in the model by the BRINE keyword in the RUNSPEC section and the DISPERSE keyword in the PROPS section is in the input file. They keyword cause the lookup and interpolation of the DISPERSE tracer concentration to water density, that is the tracer concentration data on the DISPERSE keyword has been replaced by the water density data.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.36 ECHO – ACTIVATE ECHOING OF USER INPUT FILES TO THE PRINT FILE

Turns on echoing of all the input files to the print file; note that this keyword is activated by default and can subsequently be switched off by the NOECHO activation keyword.

See *ECHO – Activate Echoing of User Input Files to the Print File* in the GLOBAL section for a full description.

8.3.37 EHYSTR – DEFINE HYSTERESIS MODEL AND PARAMETERS

Description

The EHYSTR keyword defines the hysteresis model and associated parameters when the hysteresis option has been activated by the HYSTER variable on the SATOPTS keyword in the RUNSPEC section. Both the Carlson[189] and Killough[190] models are available.

<table>
<tr><th>No.</th><th>Name</th><th>Description</th><th>Default</th></tr>
<tr><td>1</td><td>HYSTRCP</td><td>HYSTRCP is a positive real value that defines the Killough curvature parameter for capillary pressure hysteresis model.
The value should range from 0.05 to 0.10.
This option is ignored by OPM Flow.</td><td>0.1</td></tr>
<tr><td>2</td><td>HYSTMOD</td><td>An integer value that determines the relative permeability hysteresis model to be used depending on the phase and the wettability of the system. HYSTMOD should be set to one of the following values:
<table>
<tr><td colspan="3">Water Wet Hysteresis Models</td></tr>
<tr><td>HYSMOD</td><td>Non-Wetting Phases</td><td>Wetting Phase</td></tr>
<tr><td>-1</td><td colspan="2">Equilibration option for equilibrating the model with the SATNUM (drainage curves) and running the model with imbibition curves (IMBNUM).
This option implies no hysteresis.</td></tr>
<tr><td>0</td><td>Carlson Hysteresis Model</td><td>SATNUM</td></tr>
<tr><td>1</td><td>Carlson Hysteresis Model</td><td>IMBNUM</td></tr>
<tr><td>2</td><td>Killough Hysteresis Model</td><td>SATNUM</td></tr>
<tr><td>3</td><td>Killough Hysteresis Model</td><td>IMBNUM</td></tr>
<tr><td>4</td><td>Killough Hysteresis Model</td><td>Killough Hysteresis Model</td></tr>
<tr><td colspan="3">Oil Wet to Water Wet Water Wet</td></tr>
<tr><td>5</td><td>Carlson Non-Wetting Modeling for Gas and Water</td><td>SATNUM</td></tr>
<tr><td>6</td><td>Killough Non-Wetting Modeling for Gas and Water</td><td>SATNUM</td></tr>
<tr><td>7</td><td>Killough Non-Wetting Modeling for Gas and Water</td><td>Killough Non-Wetting Modeling for the Wetting Oil Phase</td></tr>
<tr><td colspan="3">Only the HYSMOD options not colored orange are currently support by OPM Flow.</td></tr>
</table></td><td>0</td></tr>
</table>

[189] *Carlson, F. M. "Simulation of Relative Permeability Hysteresis to the Non-Wetting Phase," paper SPE 10157, presented at the SPE Annual Technical Conference & Exhibition, San Antonio, Texas, USA (October 5-7, 1981).*

[190] *Killough, J. E. "Reservoir Simulation with History-dependent Saturation Functions," paper SPE 5106, Society of Petroleum Engineers Journal (1976) 16, No. 1, 37-48.*

No.	Name	Description	Default
3	HYSTREL	HYSTREL is a positive real number that defines the Killough's wetting phase relative permeability curvature parameter. This parameter is only applicable if HYSMOD is set to either 4 or 7. This option is ignored by OPM Flow.	1.0
4	HYSTSGR	HYSTSGR is a positive real number that sets a scaling parameter for the trapped non-wetting phase saturation in the Killough model. This option is ignored by OPM Flow.	0.1
5	HYSTOPT	A character string that determines if the hysteresis model should be activated for relative permeability, capillary pressure curves, or both, and should be set to one of the following: 1) BOTH: apply hysteresis modeling to both relative permeability, and capillary pressure curves. 2) PC: apply hysteresis modeling to capillary pressure curves only. 3) KR: apply hysteresis modeling to relative permeability curves only. For all cases HYSTMOD defines the model to be used for relative permeability hysteresis modeling (if applicable). Capillary pressure hysteresis always uses the Killough capillary pressure model. All three options are supported by OPM Flow.	BOTH
6	HYSTSCAN	A character string that determines the shape of Killough capillary pressure scanning curves when secondary reversal curves, that is for a drainage, imbibition, drainage cycle. 1) RETR: Secondary drainage curves re-traverses the same scanning curve. 2) NEW: Secondary drainage curves follows a new scanning curve and further reversals also generate a new scanning curve. This option is ignored by OPM Flow.	RETR
7	HYSTMOB	A character string that determines how to apply the mobility control correction invoked by the MOBILE variable on the EQLOPTS keyword in the RUNSPEC section. HYSTMOB should be set to one of the following: 1) DRAIN: Only the drainage curve end-points are modified. 2) BOTH: Both the drainage and imbibition curve end-points are modified. The Mobility Control option is not supported in OPM Flow so this parameter has no effect.	DRAIN
8	HYSTWET	A character string that sets the wetting phase in three phase systems to either oil or gas and should be set to one of the following: 1) OIL: Oil is set as the wetting phase and the oil-gas relative permeability curves are determined by HYSTMOD for the wetting phase. 2) GAS: Oil is set as the non-wetting phase and the oil-gas relative permeability curves are determined by HYSTMOD for the non-wetting phase. Note for all the above cases the gas relative permeability curves are always treated as as a non-wetting phase. This option is ignored by OPM Flow.	None

No.	Name	Description	Default
9	HYBAKOIL	Baker model one or model two relative permeability oil phase hysteresis option used in the commercial compositional simulator. Not used and should be defaulted with 1*.	NO
10	HYBAKGAS	Baker model one or model two relative permeability gas phase hysteresis option used in the commercial compositional simulator. Not used and should be defaulted with 1*.	NO
11	HYBAKWAT	Baker model one or model two relative permeability water phase hysteresis option used in the commercial compositional simulator. Not used and should be defaulted with 1*.	NO
12	HYTHRESH	Killough's hysteresis threshold saturation value used in the commercial compositional simulator. Not used and should be defaulted with 1*.	0.0
13	HYSWETRP	Killough's hysteresis wetting phase modification used in the commercial black-oil simulator. Not used and should be defaulted with 1*.	0
Notes: 1) The keyword is terminated by a "/".			

Table 8.32: EHYSTR Keyword Description

Example

```
--
--        HYSTERESIS MODEL AND PARAMETERS
--
--        PC-CUR  MODEL   RELPERM TRAPPED OPTION  SHAPE    MOBILIT  WET
--        HYSTRCP HYSTMOD HYSTREL HYSTSGR HYSTOPT HYSTSCAN HYSTMOB  HYSTWET
EHYSTR
          0.1     0       0.1     1*      KR      1*       1*       1*          /
```

The above example defines the hysteresis model and parameters used in the Norne model. Here the default value is used for the Killough curvature parameter for capillary pressure hysteresis mode, the Carlson hysteresis model is used for the non-wetting phase and SATNUM for the wetting phase, 0.1 is used for Killough's wetting phase relative permeability curvature parameter (this parameter is ignored because the Carlson model has been selected), the default values for the trapped non-wetting phase saturation in the Killough mode (again, this parameter is ignored because the Carlson model has been selected, and the hysteresis modeling is only applied to relative permeability curves.

8.3.38 EHYSTRR – DEFINE HYSTERESIS MODEL AND PARAMETERS VIA SATNUM

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The EHYSTRR keyword defines the hysteresis model and associated parameters via the drainage SATNUM allocation region array, for when the hysteresis option has been activated by the HYSTER variable on the SATOPTS keyword in the RUNSPEC section. Only the Killough[191] model is available for this keyword and the keyword is optional.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped. See also the EHYSTR keyword in the RUNSPEC for an alternative keyword to enter the hysteresis model and associated parameters that is supported by OPM Flow

No.	Name	Description	Default
1	HYSTRCP	HYSTRCP is a positive real value that defines the Killough curvature parameter for capillary pressure hysteresis model. The value should range from 0.05 to 0.10.	0.1
2	HYSTREL	HYSTREL is a positive real number that defines the Killough's wetting phase relative permeability curvature parameter. This parameter is ignored if HYSMOD on the EHYSTR keyword is not set to 4.	1.0
3	HYSTSGR	HYSTSGR is a positive real number that sets a scaling parameter for the trapped non-wetting phase saturation in the Killough model.	0.1

Notes:

1) The keyword is followed by NTSFUN rows as declared on the TABDIMS keyword in the RUNSPEC section.
2) There is no "/" terminator for the keyword.

Table 8.33: EHYSTRR Keyword Description

Example

```
--
--       HYSTERESIS MODEL AND PARAMETERS VIA SATNUM
--
--       PC-CUR  RELPERM TRAPPED
--       HYSTRCP HYSTREL HYSTSGR
EHYSTRR
         0.04    1.0     1*                          / SATNUM REGION 1
         0.06    1.0     1*                          / SATNUM REGION 2
         0.08    1.0     1*                          / SATNUM REGION 3
         0.10    1.0     1*                          / SATNUM REGION 4
         0.10    1.0     1*                          / SATNUM REGION 5
```

The above example defines the hysteresis model and parameters for when NTSFUN equals five on the TABDIMS keyword in the RUNSPEC section, that is for five SATNUM regions.

[191] *Killough, J. E. "Reservoir Simulation with History-dependent Saturation Functions," paper SPE 5106, Society of Petroleum Engineers Journal (1976) 16, No. 1, 37-48.*

8.3.39 END – DEFINE THE END OF THE INPUT FILE

This keyword marks the end of the input file and can occur in any section. Any keywords and data after the END keyword are ignored.

See *END – Define the End of the Input File* in the GLOBAL section for a full description.

8.3.40 ENDBOX – DEFINE THE END OF THE BOX DEFINED GRID

This keyword marks the end of a previously defined BOX sub-grid as defined by a previously entered BOX keyword. The keyword resets the input grid to be the full grid as defined by the NX, NY, and NZ variables on the DIMENS keyword in the RUNSPEC section.

See *ENDBOX – Define the End of the BOX Defined Grid* in the GRID section for a full description.

8.3.41 ENDFIN – END THE DEFINITION OF A LOCAL GRID REFINEMENT

ENDFIN defines the end of a Cartesian or radial local grid refinement ("LGR") definition and a LGR property definition data set.

See *ENDFIN – End the Definition of a Local Grid Refinement* in the GRID section for a full description.

8.3.42 ENDINC – DEFINE THE END OF AN INCLUDE FILE

This keyword marks the end of an include file specified on the INCLUDE keyword. When the ENDINC keyword is encountered in the INCLUDE file, input data is read from the next keyword in the current file. Any keywords and data after the ENDINC keyword in the INCLUDE file are ignored.

See *ENDINC – Define the End of an Include File* in the GLOBAL section for a full description.

8.3.43 ENDSKIP – DEACTIVATE SKIPPING OF KEYWORDS AND INPUT DATA

Turns off skipping of keywords that was activated by the SKIP, SKIP100, or SKIP300 keywords. Note that for each SKIP keyword activated there must be a corresponding ENDSKIP keyword.

See *ENDSKIP – DeActivate Skipping of Keywords and Input Data* in the GLOBAL section for a full description.

8.3.44 ENKRVD – DEFINE RELATIVE PERMEABILITY END-POINTS VERSUS DEPTH FUNCTIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword defines the maximum oil, gas, and water relative permeability values versus depth for the three phases and for when the end-point scaling option has been invoked by the ENDSCALE keyword in the RUNSPEC section. This functionality is not supported in OPM Flow.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.45 ENPCVD – DEFINE MAXIMUM CAPILLARY PRESSURE VERSUS DEPTH FUNCTIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword defines the maximum gas-oil and water-oil capillary pressure values versus depth for when the end-point scaling option has been invoked by the ENDSCALE keyword in the RUNSPEC section

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.46 ENPTVD – DEFINE RELATIVE PERMEABILITY SATURATION END-POINTS VERSUS DEPTH

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword defines the variation of the relative permeability saturation end-points (SWL, SWCR, etc.) for all three phases versus depth for when the end-point scaling option has been invoked by the ENDSCALE keyword in the RUNSPEC section. This functionality is not supported in OPM Flow.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.47 ENSPCVD – DEFINE CAPILLARY PRESSURE END-POINTS VERSUS DEPTH

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword defines the variation of the capillary pressure saturation end-points, connate gas (SGL) and connate water (SWL), versus depth for when the end-point scaling option has been invoked by the ENDSCALE keyword in the RUNSPEC section. This functionality is not supported in OPM Flow.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.48 EPSDBGS - WRITE END-POINT DEBUG DATA TO THE DEBUG FILE (MULTIPLE)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, EPSDBGS, defines the end-point debug data for multiple grid blocks that should be written to the DEBUG file (*.DBG) for when the End-Point Scaling option has been activated by the ENDSCALE keyword in the RONSPEC section.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

8.3.49 EPSDEBUG - WRITE END-POINT DEBUG DATA TO THE DEBUG FILE (INDIVIDUAL)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, EPSDEBUG, defines the end-point debug data for individual grid blocks that should be written to the DEBUG file (*.DBG) for when the End-Point Scaling option has been activated by the ENDSCALE keyword in the RONSPEC section.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

8.3.50 EQUALREG – SETS AN ARRAY TO A CONSTANT BY REGION NUMBER

The EQUALREG keyword sets a specified array to a constant for cells with a specific region number. The region number array can be FLUXNUM, MULTNUM or OPERNUM and these arrays must be defined and be available before the EQUALREG keyword is read by the simulator. The constant can be real or integer depending on the property array type; however, the property arrays that can be operated on is dependent on which section the EQUALREG keyword is being used.

See *EQUALREG – Sets an Array to a Constant by Region Number* in the GRID section for a full description.

8.3.51 EQUALS – SETS A SPECIFIED ARRAY TO A CONSTANT

The EQUALS keyword sets a specified array or part of an array to a constant. The constant can be real or integer depending on the array type; however, the arrays that can be operated on is dependent on which section the EQUALS keyword is being used.

See *EQUALS – Sets a Specified Array to a Constant* in the GRID section for a full description.

8.3.52 ESSNODE – DEFINE SALT CONCENTRATION DATA FOR WATER-OIL SURFACE TENSION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, ESSNODE, defines the salt concentration data that is used in calculating the water-oil surface tension for when the Brine option has been activated by the BRINE keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.53 EXTRAPMS – ACTIVATE EXTRAPOLATION WARNING MESSAGES

The EXTRAPMS keyword activates extrapolation warning messages for when OPM Flow extrapolates the PVT or VFP tables. Frequent extrapolation warning messages should be investigated and resolved as this would indicate possible incorrect data and may result in the simulator extrapolating to unrealistic values.

See *EXTRAPMS – Activate Extrapolation Warning Messages* in the GLOBAL section for a full description.

8.3.54 FHERCHBL – DEFINE HERSCHEL-BULKLEY DATA VERSUS POLYMER CONCENTRATION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The FHERCHBL keyword defines Herschel-Bulkley rheological property data for Non-Newtonian fluids versus polymer concentration, for when the Polymer option has been invoked via the POLYMER keyword in the RUNSPEC section and Non-Newtonian Fluid phase has been declared active by the NNEWTF keyword, also in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.55 FILEUNIT – ACTIVATE UNIT CONSISTENCY VERIFICATION

The FILEUNIT keyword defines the units of the of the data set, and is used to verify that the units in the input deck and any associated include files are consistent grid data. The keyword does not provide for the conversion between different sets of units.

See *FILEUNIT – Activate Unit Consistency Checking* in the GRID section for a full description.

8.3.56 FILLEPS – ACTIVATE SATURATION END-POINT EXPORT TO THE INIT FILE

Description

This keyword switches on the export of the saturation end-point data (SWL, SWCR, SOWCR array etc.) to the *.INIT file so that the data can be viewed in post-processing software like OPM ResInsight.

There is no data required for this keyword.

Example

```
--
--       ACTIVATE SATURATION END-POINT EXPORT TO THE INIT FILE
--
FILLEPS
```

The above example switches on the export of the end-point saturation data to the *.INIT file.

8.3.57 FOAMADS - DEFINE FOAM ROCK ADSORPTION TABLES

Description

The FOAMADS keyword defines the foam rock adsorption tables for when the Foam option has been activated by the FOAM keyword in the RUNSPEC section.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	FOAMCON	A columnar vector of real monotonically increasing down the column values that defines the foam concentration in the solution surrounding the rock. The first entry should be zero to define a no foam concentration data set. Units are dependent on the transport phase specified via the FOAMOPT1 variable on the FOAMOPTS keywod in the PROPS section.			
		Gas: lb/Mscf Water: lb/stb	Gas: kg/sm^3 Water: kg/sm^3	Gas: gm/scc Water: gm/scc	None
2	FOAMRATI	A columnar vector of real increasing down the column values that defines the mass of adsorbed foam per unit mass of rock for a given FOAMCON. The first table data set entry should be zero to define a no foam concentration data set.			
		lb/lb	kg/kg	gm/gm	None

Notes:

1) The keyword is followed by NTSFUN tables as declared on the TABDIMS keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NSSFUN rows as declared on the TABDIMS keyword in the RUNSPEC section.
3) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.34: FOAMADS Keyword Description

Example

```
--
--        FOAM ROCK ADSORPTION TABLE
--
FOAMADS
--        FOAM        FOAM
--        FOAMCON     FOAMRATI
--        -------     --------
              0.0      0.00000
              2.0      0.00003
              4.0      0.00005
              6.0      0.00007
              8.0      0.00009
             10.0      0.00011
             12.0      0.00012
             14.0      0.00015                              / TABLE NO. 01
--        FOAM        FOAM
--        FOAMCON     FOAMRATI
--        -------     --------
              0.0      0.00000
              3.0      0.00004
              5.0      0.00006
              7.0      0.00008
              8.0      0.00009
             10.0      0.00011                              / TABLE NO. 02
```

The above example defines two foam rock adsorption tables assuming NTSFUN equals two and NSSFUN is greater than or equal to eight on the TABDIMS keyword in the RUNSPEC section.

8.3.58 FOAMDCYO – DEFINE FOAM DECAY VERSUS OIL SATURATION TABLES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The FOAMDCYO keyword defines the foam decay half-life versus oil saturation for when the Foam option has been activated by the FOAM keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.59 FOAMDCYW – DEFINE FOAM DECAY VERSUS WATER SATURATION TABLES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The FOAMDCYW keyword defines the foam decay half-life versus water saturation for when the Foam option has been activated by the FOAM keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.60 FOAMFCN – DEFINE FOAM GAS MOBILITY REDUCTION VERSUS CAPILLARY NUMBER

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The FOAMFCN keyword defines the reduction in gas mobility versus capillary number, for when the Foam option has been activated by the FOAM keyword in the RUNSPEC section, and the FOAMOPT2 parameter on the FOAMOPTS keyword in the PROPS section has been set to the character string FUNC.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.61 FOAMFRM – DEFINE FOAM GAS MOBILITY REDUCTION VERSUS REFERENCE MOBILITY

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The FOAMFRM keyword defines the reduction in gas mobility versus the reference mobility reduction factor, for when the Foam option has been activated by the FOAM keyword in the RUNSPEC section, and the FOAMOPT2 parameter on the FOAMOPTS keyword in the PROPS section has been set to the character string FUNC.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.62 FOAMFSC – DEFINE FOAM GAS MOBILITY VERSUS SURFACTANT CONCENTRATION FUNCTIONS

Description

The FOAMFSC keyword defines the reduction in gas mobility as a function of the foam surfactant concentration within a grid block. The Foam option must be activated by the FOAM keyword in the RUNSPEC section in order to use this keyword. In addition, the FOAMOPT2 parameter on the FOAMOPTS keyword in the PROPS section must be set to the character string FUNC, in order to activate the functional form of the gas mobility reduction calculations.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	FOAMCON	A real positive value that defines the foam surfactant concentration at which foam modeling becomes active in the model and a strong foam is formed. FOAMCON cannot be defaulted and must be specified for the first table. Subsequent tables can be defaulted and will in this case use the previous tables' entries as the default value.			
		lb/stb	kg/sm^3	gm/scc	None
2	FOAMEXP	A real positive value that defines an exponent that determines the gradient in the change of the reduction in gas mobility due to foam (es in equation (8.51). Note if e_s is less than one then the slope of F_s in equation (8.51) will be infinite at C_s equal to zero. In this case, small surfactant concentrations have a significant effect on the mobility, especially if the reference concentration Csr is also small. If this is the case use MINSURF on this keyword to set a minimum surfactant concentration to avoid small-scale numerical errors from affecting the simulation.			
		dimensionless 1.0	dimensionless 1.0	dimensionless 1.0	Defined
3	MINSURF	MINSURF is a real positive value that defines the minimum surfactant concentration for which the reduction in gas mobility will be calculated. The default value of 1 x 10-20 implies that there is no minimum			
		lb/stb 1×10^{-20}	kg/sm^3 1×10^{-20}	gm/scc 1×10^{-20}	Defined
4	MINSWAT	MINSWAT is a real positive value less than 1.0 that sets the minimum water saturation below which foam has no effect. The default value of 1 x 10-6 implies that there is no minimum. Note that this parameter is only used in the commercial simulator's compositional simulator and is therefore not used by OPM Flow or the commercial simulators black-oil simulator.			
		dimensionless 1×10^{-6}	dimensionless 1×10^{-6}	dimensionless 1×10^{-6}	Defined

No.	Name	Description			Default
		Field	Metric	Laboratory	
Notes: 1) The keyword is followed by NTSFUN tables as declared on the TABDIMS keyword in the RUNSPEC section. 2) Each table must contain just one row and one row only. 3) Each table is terminated by a "/" and there is no "/" terminator for the keyword.					

Table 8.35: FOAMFSC Keyword Description

The gas mobility reduction as a function of surfactant concentration is of the form:

$$F_s = \left(\frac{C_s}{C_s^r} \right)^{e_s} \tag{8.51}$$

Where:

Fs = the resulting gas mobility reduction factor as a function of surfactant concentration,
Cs = surfactant concentration,
Crs = reference surfactant concentration, that is Cs < Csr defines a weak foam and
Cs > Csr defines a strong foam (FOAMCON), and
es = exponent that determines the gradient in the change of the reduction in gas mobility due to foam (FAOAMEXP).

The functional form of the reduction in gas mobility factor (Mrf) is:

$$M_{rf} = \frac{1}{1 + \left(M_r \times F_s \times F_w \times F_o \times F_c \right)} \tag{8.52}$$

Where:

Mr = the reference mobility reduction factor, see the FOAMFRM keyword in the PROPS section,
Fs = gas mobility reduction factor as a function of surfactant concentration, see the FOAMFSC keyword in the PROPS section,
Fw = gas mobility reduction factor as a function of water saturation, see the FOAMFSW keyword in the PROPS section,
Fo = gas mobility reduction factor as a function of oil saturation, see the FOAMFSO keyword in the PROPS section, and
Fc = gas mobility reduction factor as a function of capillary number, see the FOAMFCN keyword in the PROPS section.

See also the FOAM keyword in the RUNSPEC section, the FOAMADS, FOAMMOB, FOAMOPTS and FOAMROCK keywords in the PROPS section.

Example

```
--
--       FOAM GAS MOBILITY VERSUS SURFACTANT CONCENTRATION FUNCTIONS
--
FOAMFSC
--       FOAM       FOAM      FOAM      FOAM
--       FOAMCON    FOAMEXP   MINSURF   MINSWAT
--       -------    -------   -------   -------
           0.001      1.010                        / TABLE NO. 01
           0.002      1.000                        / TABLE NO. 02
                                                   / TABLE NO. 03 (DEFAULTED)
           0.001      0.850  1.0E-10               / TABLE NO. 04
           0.002      1.030                        / TABLE NO. 05
           0.002      1.000                        / TABLE NO. 06
```

Here, NTSFUN equals six on the TABDIMS keyword in the RUNSPEC section and therefore six entries are required for the FOAMFSC keyword. Table number three is completed defaulted and will therefore use all the properties from the previous table, that is table number two.

8.3.63 FOAMFSO – DEFINE FOAM GAS MOBILITY REDUCTION VERSUS OIL SATURATION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The FOAMFSO keyword defines the reduction in gas mobility versus oil saturation, for when the Foam option has been activated by the FOAM keyword in the RUNSPEC section, and the FOAMOPT2 parameter on the FOAMOPTS keyword in the PROPS section has been set to the character string FUNC.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.64 FOAMFST – DEFINE FOAM GAS-WATER SURFACE TENSION VERSUS SURFACTANT CONCENTRATION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The FOAMFST keyword defines the gas-water surface tension versus the foam surfactant concentration, for when the Foam option has been activated by the FOAM keyword in the RUNSPEC section, and the FOAMOPT2 parameter on the FOAMOPTS keyword in the PROPS section has been set to the character string FUNC.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.65 FOAMFSW – DEFINE FOAM GAS MOBILITY REDUCTION VERSUS WATER SATURATION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The FOAMFRM keyword defines the reduction in gas mobility versus water saturation, for when the Foam option has been activated by the FOAM keyword in the RUNSPEC section, and the FOAMOPT2 parameter on the FOAMOPTS keyword in the PROPS section has been set to the character string FUNC.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.66 FOAMMOB - DEFINE FOAM GAS MOBILITY VERSUS FOAM CONCENTRATION TABLES

Description

The FOAMMOB keyword defines the reduction in gas mobility as a function of the foam concentration within a grid block. The Foam option must be activated by the FOAM keyword in the RUNSPEC section in order to use this keyword. In addition, this keyword must be supplied if the foam model is activated.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	FOAMCON	A columnar vector of real monotonically increasing down the column values that defines the foam concentration for the corresponding gas mobility reduction factor (FOAMRATI). The first entry should be zero to define a no foam concentration data set. Units are dependent on the transport phase specified via the FOAMOPT1 variable on the FOAMOPTS keyword in the PROPS section. FOAMOPT1 should be set to either GAS or WATER.			
		Gas: lb/Mscf Water: lb/stb	Gas: kg/sm^3 Water: kg/sm^3	Gas: gm/scc Water: gm/scc	None
2	FOAMRATI	A columnar vector of real decreasing down the column values that defines the corresponding gas mobility reduction factor for a given FOAMCON. The first table data set entry should be one to define a no foam concentration data set. Each FOAMCON/FOAMRATI data set should be terminated by a "/"			
		dimensionless	dimensionless	dimensionless	None

Notes:

1) The keyword is followed by NTPVT tables as declared on the TABDIMS keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NPPVT rows as declared on the TABDIMS keyword in the RUNSPEC section.
3) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.36: FOAMMOB Keyword Description

See also the FOAM keyword in the RUNSPEC section, the FOAMADS, FOAMOPTS and FOAMROCK keywords in the PROPS section.

Example

```
--
--        FOAM GAS MOBILITY VERSUS FOAM CONCENTRATION TABLES
--
FOAMMOB
--        FOAM        FOAM
--        FOAMCON     FOAMRATI
--        -------     --------
            0.000      1.00000
            0.005      0.50000
            0.010      0.20000
            0.015      0.10000
            0.020      0.07500
            0.025      0.07000
            0.030      0.06500
            0.035      0.06500                              / TABLE NO. 01
--        FOAM        FOAM
--        FOAMCON     FOAMRATI
--        -------     --------
            0.000      1.00000
            0.010      0.50000
            0.015      0.25000
            0.020      0.07500
            0.025      0.07000
            0.030      0.07000                              / TABLE NO. 02
```

Given NTPVT equals two and NPPVT is greater and or equal to eight on the TABDIMS keyword in the RUNSPEC section, the example defines the foam gas mobility versus foam concentration tables for two tables.

There is no terminating "/" for this keyword.

8.3.67 FOAMMOBP – DEFINE FOAM MOBILITY REDUCTION VERSUS OIL PRESSURE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The FOAMMOBP keyword defines the reduction in foam mobility reduction versus oil pressure, for when the Foam option has been activated by the FOAM keyword in the RUNSPEC.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.68 FOAMMOBS – DEFINE FOAM MOBILITY REDUCTION VERSUS SHEAR

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The FOAMMOBS keyword defines the reduction in foam mobility reduction versus shear, for when the Foam option has been activated by the FOAM keyword in the RUNSPEC.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.69 FOAMOPTS - DEFINE FOAM MODEL OPTIONS

Description

The FOAMOPTS keyword defines the transport phase for the foam (gas, water or solvent) and how gas mobility reduction should be calculated for when the Foam option has been activated by the FOAM keyword in the RUNSPEC section.

No.	Name	Description	Default
1	FOAMOPT1	A defined character string that specifies the transport phase for the foam, and should be set to one of the following: 1) GAS: for the foam to be transport in the gas phase., 2) WATER: for the foam to be transported in the water phase, or 3) SOLVENT: for the foam to be transported in the solvent phase.	GAS
2	FOAMOPT2	A defined character string that specifies the method to be used to calculate the reduction in gas mobility, and should be set to one of the following: 1) TAB: Sets the reduction in gas mobility is to be calculated based on tables using the FOAMMOB keyword as a function of foam concentration, the FOAMMOBS keyword as a function of shear, or as a function of pressure using the FOAMMOBP keyword. All keywords are in the PROPS section. 2) FUNC: Sets the reduction in gas mobility to be calculated based on a function defined via the FOAMFRM, FOAMFSC, FOAMFSW, FOAMFSO, FOAMFCN, or FOAMFST keywords in the PROPS section. Only the default value of TAB is currently supported by OPM Flow.	TAB
Notes: 1) The keyword is terminated by a "/".			

Table 8.37: FOAMOPTS Keyword Description

Example

```
--
--        FOAMOPT1  FOAMOPT2
--        PHASE     MOBILITY
--        --------  --------
FOAMOPTS
          GAS       TAB                                  / FOAM MODEL OPTIONS
```

The above example defines the transport phase is to be gas and the gas mobility reduction is to use a table as defined by the FOAMMOB keyword as a function of foam concentration, the FOAMMOBS keyword as a function of shear, or as a function of pressure using the FOAMMOBP keyword.

8.3.70 FOAMROCK - DEFINE FOAM ROCK PROPERTIES

Description

The FOAMROCK keyword defines the foam rock properties for when the Foam option has been activated by the FOAM keyword in the RUNSPEC section.

The keyword is recognized by the input deck parser and simulator support is available in the experimental "ebos" simulator.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	ADINDX	A positive integer of 1 or 2 that defines foam desorption option, as per: 1) then foam desorption may occur by retracing the foam adsorption isotherm when the local foam concentration in the solution decreases. 2) then no foam desorption may occur. Only the default value of 1 is supported by OPM Flow.			
		dimensionless 1	dimensionless 1	dimensionless 1	Defined
2	DENSITY	A real value that defines the rock in situ density, that is at reservoir conditions.			
		lb/rb	kg/rm^3	gm/rcc	None

Notes:

1) The keyword is followed by NTSFUN tables as declared on the TABDIMS keyword in the RUNSPEC section.
2) Each table must contain just one row and one row only.
3) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.38: FOAMROCK Keyword Description

Note

In the commercial simulator if the POLYMER and SURFACT phases have been activated in conjunction with the FOAM phase then the mass density of rock will be set by the PLYROCK, SURFROCK, or the FOAMROCK keywords depending on the order entered in the run deck. This is not the case for OPM Flow.

OPM Flow's FOAM phase is a standalone implementation and cannot be used in conjunction with the either the POLYMER or SURFACT phases.

Example

```
--
--       FOAM-ROCK PROPERTIES
--
FOAMROCK
--       DESORP   INSITU
--       OPTN     DENSITY
--       ------   -------
           1      1800.0                                  / TABLE NO. 01
           2      1980.0                                  / TABLE NO. 02
           1      2005.0                                  / TABLE NO. 03
```

The above example defines three foam-rock tables, based on the NTSFUN variable on the TABDIMS keyword in the RUNSPEC section being equal to three.

There is no terminating "/" for this keyword.

8.3.71 FORMFEED – DEFINED THE PRINT FILE FORM-FEED CHARACTER

The FORMFEED keyword defines the form-feed character, or carriage control character, for the output print (*.PRT) run summary (*.RSM) files. The keyword should be place at the very top of the input file.

See *FORMFEED – Defined the Print File Form-Feed Character* in the GLOBAL section for a dull description.

8.3.72 GASDENT – DEFINE GAS DENSITY TEMPERATURE COEFFICIENTS

Description

GASDENT defines the gas density as a function of temperature coefficients for when OPM Flow's thermal option has been activated by the THERMAL keyword in the RUNSPEC. Note this is an OPM Flow keyword used with OPM Flow's black-oil thermal model that is not available in the commercial simulator's black-oil thermal formulation.

This keyword can only be used if OPM Flow's thermal option has been activated by the THERMAL keyword in the RUNSPEC section. Note this is different to the commercial simulator that uses the TEMP keyword in the RUNSPEC section to activate the black-oil thermal model.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	TEMP	TEMP is a real positive value greater than zero that defines the absolute reference temperature used with TEXP1 and TEXP2 to estimate the change in gas density with respect to temperature.			
		°R 527.67	°K 293.15	°K 293.15	Defined
2	TEXP1	TEXP1 is a real positive value greater than zero that defines the gas thermal expansion coefficient of the first order.			
		1/°R 1.67×10^{-4}	1/°K 3.0×10^{-4}	1/°K 3.0×10^{-4}	Defined
3	TEXP2	TEXP2 is a real positive value greater than zero that defines the gas thermal expansion coefficient of the second order.			
		1/°R2 9.26×10^{-7}	1/°K2 3.0×10^{-6}	1/°K2 3.0×10^{-6}	Defined

Notes:

1) The keyword is followed by NTPVT records as declared on the TABDIMS keyword in the RUNSPEC section.
2) Each data set is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.39: GASDENT Keyword Description

The gas density at a given pressure and temperature is calculated from its value at surface conditions and the gas expansion factor (the reciprocal of the gas formation volume factor) as shown in the following equation:

$$\rho_g(p,T) = \rho_g(p_s,T_s)\, b_g(p,T) \tag{8.53}$$

Where the temperature dependence of the gas expansion factor relative to its value at the reference temperature is calculated as shown in the following equation:

$$b_g(p,T)=\frac{b_g(p,T_{ref})}{1+c_1(T-T_{ref})+c_2(T-T_{ref})^2} \tag{8.54}$$

Where:

ρ_g = gas density
b_g = gas expansion factor
p = pressure
T = temperature
c_1, c_2 = thermal expansion coefficients to first and second order
s = subscript indicating surface conditions
ref = subscript indicating reference conditions

Example

The following example shows the GASDENT keyword using the default values, for when the thermal option has been activated by the THERMAL keyword in the RUNSPEC section and for when NTPVT on the TABDIMS keyword in the RUNSPEC section is set to two.

```
--
--       GAS DENSITY TEMPERATURE COEFFICIENTS (OPM FLOW THERMAL KEYWORD)
--
--       GAS         DENSITY   DENSITY
--       TEMP        COEFF1    COEFF2
--       --------    -------   -------
GASDENT
         1*          1*        1*                          / TABLE NO. 01
         1*          1*        1*                          / TABLE NO. 02
```

There is no terminating "/" for this keyword.

8.3.73 GASJT – DEFINE GAS JOULE-THOMSON COEFFICIENT

Description

GASJT activates the gas Joule-Thomson[192] effect in temperature calculations, and defines the gas Joule-Thomson Coefficient ("JTC") at a given reference pressure, for when OPM Flow's thermal option has been activated by the THERMAL keyword in the RUNSPEC.

Note

This is an OPM Flow keyword used with OPM Flow's black-oil thermal model, that is not available in the commercial simulator's black-oil thermal formulation.

This keyword can only be used if OPM Flow's thermal option has been activated by the THERMAL keyword in the RUNSPEC section. Note this is different to the commercial simulator that uses the TEMP keyword in the RUNSPEC section to activate the black-oil thermal model, and does not include the Joule-Thomson effect in temperature calculations.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	PRESS	A real positive value that defines the reference pressure for the corresponding Joule-Thomson Coefficient, GASJTC.			
		psia	barsa	atma	None
2	GASJTC	GASJTC is a real positive or negative value that defines the gas phase Joule-Thomson Coefficient. If the value is defaulted (1*) or set to 0, then GASJTC is internally calculated using the thermal gas density data on the GASDENT keyword in the PROPS section. If a non-zero value is specified, then the GASJTC is assumed to be constant and equal to that value.			
		oF/psia	°C/barsa	°C/atma	0

Notes:

1) The keyword is followed by NTPVT records as declared on the TABDIMS keyword in the RUNSPEC section.
2) Each data set is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.40: GASJT Keyword Description

The Joule–Thomson effect is when a real gas, as oppose to an ideal gas, expands, resulting in the temperature of the gas dropping. During passage of a gas through a choke, the internal energy is transferred to kinetic energy with a corresponding reduction in temperature as the velocity increases. The effect for natural gas is approximately 7 °F for every 100 psi pressure reduction, or 0.5 oC per barsa[193], is valid for "normal" pressures and temperatures at the surface.

[192] *Natural Gas Engineering (McGraw-Hill chemical engineering series), Donald L. Katz, Robert I Lee, McGraw-Hill Education, 1990 (ISBN 0071007776, 9780071007771).*

[193] *https://petrowiki.spe.org/Glossary:Joule-thompson_effect.*

Thermodynamically, the Joule–Thomson coefficient is defined as the isenthalpic[194] change in temperature in a fluid caused by a unitary pressure drop, as shown in the following equation:

$$\eta = \left(\frac{\partial T}{\partial P}\right) \tag{8.55}$$

One can also express enthalpy changes in terms of pressure, temperature and volume changes:

$$\eta = \frac{RT^2}{Pc_p}\left(\frac{\partial Z}{\partial T}\right)_P \tag{8.56}$$

Where:

η	=	Joule–Thomson coefficient,
C_g	=	specific heat at constant pressure,
P	=	pressure,
R	=	gas constant,
T	=	temperature, and
Z	=	gas compressibility factor.

Example

The following example shows the GASJT keyword for when the thermal option has been activated by the THERMAL keyword in the RUNSPEC section, and for when NTPVT on the TABDIMS keyword in the RUNSPEC section is set equal to two.

```
--
--       GAS JOULE-THOMSON COEFFICIENT (OPM FLOW EXTENSION KEYWORD)
--
--       REF         GAS
--       PRESS       JTC
--       --------    -------
GASJT
         20.0        1*                  / TABLE NO. 01
         20.0        0.50                / TABLE NO. 02
```

Here the first entry is defaulted, and the simulator will therefore calculate the gas JTC internally using the data on the GASDENT keyword in the PROPS section.

There is no terminating "/" for this keyword.

[194] *An isenthalpic process or isoenthalpic process, is a process that proceeds without any change in enthalpy, H; or specific enthalpy, h.*

8.3.74 GASVISCT - DEFINE GAS VISCOSITY VERSUS TEMPERATURE FUNCTIONS

Description

GASVISCT defines the gas viscosity as a function of temperature for when OPM Flow's thermal option has been activated by the THERMAL keyword in the RUNSPEC. The reference pressure for this table is given by the VISCREF keyword in the PROPS section. Note this is an OPM Flow keyword used with OPM Flow's black-oil thermal model that is not available in the commercial simulator's black-oil thermal formulation. However, the keyword and similar functionality is available in the commercial compositional simulator.

This keyword can only be used if OPM Flow's thermal option has been activated by the THERMAL keyword in the RUNSPEC section. Note this is different to the commercial simulator that uses the TEMP keyword in the RUNSPEC section to activate the black-oil thermal model.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	TEMP	A columnar vector of real monotonically increasing down the column values that defines the temperature values.			
		°F	°C	°C	None
2	VIS	A columnar vector of real increasing down the column values that defines the gas viscosity for the corresponding temperature values (TEMP). VIS should be given at the reference pressure defined by the PRESS variable on the VISCREF keyword.			
		cP	cP	cP	None

Notes:

1) The keyword is followed by NTPVT tables as declared on the TABDIMS keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NPPVT rows as declared on the TABDIMS keyword in the RUNSPEC section.
3) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.41: GASVISCT Keyword Description

Example

The following example shows the GASVISCT keyword for when the thermal option has been activated by the THERMAL keyword in the RUNSPEC section and for when NTPVT on the TABDIMS keyword in the RUNSPEC section is set equal to one.

```
--
--       GAS VISCOSITY VERSUS TEMPERATURE TABLES (OPM FLOW EXTENSION KEYWORD)
--
--       GAS         GAS
--       TEMP        VISC
--       --------    -------
GASVISCT
           100.0     0.0500
           110.0     0.0550
           120.0     0.0580
           150.0     0.0620
           165.0     0.0625                                  / TABLE NO. 01
```

There is no terminating "/" for this keyword.

8.3.75 GETDATA – LOAD AND ASSIGN DATA ARRAY FROM INIT OR RESTART FILE

The GETDATA keyword loads a data array from a previously generated INIT or RESTART file and assigns the loaded array to either same array in the run or another array name.

See *GETDATA – Load and Assign Data Array from INIT or RESTART Files* in the GRID section for a full description.

8.3.76 GIALL – DEFINE GI VALUES AND PVT PROPERTIES VERSUS PRESSURE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The GIALL keyword defines the GI values and the associated RVGI, RSGI, BGGI and BOGI values as a function of pressure, for when the GI Pseudo Compositional option has been activated in the model via the GIMODEL keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.77 GINODE – DEFINE GI NODE VALUES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The GINODE keyword defines the Gi node values used when the GIMODEL keyword in the RUNSPEC section has been used to activate the GI Pseudo Compositional option for the run. The keyword is used in conjunction with the RSGI, RVGI, BGGI and BOGI keywords in the PROPS section to describe the fluid properties for the GI Pseudo Compositional option.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.78 GRAVCONS – RE-DEFINE GRAVITY CONSTANT

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The GRAVCONS keyword re-defines the gravity constant used in various calculations from the default value used by the simulator. Normally this keyword should not be used.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	GRAVCONS	GRAVCONS is a positive real number number that defines the gravity constant used in various calculations.			
		ft^2psi/lb 0.00694	m^2bars/kg 0.0000981	cm^2atm/gm 0.000968	Defined
Notes: 1) The keyword is terminated by a "/".					

Table 8.42: GRAVCONS Keyword Description

Example

```
--
--       RE-DEFINE GRAVITY CONSTANT
--
GRAVITY
         0.0000980665                                    /
```

The above example re-defines the gravity constant to be 0.0000980665 ft2psi/lb from the default value of 0.00694 ft2psi/lb.

8.3.79 GRAVITY– DEFINE THE SURFACE OIL, WATER GAS GRAVITIES FOR THE FLUIDS

Description

GRAVITY defines the oil API gravity and water and gas surface specific gravities for the fluids for various regions in the model. The number of GRAVITY vector data sets is defined by the NTPVT parameter on the TABDIMS keyword in the RUNSPEC section and the allocation of the GRAVITY data sets to different grid blocks in the model is done via the PVTNUM keyword in the REGION section. One data set consists of one record or line which is terminated by a "/".

This surface density or gravity must be entered using either the DENSITY or GRAVITY keywords irrespective of which phases are active in the model.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	OILAPI	OILAPI is a real number defining the API gravity of the oil phase at surface conditions. The American Petroleum Institute ("API") classifies oils based on an API gravity (γ_{API}), or degrees API (°API), the relationship between relative density (γ_o) of oil and API gravity (γ_{API}) is given by: $$\gamma_{API} = \frac{141.5}{\gamma_o} - 131.5$$			
		°API	°API	°API	None
2	WATGRAV	WATGRAV is a real number defining the specific gravity of the water phase relative to pure water at surface conditions.			
		(water =1.0) 0.7773	(water =1.0) 0.7773	(water =1.0) 0.7773	Defined
3	GASGRAV	GASGRAV is a real number defining the specific gravity of the gas phase relative to air at surface conditions.			
		(air =1.0) 1.000	(air =1.0) 1.000	(air =1.0) 1.000	Defined
Notes: 1) The keyword is followed by NTPVT tables as declared on the TABDIMS keyword in the RUNSPEC section. 2) Each data set is by terminated by a "/" at the end of the line and there is no "/" terminator for the keyword.					

Table 8.43: GRAVITY Keyword Description

According to the SPE SI standard[195], ***Relative Density*** (γ) replaces ***Specific Gravity*** as the term used to define the ratio of the density of a known material to the density of reference material, at standard conditions of pressure and temperature. Standard conditions vary throughout the world, but for oil field units one normally uses14.7 psia and 60 °F, while for SI units some areas use 101.325 kPa and 15 °C.

Note that provided the first table has been entered, subsequent tables may be defaulted, in this case the

[195] *The SI Metric System of Units and SPE Metric Standard, Adopted for Use as a Voluntary Standard by the SPE Board of Directors, June 1983, Society of Petroleum Engineers.*

prior table is copied to the current table. See the third example for an illustration on how to use this feature. See also the DENSITY keyword in the PROPS section.

Examples

The following shows the GRAVITY keyword for when NTPVT on the TABDIMS keyword in the RUNSPEC section is set to one.

```
--
--        OIL      WAT        GAS
--        GRAVITY  GRAVITY    GRAVITY
--        -------  -------    -------
GRAVITY
           39.0     1.012      0.650                / GRAVITY PVT DATA REGION 1
```

The next example shows the GRAVITY keyword for when NTPVT on the TABDIMS keyword in the RUNSPEC section is set to three.

```
--
--        OIL      WAT        GAS
--        GRAVITY  GRAVITY    GRAVITY
--        -------  -------    -------
GRAVITY
           37.0     1.012      0.650                / GRAVITY PVT DATA REGION 1
           38.0     1.012      0.646                / GRAVITY PVT DATA REGION 2
           39.0     1.012      0.640                / GRAVITY PVT DATA REGION 3
```

The third and final example shows the GRAVITY keyword for when NTPVT on the TABDIMS keyword in the RUNSPEC section is set to four. Here table two defaults to table one, and table four defaults to table three.

```
--
--        OIL      WAT        GAS
--        GRAVITY  GRAVITY    GRAVITY
--        -------  -------    -------
GRAVITY
           37.0     1.012      0.650                / GRAVITY PVT DATA REGION 1
                                                    / GRAVITY PVT DATA REGION 2
           38.0     1.012      0.646                / GRAVITY PVT DATA REGION 3
                                                    / GRAVITY PVT DATA REGION 4
```

Again, note that there is no terminating "/" for this keyword.

8.3.80 GSF - GAS SATURATION FUNCTION TABLES (GAS-WATER SYSTEMS)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The GSF keyword defines the gas relative permeability and gas-water capillary pressure data versus gas saturation tables for when only gas and water are present in the input deck. This keyword should only be used if the gas and water phases are present in the run, and can therefore also be used with the CO2STORE and H2STORE models. In addition, the keyword must be used in conjunction with the WSF keyword in the PROPS section, that defines the water relative permeability versus water saturation for gas-water systems.

Note

GSF is a compositional keyword in the commercial compositional simulator, and will therefore cause an error in the commercial black-oil simulator.

Currently, both the GSF and WSF keywords can only be used with the CO2STORE and H2STORE models.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SGAS	A columnar vector of real monotonically increasing down the column values starting from zero and terminating at one, that defines the gas saturation.			
		dimensionless	dimensionless	dimensionless	None
2	KRG	A columnar vector of real values that are either equal or increasing down the column and that are greater than or equal to zero and less than or equal to one that defines the gas relative permeability. Note that the first entry in the column must be zero.			
		dimensionless	dimensionless	dimensionless	None
3	PCWG	A columnar vector of real values that are either equal or increasing down the column that defines the gas-water capillary pressure.			
		psia	bars	atm	None

Notes:

1) The keyword is followed by NTSFUN tables as declared on the TABDIMS keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NSSFUN rows as declared on the TABDIMS keyword in the RUNSPEC section.
3) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.44: GSF Keyword Description

See also the *WSF - Water Saturation Tables versus Water Saturation (Gas-Water and CO2STORE Systems)* keyword in the PROPS section, that defines the water saturation as a function of water saturation for when only the gas and water phases are present in the model.

Example

```
--
--        GAS RELATIVE PERMEABILITY TABLES (OPM FLOW KEYWORD)
--
GSF
--        SGAS        KRG         PCGW
--        FRAC                    PSIA
--        --------  --------    -------
              0.00    0.0000      0.0
              0.05    0.0000      0.0
              0.10    0.0000      0.0
              0.15    0.0000      0.0
              0.20    0.0002      0.0
              0.25    0.0010      0.0
              0.30    0.0062      0.0
              0.35    0.0140      0.0
              0.40    0.0273      0.0
              0.45    0.0450      0.0
              0.50    0.0707      0.0
              0.55    0.1020      0.0
              0.60    0.1412      0.0
              0.65    0.1870      0.0
              0.70    0.2412      0.0
              0.77    0.3288      0.0
              0.82    0.4000      0.0
              0.85    0.4450      0.0                        / TABLE NO. 01
--        SGAS        KRG         PCGW
--        FRAC                    PSIA
--        --------  --------    -------
              0.00    0.0000      0.0
              0.05    0.0000      0.0
              0.10    0.0000      0.0
              0.15    0.0000      0.0
              0.20    0.0002      0.0
              0.25    0.0010      0.0
              0.30    0.0062      0.0
              0.35    0.0140      0.0
              0.40    0.0273      0.0
              0.45    0.0450      0.0
              0.50    0.0707      0.0
              0.55    0.1020      0.0
              0.60    0.1412      0.0
              0.65    0.1870      0.0
              0.70    0.2412      0.0
              0.77    0.3288      0.0
              0.82    0.4000      0.0
              0.85    0.4450      0.0                        / TABLE NO. 02
```

The example defines two GSF tables for when gas and water are present in the input deck. In the tables the gas-water capillary pressure data has been set to zero.

8.3.81 HA – HISTORY MATCH END-POINT GRADIENT ADDITIVE MODIFIER

Description

The HA series of keywords defines the history match end-point gradient parameters used to set the additive cumulative end point data, for when the History Match Gradient option has been activated by the HMDIMS keyword in the RUNSPEC section. In addition, the End-Point Scaling option must also be activated by the ENDSCALE keyword which is also in the RUNSPEC section. The keyword consists of the first two characters of "HA" followed by the end-point keyword shown in Table 8.45, for example, HASWL.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

Type	End-Point Keyword	Oil-Water End-Point Definitions
Saturation	SWL	Connate water saturation, that is the smallest water saturation in a water saturation function table.
	SWCR	Critical water saturation, that is the largest water saturation for which the water relative permeability is zero.
	SOWCR	Critical oil-in-water saturation, that is the largest oil saturation for which the oil relative permeability is zero in an oil-water system.
Relative Permeability	KRW	Relative permeability of water at the maximum water saturation (normally the maximum water saturation is one).
	KRO	Relative permeability of oil at the maximum oil saturation.
	KRWR	Relative permeability of water at the residual oil saturation or the residual gas saturation in a gas-water run.
	KRORW	Relative permeability of oil at the critical water saturation.
Capillary Pressure	SWLPC	Capillary pressure connate water saturation, that is the smallest water saturation in a water saturation function table.
Type	**End-Point Keyword**	**Gas-Oil End-Point Definitions**
Saturation	SGL	Connate gas saturation, that is the smallest gas saturation in a gas saturation function table.
	SGCR	Critical gas saturation, that is the largest gas saturation for which the gas relative permeability is zero.
	SOGCR	Critical oil-in-gas saturation, that is the largest oil saturation for which the oil relative permeability is zero in an oil-gas-connate water system.
Relative Permeability	KRG	Relative permeability of gas at the maximum gas saturation.
	KRGR	Relative permeability of gas at the residual oil saturation or the critical water saturation in a gas-water run.
	KRORG	Relative permeability of oil at the critical gas saturation.

Type	End-Point Keyword	Oil-Water End-Point Definitions
Capillary Pressure	SGLPC	Capillary pressure connate gas saturation, that is the smallest gas saturation in a gas saturation function table.

Table 8.45: HA Keyword List

See also the HMPROPS keyword in the PROPS section that allows the use of the ADD, BOX, EQUALS, COPY, MINVALUE, and MAXVALUE keywords to be used with the HA and HM series of keywords.

8.3.82 HDISP – DEFINE TRACER MECHANICAL DISPERSIVITY PARAMETERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The HDISP keyword is combined with three character tracer name, specified by the TRACER keyword in the PROPS section, to define the tracer's mechanical dispersivity parameters.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.83 HM – HISTORY MATCH END-POINT GRADIENT MULTIPLICATIVE MODIFIER

Description

The HM series of keywords defines the history match end-point gradient parameters used to set the multiplicative cumulative end point data, for when the History Match Gradient option has been activated by the HMDIMS keyword in the RUNSPEC section. In addition, the End-Point Scaling option must also be activated by the ENDSCALE keyword which is also in the RUNSPEC section. The keyword consists of the first two characters of "HM" followed by the end-point keyword shown in Table 8.46, for example, HMSWL.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

Type	End-Point Keyword	Oil-Water End-Point Definitions
Saturation	SWL	Connate water saturation, that is the smallest water saturation in a water saturation function table.
	SWCR	Critical water saturation, that is the largest water saturation for which the water relative permeability is zero.
	SOWCR	Critical oil-in-water saturation, that is the largest oil saturation for which the oil relative permeability is zero in an oil-water system.
Relative Permeability	KRW	Relative permeability of water at the maximum water saturation (normally the maximum water saturation is one).
	KRO	Relative permeability of oil at the maximum oil saturation.
	KRWR	Relative permeability of water at the residual oil saturation or the residual gas saturation in a gas-water run.
	KRORW	Relative permeability of oil at the critical water saturation.
Capillary Pressure	SWLPC	Capillary pressure connate water saturation, that is the smallest water saturation in a water saturation function table.
Type	**End-Point Keyword**	**Gas-Oil End-Point Definitions**
Saturation	SGL	Connate gas saturation, that is the smallest gas saturation in a gas saturation function table.
	SGCR	Critical gas saturation, that is the largest gas saturation for which the gas relative permeability is zero.
	SOGCR	Critical oil-in-gas saturation, that is the largest oil saturation for which the oil relative permeability is zero in an oil-gas-connate water system.
Relative Permeability	KRG	Relative permeability of gas at the maximum gas saturation.
	KRGR	Relative permeability of gas at the residual oil saturation or the critical water saturation in a gas-water run.
	KRORG	Relative permeability of oil at the critical gas saturation.

Type	End-Point Keyword	Oil-Water End-Point Definitions
Capillary Pressure	SGLPC	Capillary pressure connate gas saturation, that is the smallest gas saturation in a gas saturation function table.

Table 8.46: HM Keyword List

See also the HMPROPS keyword in the PROPS section that allows the use of the ADD, BOX, EQUALS, COPY, MINVALUE, and MAXVALUE keywords to be used with the HA and HM series of keywords.

8.3.84 HMMROCK – HISTORY MATCH ROCK COMPRESSIBILITY GRADIENT CUMULATIVE MULTIPLIERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

HMMROCK defines the rock compressibility gradient cumulative multipliers to be applied to the rock compressibility as defined by the ROCK keyword in the PROPS section, for when the History Match Gradient option has been activated by the HMDIMS keyword in the RUNSPEC section. The constant should be a real number.

The allocation of the ROCK tables to different grid blocks in the model is done via the PVTNUM or the SATNUM keywords in the REGION section.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

8.3.85 HMMROCKT – HISTORY MATCH ROCK COMPACTION GRADIENT CUMULATIVE MULTIPLIERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

HMMROCKT defines the rock compaction gradient cumulative multipliers to be applied to the compaction data entered by the ROCTAB or ROCKTABH keywords in the PRROPS section, for when the History Match Gradient option has been activated by the HMDIMS keyword in the RUNSPEC section

The ROCKTAB keyword defines the rock compaction attributes to be applied for when the rock compaction option has been invoked by the ROCKCOMP keyword in the RUNSPEC section. ROCKTAB defines pore volume and transmissibility multipliers versus pressure that are used in the compaction calculations. If the RKTRMDIR has been activated in the PROPS section, then the transmissibility multiplier is directional dependent

This keyword should only be used if compaction option has been enabled.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

8.3.86 HMPROPS – HISTORY MATCH END-POINT SECTION START

Description

HMPROPS defines the start of a history match end-points section, for when the History Match Gradient option has been activated by the HMDIMS keyword in the RUNSPEC section. In addition, the End-Point Scaling option must also be activated by the ENDSCALE keyword which is also in the RUNSPEC section. The keyword allows for the BOX, EQUALS, COPY, MINVALUE, MAXVALUE and ADD keywords to be used with the HA and HM series of keywords that reference the end-point scaling arrays, that is: HMKRG, HMKRGR, HMKRO, HMKRORG, HMKRORW, HMKRW, HMKRWR, HMPCW, HMPCG, HMSGCR, HMSOWCR, HMSOGCR, HMSWCR, and HMSWL keywords.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.87 HMROCK – HISTORY MATCH ROCK COMPRESSIBILITY GRADIENT PARAMETERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The HMROCK keyword defines the history match rock compressibility gradient parameters for when the History Match Gradient option has been activated by the HMDIMS keyword in the RUNSPEC section

See also the HMDIMS keyword in the RUNSPEC section that specifies the dimensions for the gradient option, including the maximum number of rock gradient parameters that can be used with the History Match Gradient option.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.88 HMROCKT – HISTORY MATCH ROCK COMPACTION GRADIENT PARAMETERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The HMROCKT keyword defines the history match rock compaction gradient parameters for when the History Match Gradient option has been activated by the HMDIMS keyword in the RUNSPEC section, and the history match rock compaction data has been entered via the HMMROCKT, ROCKTAB and ROCKTABH keywords in the PROPS section.

See also the HMDIMS keyword in the RUNSPEC section that specifies the dimensions for the gradient option, including the maximum number of rock gradient parameters that can be used with the History Match Gradient option.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.89 HMRREF – HISTORY MATCH ROCK TABLE REFERENCE PRESSURE VALUES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The HMRREF keyword defines the history match rock compaction reference pressure gradient values to be used in conjunction with HMMROCKT, ROCKTAB and ROCKTABH keywords in the PROPS section, for when the History Match Gradient option has been activated by the HMDIMS keyword in the RUNSPEC section. The history match rock compaction data is entered via the HMMROCKT, ROCKTAB and ROCKTABH keywords in the PROPS section.

See also the HMDIMS keyword in the RUNSPEC section that specifies the dimensions for the gradient option, including the maximum number of rock gradient parameters that can be used with the History Match Gradient option.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.90 HWKRO – END-POINT SCALING OF GRID CELL KRO(SWL) (HIGH SALINITY AND WATER WET)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

HWKRO defines the scaling parameter for the oil relative permeability value at the connate water saturation (SWL), for all the cells in the model via an array, for when the Low Salt and Surfactant Wettability options have been selected. The data is used to scale the oil relative permeability in the high salinity water wet oil relative permeability saturation tables. The ENDSCALE keyword in the RUNSPEC section should be activated to enable end-point scaling and the use of this keyword. In addition the Low Salt option should be activated by the LOWSALT keyword in the RUNSPEC section and the Surfactant Wettability option activated by the SURFACT or SURFACTW keywords, which are also in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.91 HWKRORG – END-POINT SCALING OF GRID CELL KRO(SGCR) (HIGH SALINITY AND WATER WET)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

HWKRORG defines the scaling parameter for the relative permeability of oil at the critical gas saturation (SGCR), for all the cells in the model via an array, for when the Low Salt and Surfactant Wettability options have been selected. The data is used to scale the oil relative permeability in the high salinity water wet oil relative permeability saturation tables. The ENDSCALE keyword in the RUNSPEC section should be activated to enable end-point scaling and the use of this keyword. In addition the Low Salt option should be activated by the LOWSALT keyword in the RUNSPEC section and the Surfactant Wettability option activated by the SURFACT or SURFACTW keywords, which are also in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.92 HWKRORW – END-POINT SCALING OF GRID CELL KRO(SWCR) (HIGH SALINITY AND WATER WET)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

HWKRORW defines the scaling parameter for the relative permeability of oil at the critical water saturation (SWCR), for all the cells in the model via an array, for when the Low Salt and Surfactant Wettability options have been selected. The data is used to scale the oil relative permeability in the high salinity water wet oil relative permeability saturation tables. The ENDSCALE keyword in the RUNSPEC section should be activated to enable end-point scaling and the use of this keyword. In addition the Low Salt option should be activated by the LOWSALT keyword in the RUNSPEC section and the Surfactant Wettability option activated by the SURFACT or SURFACTW keywords, which are also in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.93 HWKRW – END-POINT SCALING OF GRID CELL KRW(SW =1.0) (HIGH SALINITY AND WATER WET)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

HWKRW defines the scaling parameter at the maximum oil relative permeability value (SWU), that is for Sw = 1.0, for all the cells in the model via an array, for when the Low Salt and Surfactant Wettability options have been selected. The data is used to scale the water relative permeability in the high salinity water wet water relative permeability saturation tables. The ENDSCALE keyword in the RUNSPEC section should be activated to enable end-point scaling and the use of this keyword. In addition the Low Salt option should be activated by the LOWSALT keyword in the RUNSPEC section and the Surfactant Wettability option activated by the SURFACT or SURFACTW keywords, which are also in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.94 HWKRWR – END-POINT SCALING OF GRID CELL KRWR(SW=1.0) (HIGH SALINITY AND WATER WET)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

HWKRWR defines the scaling parameter at the maximum oil relative permeability value (SWU), that is for Sw = 1.0, for all the cells in the model via an array, for when the Low Salt and Surfactant Wettability options have been selected. The data is used to scale the water relative permeability in the high salinity water wet water relative permeability saturation tables. The ENDSCALE keyword in the RUNSPEC section should be activated to enable end-point scaling and the use of this keyword. In addition the Low Salt option should be activated by the LOWSALT keyword in the RUNSPEC section and the Surfactant Wettability option activated by the SURFACT or SURFACTW keywords, which are also in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.95 HWPCW – END-POINT SCALING OF GRID CELL WATER CAPILLARY PRESSURE (HIGH SALINITY AND WATER WET)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

HWPCW defines the maximum water-oil pressure values for all the cells in the model via an array, for when the Low Salt and Surfactant Wettability options have been selected. The ENDSCALE keyword in the RUNSPEC section should be activated to enable end-point scaling and the use of this keyword. In addition the Low Salt option should be activated by the LOWSALT keyword in the RUNSPEC section and the Surfactant Wettability option activated by the SURFACT or SURFACTW keywords, which are also in the RUNSPEC section. The keyword re-scales the oil-water capillary pressure in the high salinity water wet capillary saturation tables from a cell's assigned saturation function by the grid block's HWPCW value.

The capillary pressure for a grid block is scaled by:

$$P_c = P_{c_{TABLE}} \left(\frac{HWPCW}{P_{c_{TABLE-MAX}}} \right) \quad (8.57)$$

Where:

P_c	=	the resulting high salinity water wet water capillary pressure for a grid cell.
HWPCW	=	the maximum capillary pressure from the HWPCW array for a given cell.
$P_{c_{TABLE}}$	=	the capillary pressure in the high salinity water wet water capillary pressure table allocated to the grid block.
$P_{c_{TABLE-MAX}}$	=	the maximum capillary pressure in the high salinity water wet water capillary pressure table allocated to the grid block (that is at the connate water saturation).

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.96 HWSOGCR – END-POINT SCALING GRID CELL SOGCR (HIGH SALINITY AND WATER WET)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

HWSOGCR defines the critical oil saturation with respect to gas ("SOGCR"), for all the cells in the model via an array, for when the Low Salt and Surfactant Wettability options have been selected. The data is used to scale the oil saturation in the high salinity water wet oil-gas relative permeability saturation tables. The ENDSCALE keyword in the RUNSPEC section should be activated to enable end-point scaling and the use of this keyword. In addition, the Low Salt option should be activated by the LOWSALT keyword in the RUNSPEC section and the Surfactant Wettability option activated by the SURFACT or SURFACTW keywords, which are also in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.97 HWSOWCR – END-POINT SCALING GRID CELL SOWCR (HIGH SALINITY AND WATER WET)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

HWSOWCR defines the critical oil saturation with respect to water ("SOWCR"), for all the cells in the model via an array, for when the Low Salt and Surfactant Wettability options have been selected. The data is used to scale the oil saturation in the high salinity water wet water-oil relative permeability saturation tables. The ENDSCALE keyword in the RUNSPEC section should be activated to enable end-point scaling and the use of this keyword. In addition, the Low Salt option should be activated by the LOWSALT keyword in the RUNSPEC section and the Surfactant Wettability option activated by the SURFACT or SURFACTW keywords, which are also in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.98 HWSWCR – END-POINT SCALING GRID CELL SWCR (HIGH SALINITY AND WATER WET)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

HWSWCR defines the critical water saturation ("SWCR"), for all the cells in the model via an array, for when the Low Salt and Surfactant Wettability options have been selected. The data is used to scale the water saturation in the high salinity water wet water-oil relative permeability saturation tables. The ENDSCALE keyword in the RUNSPEC section should be activated to enable end-point scaling and the use of this keyword. In addition, the Low Salt option should be activated by the LOWSALT keyword in the RUNSPEC section and the Surfactant Wettability option activated by the SURFACT or SURFACTW keywords, which are also in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.99 HWSWL – END-POINT SCALING GRID CELL SWL (HIGH SALINITY AND WATER WET)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

HWSWL defines the connate water saturation ("SWL"), for all the cells in the model via an array, for when the Low Salt and Surfactant Wettability options have been selected. The data is used to scale the water saturation in the high salinity water wet water-oil relative permeability saturation tables, as well as the associated capillary pressure tables. The ENDSCALE keyword in the RUNSPEC section should be activated to enable end-point scaling and the use of this keyword. In addition, the Low Salt option should be activated by the LOWSALT keyword in the RUNSPEC section and the Surfactant Wettability option activated by the SURFACT or SURFACTW keywords, which are also in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.100 HWSWLPC – END-POINT SCALING GRID CELL SWLPC (HIGH SALINITY AND WATER WET)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

HWSWLPC defines the capillary pressure connate water saturation ("SWLPC"), for all the cells in the model via an array, for when the Low Salt and Surfactant Wettability options have been selected. The data is used to scale the water saturation in the high salinity water wet water-oil capillary pressure tables. The ENDSCALE keyword in the RUNSPEC section should be activated to enable end-point scaling and the use of this keyword. In addition, the Low Salt option should be activated by the LOWSALT keyword in the RUNSPEC section and the Surfactant Wettability option activated by the SURFACT or SURFACTW keywords, which are also in the RUNSPEC section.

Note the keyword only applies the scaling to the capillary pressures tables, unlike the HWSWL keyword that applies the scaling to both the capillary pressure and relative permeability tables.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.101 HWSWU – END-POINT SCALING GRID CELL SWU (HIGH SALINITY AND WATER WET)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

HWSWU defines the maximum water saturation("SWU"), for all the cells in the model via an array, for when the Low Salt and Surfactant Wettability options have been selected. The data is used to scale the water saturation in the high salinity water wet water-oil relative permeability saturation tables, as well as the associated capillary pressure tables. The ENDSCALE keyword in the RUNSPEC section should be activated to enable end-point scaling and the use of this keyword. In addition, the Low Salt option should be activated by the LOWSALT keyword in the RUNSPEC section and the Surfactant Wettability option activated by the SURFACT or SURFACTW keywords, which are also in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.102 HYDRHEAD – DEFINE HYDRAULIC HEAD OUTPUT REFERENCE DATA

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The HYDRHEAD keyword defines the hydraulic head reference data for when the hydraulic head information is requested to be written out via one on the SUMMARY keywords (BHD, BHDF, etc.) in the SUMMARY section, or to the RESTART file via the HYDH or HYDHFW variables on the RESTART keyword

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.103 HYMOBGDR – ACTIVATE CARLSON AND KILLOUGH ALTERNATIVE DRAINAGE HYSTERESIS OPTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, HYMOBGDR, activates the Carlson and Killough alternative secondary drainage hysteresis option for when the hysteresis option has been activated by the HYSTER variable on the SATOPTS keyword in the RUNSPEC section, and either the Carlson[196] or Killough[197] models have been selected via the EHYSTR keyword in the PROPS section. Due to numerical accuracy, the gas saturation may fall below the critical gas saturation (SGCR), that is the largest gas saturation for which the gas relative permeability is zero, and gas would therefore be immobile until the gas saturation increases above SGCR. This option overcomes this situation by letting the gas become mobile once it starts increasing, effectively setting the SGCR to the current gas saturation.

There is no data required for this keyword and there is no terminating "/" for this keyword.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

Example

```
--
--       ACTIVATE CARLSON AND KILLOUGH ALTERNATIVE DRAINAGE HYSTERESIS OPTION
--
HYMOBGDR
```

[196] *Carlson, F. M. "Simulation of Relative Permeability Hysteresis to the Non-Wetting Phase," paper SPE 10157, presented at the SPE Annual Technical Conference & Exhibition, San Antonio, Texas, USA (October 5-7, 1981).*

[197] *Killough, J. E. "Reservoir Simulation with History-dependent Saturation Functions," paper SPE 5106, Society of Petroleum Engineers Journal (1976) 16, No. 1, 37-48.*

8.3.104 HYSTCHCK - ACTIVATE HYSTERESIS IMBIBITION AND DRAINAGE END-POINT VALIDATION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The HYSTCHCK keyword activate the hysteresis imbibition and drainage end-point check to validate that the two sets of end-points are consistent, for when the Hysteresis option has been activated by the HYSTER variable on the SATOPTS keyword in the RUNSPEC section, and the ENDSCALE keyword in the RUNSPEC section has been activated to enable end-point scaling.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.105 IKRG – END-POINT SCALING OF GRID CELL KRG(SGU) (IMBIBITION)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

IKRG defines the imbibition scaling parameter at the maximum gas relative permeability value (ISGU), normally ISGU is equal to 1.0 - Swc, for all the cells in the model via an array. The ENDSCALE keyword in the RUNSPEC section should be activated to enable end-point scaling and the use of this keyword. In addition, the HYSTER option on the SATOPTS keyword in the RUNSPEC section has to be activated to invoke the Hysteresis option. The SCALCERS keyword in the PROPS section defines the options used in the re-scaling process, the options are two point scaling and three point scaling.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	IKRG	IKRG is an array of positive real numbers which are greater than zero and less than or equal to 1.0, that are the assigned scaling IKRG imbibition values for each cell in the model. Repeat counts may be used, for example 50*0.400. dimensionless			Taken from cell allocated relative permeability table.
		dimensionless	dimensionless	dimensionless	

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) The keyword is terminated by a "/".
3) Note this the directional independent version of the keyword used with the end-point scaling option. If directional end-point scaling has been activated then the IKRGX±, IKRGY± and IKRGZ± series of the keywords should be used.

Table 8.47: IKRG Keyword Description

For the two point scaling option and for the IKRGR gas relative permeability array NOT present in the input deck the krg value for a grid block is scaled by:

$$k_{rg} = k_{rg\,TABLE}\left(\frac{IKRG}{k_{rg\,TABLE-MAX}}\right) \tag{8.58}$$

Where:

krg = the resulting krg value for a grid cell.
IKRG = the scaling gas relative permeability value from the IKRG array for a given cell.
$k_{rg\,TABLE}$ = the gas relative permeability from a grid block's gas-oil table at the grid blocks gas saturation.
$k_{rg\,TABLE-MAX}$ = the maximum gas relative permeability from a grid block's gas-oil table, that is at the connate water saturation (Swc).

If the IKRGR keyword is present in the input deck then the scaling matches the imbibition relative permeability at the critical saturation of the displacing phase.

If three point scaling option has been selected via the SCALECRS keyword in the PROPS section the critical displacing phase is defined as:

No	Phases Present	Critical Saturation
1	Gas-Oil	$S_{critical}$ = 1.0 – ISOGCR - ISWL
2	Gas-Oil-Water	$S_{critical}$ = 1.0 – ISOGCR - ISWL
3	Gas-Water	$S_{critical}$ = 1.0 – ISWCR

Table 8.48: Critical Displacement Relationships

End-point scaling allows the entered relative permeability functions to be re-scaled based on the saturation end-points allocated to each cell by the ISWL, ISWCR, ISWU, ISGL, ISGCR, ISGU, ISOWCR, and ISOGCR saturation grid arrays for the saturation end-points, In addition end-point scaling may be directional dependent in which case the directional dependent versions of the aforementioned arrays should be used, that is ISWUX, ISWUY and ISWUZ instead of ISWU, There is also the facility to make the directional end-point scaling reversible or non-reversible and if the non-reversible option is selected the non-reversible versions of the aforementioned arrays should be used, that is ISWUX, ISWUX-, ISWUY, ISWUY-, ISWUZ and ISWUZ-, instead of the ISWU keyword.

End-point scaling also allows the entered relative permeability functions to be scale on the relative permeability values using the IKRG, IKRGR, IKRO, IKRORG, IKRORW, IKRW and IKRWR relative permeability grid cell arrays for the relative permeability end-point data. In addition end-point scaling may be directional dependent in which case the directional dependent versions of the aforementioned arrays should be used, that is IKRGX, IKRGY and IKRGZ instead of IKRG, There is also the facility to make the directional end-point scaling reversible or non-reversible and if the non-reversible option is selected the non-reversible versions of the aforementioned arrays should be used, that is IKRGX, IKRGX-, IKRGY, IKRGY-,I KRGZ and IKRGZ-, instead of the IKRG keyword.

Example

The example below defines an input box for the whole grid and for layers one to three, for layer one IKRG is set equal to 0.550, for layer two IKRG equals 0.575, and for layer three IKRG equals 0.600.

```
--
--       DEFINE INPUT BOX FOR EDITING INPUT ARRAYS (NX=100, NY=100)
--
--       ---------- BOX ---------
--       I1  I2   J1  J2   K1  K2
BOX
         1*  1*   1*  1*   1   3                          / DEFINE BOX AREA
--
--       SET IKRG VALUES FOR THREE LAYERS IN THE MODEL
--
IKRG
         1000*0.550  1000*0.575  1000*0.600               /
--
--       DEFINE END OF INPUT BOX EDITING OF INPUT ARRAYS
--
ENDBOX
```

8.3.106 IKRGR – END-POINT SCALING OF GRID CELL KRGR(1-SOGCR) (IMBIBITION)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

IKRGR defines the imbibition scaling parameter at the relative permeability of gas at residual oil saturation (1 – ISOGCR), or critical water saturation in a gas-water run (Swc), for all the cells in the model via an array. The ENDSCALE keyword in the RUNSPEC section should be activated to enable end-point scaling and the use of this keyword. In addition, the HYSTER option on the SATOPTS keyword in the RUNSPEC section has to be activated to invoke the Hysteresis option. The SCALCERS keyword in the PROPS section defines the options used in the re-scaling process, the options are two point scaling and three point scaling.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	IKRGR	IKRGR is an array of positive real numbers which are greater than zero and less than or equal to 1.0, that are the assigned imbibition scaling IKRGR values for each cell in the model. In addition, for a given grid block IKGRGT should be less than IKRG. Repeat counts may be used, for example 50*0.400.			Taken from cell allocated relative permeability table.
		dimensionless	dimensionless	dimensionless	

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) The keyword is terminated by a "/".
3) Note this the directional independent version of the keyword used with the end-point scaling option. If directional end-point scaling has been activated then the IKRGRX± , IKRGRY± and IKRGRZ± series of the keywords should be used

Table 8.49: IKRGR Keyword Description

When the IKRGR keyword is present in the input deck then the scaling matches the imbibition relative permeability at the critical saturation of the displacing phase (oil or water).

If three point scaling option has been selected via the SCALECRS keyword in the PROPS section the critical displacing phase is defined as:

No	Phases Present	Critical Saturation
1	Gas-Oil	$S_{critical}$ = 1.0 – ISOGCR - ISWL
2	Gas-Oil-Water	$S_{critical}$ = 1.0 – ISOGCR - ISWL
3	Gas-Water	$S_{critical}$ = 1.0 – ISWCR

Table 8.50: Critical Displacement Relationships

End-point scaling allows the entered relative permeability functions to be re-scaled based on the saturation end-points allocated to each cell by the ISWL, ISWCR, ISWU, ISGL, ISGCR, ISGU, ISOWCR, and ISOGCR saturation grid arrays for the saturation end-points, In addition end-point scaling may be directional dependent in which case the directional dependent versions of the aforementioned arrays should be used,

that is ISWUX, ISWUY and ISWUZ instead of ISWU, There is also the facility to make the directional end-point scaling reversible or non-reversible and if the non-reversible option is selected the non-reversible versions of the aforementioned arrays should be used, that is ISWUX, ISWUX-, ISWUY, ISWUY-, ISWUZ and ISWUZ-, instead of the ISWU keyword.

End-point scaling also allows the entered relative permeability functions to be scale on the relative permeability values using the IKRG, IKRGR, IKRO, IKRORG, IKRORW, IKRW and IKRWR relative permeability grid cell arrays for the relative permeability end-point data. In addition end-point scaling may be directional dependent in which case the directional dependent versions of the aforementioned arrays should be used, that is IKRGX, IKRGY and IKRGZ instead of IKRG, There is also the facility to make the directional end-point scaling reversible or non-reversible and if the non-reversible option is selected the non-reversible versions of the aforementioned arrays should be used, that is IKRGX, IKRGX-, IKRGY, IKRGY-,I KRGZ and IKRGZ-, instead of the IKRG keyword.

Examples

The first example defines an input box for the whole grid and for layers one to three, for layer one IKRRG is set equal to 0.500, for layer two IKRGR equals 0.570, and for layer three IKRGR equals 0.580.

```
--
--       DEFINE INPUT BOX FOR EDITING INPUT ARRAYS (NX=100, NY=100)
--
--       ---------- BOX ---------
--       I1  I2   J1  J2   K1  K2
BOX
         1*  1*   1*  1*   1   3                          / DEFINE BOX AREA
--
--       SET IKRGR VALUES FOR THREE LAYERS IN THE MODEL
--
IKRGR
         10000*0.500  10000*0.570  10000*0.580               /
--
--       DEFINE END OF INPUT BOX EDITING OF INPUT ARRAYS
--
ENDBOX
```

The next example does exactly the same thing using the EQUALS keyword instead.

```
--
--       ARRAY       CONSTANT     ---------- BOX ---------
--                                I1  I2   J1  J2   K1  K2
EQUALS
         IKRGR'      0.5000       1*  1*   1*  1*    1   1  / IKRGR FOR LAYER 1
         IKRGR'      0.5700       1*  1*   1*  1*    2   2  / IKRGR FOR LAYER 2
         IKRGR'      0.5800       1*  1*   1*  1*    3   3  / IKRGR FOR LAYER 3
/
```

8.3.107 IKRO – END-POINT SCALING OF GRID CELL KRO(SWL) (IMBIBITION)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

IKRO defines the scaling parameter for the imbibition oil relative permeability value at the connate water saturation (ISWL), for all the cells in the model via an array. The ENDSCALE keyword in the RUNSPEC section should be activated to enable end-point scaling and the use of this keyword. In addition, the HYSTER option on the SATOPTS keyword in the RUNSPEC section has to be activated to invoke the Hysteresis option. The SCALCERS keyword in the PROPS section defines the options used in the re-scaling process, the options are two point scaling and three point scaling.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	IKRO	IKRO is an array of positive real numbers which are greater than zero and less than or equal to 1.0, that are the assigned imbibition scaling IKRO values for each cell in the model. Repeat counts may be used, for example 50*0.500.			Taken from cell allocated relative permeability table.
		dimensionless	dimensionless	dimensionless	

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) The keyword is terminated by a "/".
3) Note this the directional independent version of the keyword used with the end-point scaling option. If directional end-point scaling has been activated then the IKROX±, IKROY± and IKROZ± series of the keywords should be used.

Table 8.51: IKRO Keyword Description

For the two point scaling option and for the IKRORW or IKRORG oil imbibition relative permeability arrays NOT being present in the input deck the kro value for a grid block is scaled by:

$$k_{ro} = k_{ro\,TABLE}\left(\frac{IKRO}{k_{ro\,TABLE-MAX}}\right) \tag{8.59}$$

Where:

kro = the resulting kro value for a grid cell.
$IKRO$ = the scaling oil relative permeability value from the IKRO array for a given cell.
$k_{ro\,TABLE}$ = the oil relative permeability from a grid block's oil relative permeability table at the grid blocks oil saturation.
$k_{ro\,TABLE-MAX}$ = the maximum oil relative permeability from a grid block's oil relative table, that is at the critical water saturation (Swcr).

If the IKRORW or IKRORG keywords are present in the input deck then the scaling matches the relative permeability at the critical saturation of the displacing phase.

If three point scaling option has been selected via the SCALECRS keyword in the PROPS section the critical displacing phase is defined as:

No	Keywords Present	Critical Saturation
1	KRORW	$S_{critical}$ = 1.0 – SWCR - SGL
2	KRORG	$S_{critical}$ = 1.0 – SGCR - SWL

Table 8.52: Critical Displacement Relationships

End-point scaling allows the entered relative permeability functions to be re-scaled based on the saturation end-points allocated to each cell by the ISWL, ISWCR, ISWU, ISGL, ISGCR, ISGU, ISOWCR, and ISOGCR saturation grid arrays for the saturation end-points, In addition end-point scaling may be directional dependent in which case the directional dependent versions of the aforementioned arrays should be used, that is ISWUX, ISWUY and ISWUZ instead of ISWU, There is also the facility to make the directional end-point scaling reversible or non-reversible and if the non-reversible option is selected the non-reversible versions of the aforementioned arrays should be used, that is ISWUX, ISWUX-, ISWUY, ISWUY-, ISWUZ and ISWUZ-, instead of the ISWU keyword.

End-point scaling also allows the entered relative permeability functions to be scale on the relative permeability values using the IKRG, IKRGR, IKRO, IKRORG, IKRORW, IKRW and IKRWR relative permeability grid cell arrays for the relative permeability end-point data. In addition end-point scaling may be directional dependent in which case the directional dependent versions of the aforementioned arrays should be used, that is IKRGX, IKRGY and IKRGZ instead of IKRG, There is also the facility to make the directional end-point scaling reversible or non-reversible and if the non-reversible option is selected the non-reversible versions of the aforementioned arrays should be used, that is IKRGX, IKRGX-, IKRGY, IKRGY-,I KRGZ and IKRGZ-, instead of the IKRG keyword.

Example

The example below defines an input box for the whole grid and for layers one to three, for layer one IKRO is set equal to 0.850, for layer two IKRO equals 0.875, and for layer three IKRO equals 0.900.

```
--
--        ARRAY       CONSTANT      ---------- BOX ---------
--                                  I1  I2   J1  J2   K1  K2
EQUALS
          IKRO        0.8500        1*  1*   1*  1*    1   1  / IKRO FOR LAYER 1
          IKRO        0.8750        1*  1*   1*  1*    2   2  / IKRO FOR LAYER 2
          IKRO        0.9000        1*  1*   1*  1*    3   3  / IKRO FOR LAYER 3
/
```

8.3.108 IKRORG – END-POINT SCALING OF GRID CELL KRO(SGCR) (IMBIBITION)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

IKRORG defines the scaling parameter for the imbibition relative permeability of oil at the critical gas saturation (ISGCR), for all the cells in the model via an array. The ENDSCALE keyword in the RUNSPEC section should be activated to enable end-point scaling and the use of this keyword. In addition, the HYSTER option on the SATOPTS keyword in the RUNSPEC section has to be activated to invoke the Hysteresis option. The SCALCERS keyword in the PROPS section defines the options used in the re-scaling process, the options are two point scaling and three point scaling.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	IKRORG	IKRORG is an array of positive real numbers which are greater than zero and less than or equal to 1.0, that are the assigned imbibition scaling IKRORG values for each cell in the model. Repeat counts may be used, for example 50*0.850.			Taken from cell allocated relative permeability table.
		dimensionless	dimensionless	dimensionless	

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) The keyword is terminated by a "/".
3) Note this the directional independent version of the keyword used with the end-point scaling option. If directional end-point scaling has been activated then the IKRORGX±, IKRORGY± and IKRORGZ± series of the keywords should be used.

Table 8.53: IKRORG Keyword Description

When the IKRORG keyword is present in the input deck then the scaling matches the imbibition relative permeability at the critical saturation of the displacing phase.

If three point scaling option has been selected via the SCALECRS keyword in the PROPS section the critical displacing phase is defined as:

No	Keywords Present	Critical Saturation
1	IKRORW	$S_{critical}$ = 1.0 – ISWCR - ISGL
2	IKRORG	$S_{critical}$ = 1.0 – ISGCR - SWL

Table 8.54: Critical Displacement Relationships

End-point scaling allows the entered relative permeability functions to be re-scaled based on the saturation end-points allocated to each cell by the ISWL, ISWCR, ISWU, ISGL, ISGCR, ISGU, ISOWCR, and ISOGCR saturation grid arrays for the saturation end-points, In addition end-point scaling may be directional dependent in which case the directional dependent versions of the aforementioned arrays should be used, that is ISWUX, ISWUY and ISWUZ instead of ISWU, There is also the facility to make the directional end-point scaling reversible or non-reversible and if the non-reversible option is selected the non-reversible

versions of the aforementioned arrays should be used, that is ISWUX, ISWUX-, ISWUY, ISWUY-, ISWUZ and ISWUZ-, instead of the ISWU keyword.

End-point scaling also allows the entered relative permeability functions to be scale on the relative permeability values using the IKRG, IKRGR, IKRO, IKRORG, IKRORW, IKRW and IKRWR relative permeability grid cell arrays for the relative permeability end-point data. In addition end-point scaling may be directional dependent in which case the directional dependent versions of the aforementioned arrays should be used, that is IKRGX, IKRGY and IKRGZ instead of IKRG, There is also the facility to make the directional end-point scaling reversible or non-reversible and if the non-reversible option is selected the non-reversible versions of the aforementioned arrays should be used, that is IKRGX, IKRGX-, IKRGY, IKRGY-,I KRGZ and IKRGZ-, instead of the IKRG keyword.

Examples

The first example defines an input box for the whole grid and for layers one to three, for layer one IKRORG is set equal to 0.755, for layer two IKRORG equals 0.775, and for layer three IKRORG equals 0.800.

```
--
--       DEFINE INPUT BOX FOR EDITING INPUT ARRAYS (NX=100, NY=100)
--
--       ---------- BOX ---------
--       I1  I2   J1  J2   K1  K2
BOX
         1*  1*   1*  1*   1   3                          / DEFINE BOX AREA
--
--       SET IKRORG VALUES FOR THREE LAYERS IN THE MODEL
--
IKRORG
         10000*0.755  10000*0.775  10000*0.800               /
--
--       DEFINE END OF INPUT BOX EDITING OF INPUT ARRAYS
--
ENDBOX
```

The next example does exactly the same thing using the EQUALS keyword instead.

```
--
--       ARRAY       CONSTANT     ---------- BOX ---------
--                                I1  I2   J1  J2   K1  K2
EQUALS
         IKRORG      0.7550       1*  1*   1*  1*    1   1  / IKRORG FOR LAYER 1
         IKRORG      0.7750       1*  1*   1*  1*    2   2  / IKRORG FOR LAYER 2
         IKRORG      0.8000       1*  1*   1*  1*    3   3  / IKRORG FOR LAYER 3
/
```

8.3.109 IKRORW – END-POINT SCALING OF GRID CELL KRO(SWCR) (IMBIBITION)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

IKRORW defines the scaling parameter for the imbibition relative permeability of oil at the critical water saturation (ISWCR), for all the cells in the model via an array. The ENDSCALE keyword in the RUNSPEC section should be activated to enable end-point scaling and the use of this keyword. In addition, the HYSTER option on the SATOPTS keyword in the RUNSPEC section has to be activated to invoke the Hysteresis option. The SCALECRS keyword in the PROPS section defines the options used in the re-scaling process, the options are two point scaling and three point scaling.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	IKRORW	IKRORW is an array of positive real numbers which are greater than zero and less than or equal to 1.0, that are the assigned imbibition scaling IKRORW values for each cell in the model. Repeat counts may be used, for example 50*0.850			Taken from cell allocated relative permeability table.
		dimensionless	dimensionless	dimensionless	

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) The keyword is terminated by a "/".
3) Note this the directional independent version of the keyword used with the end-point scaling option. If directional end-point scaling has been activated then the IKRORWX±, IKRORWY± and IKRORWZ± series of the keywords should be used.

Table 8.55: IKRORW Keyword Description

When the IKRORW keyword is present in the input deck then the scaling matches the imbibition relative permeability at the critical saturation of the displacing phase.

If three point scaling option has been selected via the SCALECRS keyword in the PROPS section the critical displacing phase is defined as:

No	Keywords Present	Critical Saturation
1	IKRORW	$S_{critical}$ = 1.0 – ISWCR - ISGL
2	IKRORG	$S_{critical}$ = 1.0 – ISGCR - ISWL

Table 8.56: Critical Displacement Relationships

End-point scaling allows the entered relative permeability functions to be re-scaled based on the saturation end-points allocated to each cell by the ISWL, ISWCR, ISWU, ISGL, ISGCR, ISGU, ISOWCR, and ISOGCR saturation grid arrays for the saturation end-points, In addition end-point scaling may be directional dependent in which case the directional dependent versions of the aforementioned arrays should be used, that is ISWUX, ISWUY and ISWUZ instead of ISWU, There is also the facility to make the directional end-point scaling reversible or non-reversible and if the non-reversible option is selected the non-reversible

versions of the aforementioned arrays should be used, that is ISWUX, ISWUX-, ISWUY, ISWUY-, ISWUZ and ISWUZ-, instead of the ISWU keyword.

End-point scaling also allows the entered relative permeability functions to be scale on the relative permeability values using the IKRG, IKRGR, IKRO, IKRORG, IKRORW, IKRW and IKRWR relative permeability grid cell arrays for the relative permeability end-point data. In addition end-point scaling may be directional dependent in which case the directional dependent versions of the aforementioned arrays should be used, that is IKRGX, IKRGY and IKRGZ instead of IKRG, There is also the facility to make the directional end-point scaling reversible or non-reversible and if the non-reversible option is selected the non-reversible versions of the aforementioned arrays should be used, that is IKRGX, IKRGX-, IKRGY, IKRGY-,I KRGZ and IKRGZ-, instead of the IKRG keyword.

Examples

The first example defines an input box for the whole grid and for layers one to three, for layer one IKRORW is set equal to 0.755, for layer two IKRORW equals 0.775, and for layer three IKRORW equals 0.800.

```
--
--       DEFINE INPUT BOX FOR EDITING INPUT ARRAYS (NX=100, NY=100)
--
--       ---------- BOX ---------
--       I1  I2   J1  J2   K1  K2
BOX
         1*  1*   1*  1*   1   3                          / DEFINE BOX AREA
--
--       SET IKRORW VALUES FOR THREE LAYERS IN THE MODEL
--
IKRORW
         10000*0.755  10000*0.775  10000*0.800               /
--
--       DEFINE END OF INPUT BOX EDITING OF INPUT ARRAYS
--
ENDBOX
```

The next example does exactly the same thing using the EQUALS keyword instead.

```
--
--       ARRAY       CONSTANT     ---------- BOX ---------
--                                I1  I2   J1  J2   K1  K2
EQUALS
         IKRORW      0.7550       1*  1*   1*  1*    1   1  / IKRORW FOR LAYER 1
         IKRORW      0.7750       1*  1*   1*  1*    2   2  / IKRORW FOR LAYER 2
         IKRORW      0.8000       1*  1*   1*  1*    3   3  / IKRORW FOR LAYER 3
/
```

8.3.110 IKRW – END-POINT SCALING OF GRID CELL KRW(SW =1.0) (IMBIBITION)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

IKRW defines the scaling parameter at the maximum imbibition water relative permeability value (ISWU), that is for Sw = 1.0, for all the cells in the model via an array. The ENDSCALE keyword in the RUNSPEC section should be activated to enable end-point scaling and the use of this keyword. In addition, the HYSTER option on the SATOPTS keyword in the RUNSPEC section has to be activated to invoke the Hysteresis option. The SCALCERS keyword in the PROPS section defines the options used in the re-scaling process, the options are two point scaling and three point scaling.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	IKRW	IKRW is an array of positive real numbers which are greater than zero and less than or equal to 1.0, that are the assigned imbibition scaling IKRW values for each cell in the model. Repeat counts may be used, for example 50*1.000.			Taken from cell allocated relative permeability table.
		dimensionless	dimensionless	dimensionless	

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) The keyword is terminated by a "/".
3) Note this the directional independent version of the keyword used with the end-point scaling option. If directional end-point scaling has been activated then the IKRWX±, IKRWY± and IKRWZ± series of the keywords should be used.

Table 8.57: IKRW Keyword Description

For the two point scaling option and for the IKRWR water relative permeability array NOT present in the input deck the krw value for a grid block is scaled by:

$$k_{rw} = k_{rw\,TABLE}\left(\frac{IKRW}{k_{rw\,TABLE-MAX}}\right) \tag{8.60}$$

Where:

krw = the resulting IKRW value for a grid cell.
$IKRW$ = the scaling water relative permeability value from the IKRW array for a given cell.
$k_{rw\,TABLE}$ = the water relative permeability from a grid block's oil relative permeability table at the grid blocks water saturation.
$k_{rw\,TABLE-MAX}$ = the maximum water relative permeability from a grid block's water relative table, that is at the maximum water saturation.

If the IKRWR keyword is present in the input deck then the scaling matches the imbibition relative permeability at the critical saturation of the displacing phase.

If three point scaling option has been selected via the SCALECRS keyword in the PROPS section the critical displacing phase is defined as:

No	Phases Present	Critical Saturation
1	Gas-Oil	$S_{critical}$ = 1.0 – ISOWCR - ISGL
2	Gas-Oil-Water	$S_{critical}$ = 1.0 – ISOWCR - ISGL
3	Gas-Water	$S_{critical}$ = 1.0 – ISGCR

Table 8.58: Critical Displacement Relationships

End-point scaling allows the entered relative permeability functions to be re-scaled based on the saturation end-points allocated to each cell by the ISWL, ISWCR, ISWU, ISGL, ISGCR, ISGU, ISOWCR, and ISOGCR saturation grid arrays for the saturation end-points, In addition end-point scaling may be directional dependent in which case the directional dependent versions of the aforementioned arrays should be used, that is ISWUX, ISWUY and ISWUZ instead of ISWU, There is also the facility to make the directional end-point scaling reversible or non-reversible and if the non-reversible option is selected the non-reversible versions of the aforementioned arrays should be used, that is ISWUX, ISWUX-, ISWUY, ISWUY-, ISWUZ and ISWUZ-, instead of the ISWU keyword.

End-point scaling also allows the entered relative permeability functions to be scale on the relative permeability values using the IKRG, IKRGR, IKRO, IKRORG, IKRORW, IKRW and IKRWR relative permeability grid cell arrays for the relative permeability end-point data. In addition end-point scaling may be directional dependent in which case the directional dependent versions of the aforementioned arrays should be used, that is IKRGX, IKRGY and IKRGZ instead of IKRG, There is also the facility to make the directional end-point scaling reversible or non-reversible and if the non-reversible option is selected the non-reversible versions of the aforementioned arrays should be used, that is IKRGX, IKRGX-, IKRGY, IKRGY-,I KRGZ and IKRGZ-, instead of the IKRG keyword.

Example

The example below defines an input box for the whole grid and for layers one to three, for layer one IKRW is set equal to 0.850, for layer two IKRW equals 0.875, and for layer three IKRW equals 0.900.

```
--
--       DEFINE INPUT BOX FOR EDITING INPUT ARRAYS (NX=100, NY=100)
--
--       ---------- BOX ---------
--       I1  I2   J1  J2   K1  K2
BOX
         1*  1*   1*  1*   1   3                          / DEFINE BOX AREA
--
--       SET IKRW VALUES FOR THREE LAYERS IN THE MODEL
--
IKRW
         10000*0.850  10000*0.875  10000*0.900               /
--
--       DEFINE END OF INPUT BOX EDITING OF INPUT ARRAYS
--
ENDBOX
```

8.3.111 IKRWR – END-POINT SCALING OF GRID CELL KRWR(SOWCR) (IMBIBITION)

Description

IKRWR defines the scaling parameter at the imbibition critical oil to water saturation value (ISOWCR), for the imbibition water relative permeability curve, for all the cells in the model via an array. The ENDSCALE keyword in the RUNSPEC section should be activated to enable end-point scaling and the use of this keyword. In addition, the HYSTER option on the SATOPTS keyword in the RUNSPEC section has to be activated to invoke the Hysteresis option. The SCALCERS keyword in the PROPS section defines the options used in the re-scaling process, the options are two point scaling and three point scaling.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	IKRWR	IKRWR is an array of positive real numbers which are greater than zero and less than or equal to 1.0, that are the assigned imbibition scaling IKRWR values for each cell in the model. Repeat counts may be used, for example 50*1.000.			Taken from cell allocated relative permeability table.
		dimensionless	dimensionless	dimensionless	

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) The keyword is terminated by a "/".
3) Note this the directional independent version of the keyword used with the end-point scaling option. If directional end-point scaling has been activated then the IKRWRX±, IKRWRY± and IKRWRZ± series of the keywords should be used.

Table 8.59: IKRWR Keyword Description

When the IKRWR keyword is present in the input deck then the scaling matches the imbibition relative permeability at the critical saturation of the displacing phase.

If three point scaling option has been selected via the SCALECRS keyword in the PROPS section the critical displacing phase is defined as:

No	Phases Present	Critical Saturation
1	Gas-Oil	$S_{critical}$ = 1.0 – ISOWCR - ISGL
2	Gas-Oil-Water	$S_{critical}$ = 1.0 – ISOWCR - ISGL
3	Gas-Water	$S_{critical}$ = 1.0 – ISGCR

Table 8.60: Critical Displacement Relationships

End-point scaling allows the entered relative permeability functions to be re-scaled based on the saturation end-points allocated to each cell by the ISWL, ISWCR, ISWU, ISGL, ISGCR, ISGU, ISOWCR, and ISOGCR saturation grid arrays for the saturation end-points, In addition end-point scaling may be directional dependent in which case the directional dependent versions of the aforementioned arrays should be used, that is ISWUX, ISWUY and ISWUZ instead of ISWU, There is also the facility to make the directional end-

point scaling reversible or non-reversible and if the non-reversible option is selected the non-reversible versions of the aforementioned arrays should be used, that is ISWUX, ISWUX-, ISWUY, ISWUY-, ISWUZ and ISWUZ-, instead of the ISWU keyword.

End-point scaling also allows the entered relative permeability functions to be scale on the relative permeability values using the IKRG, IKRGR, IKRO, IKRORG, IKRORW, IKRW and IKRWR relative permeability grid cell arrays for the relative permeability end-point data. In addition end-point scaling may be directional dependent in which case the directional dependent versions of the aforementioned arrays should be used, that is IKRGX, IKRGY and IKRGZ instead of IKRG, There is also the facility to make the directional end-point scaling reversible or non-reversible and if the non-reversible option is selected the non-reversible versions of the aforementioned arrays should be used, that is IKRGX, IKRGX-, IKRGY, IKRGY-,I KRGZ and IKRGZ-, instead of the IKRG keyword.

Examples

The first example defines an input box for the whole grid and for layers one to three, for layer one IKRWR is set equal to 0.755, for layer two IKRWR equals 0.775, and for layer three IKRWR equals 0.800.

```
--
--       DEFINE INPUT BOX FOR EDITING INPUT ARRAYS (NX=100, NY=100)
--
--       ---------- BOX ---------
--       I1  I2   J1  J2   K1  K2
BOX
         1*  1*   1*  1*   1   3                          / DEFINE BOX AREA
--
--       SET IKRWR VALUES FOR THREE LAYERS IN THE MODEL
--
IKRWR
         10000*0.755  10000*0.775  10000*0.800               /
--
--       DEFINE END OF INPUT BOX EDITING OF INPUT ARRAYS
--
ENDBOX
```

The next example does exactly the same thing using the EQUALS keyword instead.

```
--
--       ARRAY       CONSTANT     ---------- BOX ---------
--                                I1  I2   J1  J2   K1  K2
EQUALS
         IKRWR       0.7550       1*  1*   1*  1*    1   1  / IKRWR FOR LAYER 1
         IKRWR       0.7750       1*  1*   1*  1*    2   2  / IKRWR FOR LAYER 2
         IKRWR       0.8000       1*  1*   1*  1*    3   3  / IKRWR FOR LAYER 3
/
```

8.3.112 IMKRVD – IMBIBITION RELATIVE PERMEABILITY END-POINTS VERSUS DEPTH FUNCTIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword defines the maximum imbibition oil, gas, and water relative permeability versus depth for the three phases. The ENDSCALE keyword in the RUNSPEC section should be activated to enable end-point scaling and the use of this keyword. In addition, the HYSTER option on the SATOPTS keyword in the RUNSPEC section has to be activated to invoke the Hysteresis option.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.113 IMPCVD – IMBIBITION MAXIMUM CAPILLARY PRESSURE VERSUS DEPTH FUNCTIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword defines the maximum imbibition gas-oil and water-oil capillary pressure values versus depth for when the end-point scaling option has been invoked by the ENDSCALE keyword in the RUNSPEC section and the HYSTER option on the SATOPTS keyword in the RUNSPEC section has been activated to invoke the Hysteresis option.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.114 IMPORT – IMPORT GRID FILE DATA AT THE CURRENT POSITION

The IMPORT keyword informs the simulator to continue reading input data from the specified IMPORT file. When the end of the IMPORT file is reached, input data is read from the next keyword in the current file. Normally IMPORT files are generated by grid pre-processing software and the keyword allows for both formatted and unformatted (binary) files to be loaded.

See *IMPORT – Import Grid File Data at the Current Position* in the GRID section for a full description.

8.3.115 IMPTVD – IMBIBITION RELATIVE PERMEABILITY SATURATION END-POINTS VERSUS DEPTH

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword defines the variation of the imbibition relative permeability saturation end-points (SWL, SWCR, etc.) for all three phases versus depth., for when the end-point scaling option has been invoked by the ENDSCALE keyword in the RUNSPEC section, and the HYSTER option on the SATOPTS keyword in the RUNSPEC section has been activated to invoke the Hysteresis option. This functionality is not supported in OPM Flow.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.116 IMSPCVD – IMBIBITION CAPILLARY PRESSURE CONNATE SATURATIONS VERSUS DEPTH

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword defines the imbibition capillary pressure gas and water connate saturations values versus depth for when the end-point scaling option has been invoked by the ENDSCALE keyword in the RUNSPEC section, and the HYSTER option on the SATOPTS keyword in the RUNSPEC section has been activated to invoke the Hysteresis option. This functionality is not supported in OPM Flow.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.117 INCLUDE – LOAD ANOTHER DATA FILE AT THE CURRENT POSITION

The INCLUDE keyword informs OPM Flow to continue reading input data from the specified INCLUDE file. When the end of the INCLUDE file is reached, or the ENDINC keyword is encountered in the included file, input data is read from the next keyword in the current file.

See *INCLUDE – Load Another Data File at the Current Position* in the GLOBAL section for a full description.

8.3.118 INTPC – ACTIVATE DUAL POROSITY INTEGRATED CAPILLARY PRESSURE OPTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The INTPC keyword activates the integrated capillary pressure option for the oil, gas or both phases, for when a Dual Porosity model has been activated by either the DUALPORO or DUALPERM keywords in the RUNSPEC section. In addition, the keyword can only be used if the Gravity Drainage option has been specified by either the GRAVDR or GRAVDRM in the RUNSPEC section. Basically, activating this feature results in the simulator adjusting the capillary pressure curves by integrating the matrix capillary pressure curves over the matrix block height to calculate the average saturation.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.119 IONXROCK - DEFINE ION EXCHANGE CONSTANT BY SATURATION TABLE REGIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The IONXROCK keyword activates ion exchange and defines the ion exchange constant by saturation table regions, for when the brine phase has been activated by the BRINE keyword and the Multi-Component Brine model, that allows for the water phase to have multiple water salinities, has been activated by the ECLMC keyword. Both keywords are in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.120 IONXSURF - DEFINE SURFACTANT ION EXCHANGE CONSTANT BY SATURATION TABLE REGIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The IONXROCK keyword activates ion exchange on surfactant micellae[198] and defines the ion exchange constant by surfactant equivalent molecular weight for saturation table regions, for when the brine and surfactant phases has been activated by the BRINE and SURFACT keywords, and the Multi-Component Brine model, that allows for the water phase to have multiple water salinities, has been activated by the ECLMC keyword. All three keywords are in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

[198] *Particle of colloidal dimensions that exists in equilibrium with the molecules or ions in solution from which it is formed. A micella or micelle (plural micellae or micelles, respectively) is an aggregate (or supramolecular assembly) of surfactant molecules dispersed in a liquid colloid. A typical micella in aqueous solution forms an aggregate with the hydrophilic "head" regions in contact with surrounding solvent, sequestering the hydrophobic single-tail regions in the micella centre (https://en.wikipedia.org/wiki/Micelle).*

8.3.121 IPCG – END-POINT SCALING OF GRID CELL GAS CAPILLARY PRESSURE (IMBIBITION)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

IPCG defines the maximum imbibition gas-oil capillary pressure values for all the cells in the model via an array. The ENDSCALE keyword in the RUNSPEC section should be activated to enable end-point scaling and the use of this keyword. In addition, the HYSTER option on the SATOPTS keyword in the RUNSPEC section has to be activated to invoke the hysteresis option. The keyword can be used with all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	IPCG	IPCG is an array of positive real numbers assigning the maximum imbibition gas capillary pressure values for each cell in the model. Repeat counts may be used, for example 30*100.0.			
		psia	bars	atm	None

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.

2) The keyword is terminated by a "/".

Table 8.61: IPCG Keyword Description

The capillary pressure for a grid block is scaled by:

$$P_c = P_{c_{TABLE}} \left(\frac{IPCG}{P_{c_{TABLE-MAX}}} \right) \quad (8.61)$$

Where:

Pc = the resulting imbibition gas-oil capillary pressure for a grid cell.
$IPCG$ = the maximum capillary pressure from the IPCG array for a given cell.
$P_{c_{TABLE}}$ = the capillary pressure in the imbibition capillary pressure table allocated to the grid block.
$P_{c_{TABLE-MAX}}$ = the maximum capillary pressure in the imbibition capillary pressure table allocated to the grid block at $S_g = 1 - S_{wco}$.

See also the PCG keyword for the equivalent drainage functionality.

Example

```
--
--        DEFINE GRID BLOCK IPCG DATA FOR ALL CELLS (BASED ON NX x NY x NZ = 300)
--
IPCG
          100*50.0  100*75.0  100*125.0                                    /
```

The above example defines the IPCG for 300 cells in the model as defined by the DIMENS keyword in the RUNSPEC section.

8.3.122 IPCW – END-POINT SCALING OF GRID CELL WATER CAPILLARY PRESSURE (IMBIBITION)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

IPCW defines the maximum imbibition water-oil or water-gas capillary pressure values for all the cells in the model via an array. The ENDSCALE keyword in the RUNSPEC section should be activated to enable end-point scaling and the use of this keyword. In addition, the HYSTER option on the SATOPTS keyword in the RUNSPEC section has to be activated to invoke the hysteresis option. The keyword can be used with all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	IPCW	IPCW is an array of positive real numbers assigning the maximum imbibition water capillary pressure values for each cell in the model. Repeat counts may be used, for example 30*100.0.			
		psia	bars	atm	None

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.

2) The keyword is terminated by a "/".

Table 8.62: IPCW Keyword Description

The capillary pressure for a grid block is scaled by:

$$P_c = P_{c_{TABLE}} \left(\frac{IPCW}{P_{c_{TABLE-MAX}}} \right) \quad (8.62)$$

Where:

Pc = the resulting imbibition water capillary pressure for a grid cell.
$IPCW$ = the maximum capillary pressure from the IPCW array for a given cell.
$P_{c_{TABLE}}$ = the capillary pressure in the imbibition capillary pressure table allocated to the grid block.
$P_{c_{TABLE-MAX}}$ = the maximum capillary pressure in the imbibition capillary pressure table allocated to the grid block (that is at the connate water saturation).

See also the PCW keyword for the equivalent drainage functionality.

Example

```
--
--        DEFINE GRID BLOCK IPCW DATA FOR ALL CELLS (BASED ON NX x NY x NZ = 300)
--
IPCW
          100*50.0  100*75.0   100*125.0                                    /
```

The above example defines the IPCW for 300 cells in the model as defined by the DIMENS keyword in the RUNSPEC section.

8.3.123 ISGCR – END-POINT SCALING OF GRID CELL CRITICAL GAS SATURATION (IMBIBITION)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

ISGCR defines the imbibition critical gas saturation for all the cells in the model via an array when the end-point scaling option has been invoked via the ENDSCALE keyword in the RUNSPEC section and the hysteresis model option has been activated on the SATOPTS keyword in the RUNSPEC section. The critical gas saturation is defined as the maximum gas saturation for which the gas relative permeability is zero in a two-phase relative permeability table.

The keyword can be used with all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	ISGCR	ISGCR is an array of real numbers assigning the critical gas saturation values to each cell in the model. The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword. Repeat counts may be used, for example 30*0.03			Taken from cell allocated relative permeability table.
		dimensionless	dimensionless	dimensionless	

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) The keyword is terminated by a "/".
3) Note this the directional independent version of the critical gas saturation used with the end-point scaling option. If directional end-point scaling has been activated then the ISGCRX± , ISGCRX± and ISGCRX± series of keywords should be used.

Table 8.63: ISGCR Keyword Description

End-point scaling allows the entered relative permeability functions to be re-scaled based on the saturation end-points allocated to each cell by the ISWL, ISWCR, ISWU, ISGL, ISGCR, ISGU, ISOWCR, and ISOGCR saturation grid arrays for the saturation end-points, and the IKRG, IKROG, IKROW and IKRW relative permeability grid cell arrays for the relative permeability end-point data. In addition end-point scaling may be directional dependent in which case the directional dependent versions of the aforementioned arrays should be used, that is ISGCRX, ISGCRY and ISGCRZ instead of ISGCR. There is also the facility to make the directional end-point scaling reversible or non-reversible and if the non-reversible option is selected the non-reversible versions of the aforementioned arrays should be used, that is ISGCRX, ISGCRX-, ISGCRY, ISGCRY-, ISGCRZ and ISGCRZ-, instead of the ISGCR keyword.

Example

```
--
--       DEFINE GRID BLOCK END-POINT ISGCR DATA FOR CELLS (NX x NY x NZ = 300)
ISGCR
         300*0.050                                                          /
```

The above example defines a constant critical gas saturation of 0.05 to all 300 cells in the model as defined by the DIMENS keyword in the RUNSPEC section.

8.3.124 ISGL – END-POINT SCALING OF GRID CELL CONNATE GAS SATURATION (IMBIBITION)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

ISGL defines the imbibition connate gas saturation for all the cells in the model via an array when the end-point scaling option has been invoked via the ENDSCALE keyword in the RUNSPEC section and the hysteresis model option has been activated on the SATOPTS keyword in the RUNSPEC section. The connate gas saturation is defined as the minimum gas saturation in a two-phase gas relative permeability table.

The keyword can be used with all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	ISGL	ISGL is an array of real numbers assigning the connate gas saturation values to each cell in the model. The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword. Repeat counts may be used, for example 30*0.03 dimensionless			Taken from cell allocated relative permeability table.
		dimensionless	dimensionless	dimensionless	

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) The keyword is terminated by a "/".
3) Note this the directional independent version of the connate gas saturation array used with the end-point scaling option. If directional end-point scaling has been activated then the ISGLX± , ISGLY± and ISGZ± series of keywords should be used.

Table 8.64: ISGL Keyword Description

End-point scaling allows the entered relative permeability functions to be re-scaled based on the saturation end-points allocated to each cell by the ISWL, ISWCR, ISWU, ISGL, ISGCR, ISGU, ISOWCR, and ISOGCR saturation grid arrays for the saturation end-points, and the KRG, KROG, KROW and KRW relative permeability grid cell arrays for the relative permeability end-point data. In addition end-point scaling may be directional dependent in which case the directional dependent versions of the aforementioned arrays should be used, that is ISGLX, ISGLY and ISGLZ instead of ISGL. There is also the facility to make the directional end-point scaling reversible or non-reversible and if the non-reversible option is selected the non-reversible versions of the aforementioned arrays should be used, that is ISGLX, ISGLX-, ISGLY, ISGLY-, ISGLZ and ISGLZ-, instead of the ISGL keyword.

Example

```
--
--        DEFINE GRID BLOCK END-POINT ISGL DATA FOR ALL CELLS (NX x NY x NZ = 300)
ISGL
          300*0.030                                                              /
```

The above example defines a constant connate gas saturation of 0.03 to all 300 cells in the model as defined by the DIMENS keyword in the RUNSPEC section.

8.3.125 ISGLPC – END-POINT SCALING OF GRID CELL CAPILLARY PRESSURE CONNATE GAS SATURATION (IMBIBITION)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

ISGLPC defines the imbibition connate gas saturation for all the cells in the model via an array when the end-point scaling option has been invoked via the ENDSCALE keyword in the RUNSPEC section and the hysteresis model option has been activated on the SATOPTS keyword in the RUNSPEC section. The connate gas saturation is defined as the minimum gas saturation in a two-phase gas relative permeability table. The keyword only applies the scaling to the imbibition capillary pressures tables, unlike the ISGL keyword that applies the scaling to both the capillary pressure and relative permeability tables. The keyword can be used with all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	ISGLPC	ISGLPC is an array of real numbers assigning the connate gas saturation values to each cell in the model. The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword. If ISGLPC is omitted from the input deck the values will be defaulted to those on the ISGL series of keywords. If the ISGL series of keywords are missing from the input deck then the values are taken from the cell allocated capillary pressure table. Repeat counts may be used, for example 30*0.03			Taken from SGL or from the cell allocated capillary pressure table.
		dimensionless	dimensionless	dimensionless	

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) The keyword is terminated by a "/".
3) Note this the directional independent version of the connate gas saturation array used with the end-point scaling option. If directional end-point scaling has been activated then the ISGLX± , ISGLY± and ISGZ± series of keywords should be used.

Table 8.65: ISGLPC Keyword Description

End-point scaling allows the entered relative permeability functions to be re-scaled based on the saturation end-points allocated to each cell by the ISWL, ISWCR, ISWU, ISGL, ISGCR, ISGU, ISOWCR, and ISOGCR saturation grid arrays for the saturation end-points, and the KRG, KROG, KROW and KRW relative permeability grid cell arrays for the relative permeability end-point data. In addition end-point scaling may be directional dependent in which case the directional dependent versions of the aforementioned arrays should be used, that is ISGLX, ISGLY and ISGLZ instead of ISGL or ISGLPC. There is also the facility to make the directional end-point scaling reversible or non-reversible and if the non-reversible option is selected the non-reversible versions of the aforementioned arrays should be used, that is ISGLX, ISGLX-, ISGLY, ISGLY-, ISGLZ and ISGLZ-, instead of the ISGL or ISGLPC keywords.

Missing Some Functionality - Use with Caution.

Example

```
--
--       DEFINE GRID BLOCK END-POINT ISGLPC DATA FOR ALL CELLS
-       (NX x NY x NZ = 300)
ISGLPC
         300*0.030                                                    /
```

The above example defines a constant connate gas saturation of 0.03 to all 300 cells in the model as defined by the DIMENS keyword in the RUNSPEC section.

8.3.126 ISGU – END-POINT SCALING OF GRID CELL MAXIMUM GAS SATURATION (IMBIBITION)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

ISGU defines the imbibition maximum gas saturation for all the cells in the model via an array when the end-point scaling option has been invoked via the ENDSCALE keyword in the RUNSPEC section and the hysteresis model option has been activated on the SATOPTS keyword in the RUNSPEC section. The maximum gas saturation is defined as the maximum gas saturation in a two-phase gas relative permeability table. The keyword can be used with all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	ISGU	ISGU is an array of real numbers assigning the maximum gas saturation values to each cell in the model. The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword. Repeat counts may be used, for example 30*0.70			Taken from cell allocated relative permeability table.
		dimensionless	dimensionless	dimensionless	

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) The keyword is terminated by a "/".
3) Note this the directional independent version of the maximum gas saturation array used with the end-point scaling option. If directional end-point scaling has been activated then the ISGUX± , ISGUX± and ISGU± series of keywords should be used.

Table 8.66: ISGU Keyword Description

End-point scaling allows the entered relative permeability functions to be re-scaled based on the saturation end-points allocated to each cell by the ISWL, ISWCR, ISWU, ISGL, ISGCR, ISGU, ISOWCR, and ISOGCR saturation grid arrays for the saturation end-points, and the KRG, KROG, KROW and KRW relative permeability grid cell arrays for the relative permeability end-point data. In addition end-point scaling may be directional dependent in which case the directional dependent versions of the aforementioned arrays should be used, that is ISGUX, ISGUY and ISGUZ instead of ISGU. There is also the facility to make the directional end-point scaling reversible or non-reversible and if the non-reversible option is selected the non-reversible versions of the aforementioned arrays should be used, that is ISGUX, ISGUX-, ISGUY, ISGUY-, ISGUZ and ISGUZ-, instead of the ISGU keyword.

Example

```
--
--        DEFINE GRID BLOCK END-POINT ISGU DATA FOR ALL CELLS (NX x NY x NZ = 300)
ISGU
          300*0.700                                                          /
```

The above example defines a constant connate gas saturation of 0.70 to all 300 cells in the model as defined by the DIMENS keyword in the RUNSPEC section.

8.3.127 ISOGCR – END-POINT SCALING OF GRID CELL CRITICAL OIL SATURATION TO GAS (IMBIBITION)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

ISOGCR defines the imbibition critical oil saturation for all the cells in the model via an array when the end-point scaling option has been invoked via the ENDSCALE in the RUNSPEC section and the hysteresis model option has been activated on the SATOPTS keyword in the RUNSPEC section. The critical oil saturation with respect to gas is defined as the maximum oil saturation for which the oil relative permeability is zero in a two-phase gas-oil relative permeability table. The keyword can be used with all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	ISOGCR	ISOGCR is an array of real numbers assigning the critical oil saturation with respect to gas values to each cell in the model. The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword. Repeat counts may be used, for example 30*0.30			Taken from cell allocated relative permeability table.
		dimensionless	dimensionless	dimensionless	

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) The keyword is terminated by a "/".
3) Note this the directional independent version of the critical gas saturation used with the end-point scaling option. If directional end-point scaling has been activated then the ISOGCRX± , ISOGCRX± and ISOGCRX± series of keywords should be used.

Table 8.67: ISOGCR Keyword Description

End-point scaling allows the entered relative permeability functions to be re-scaled based on the saturation end-points allocated to each cell by the ISWL, ISWCR, ISWU, ISGL, ISGCR, ISGU, ISOWCR, and ISOGCR saturation grid arrays for the saturation end-points, and the KRG, KROG, KROW and KRW relative permeability grid cell arrays for the relative permeability end-point data. In addition end-point scaling may be directional dependent in which case the directional dependent versions of the aforementioned arrays should be used, that is ISOGCRX, ISOGCRY and ISOGCRZ instead of ISOGCR. There is also the facility to make the directional end-point scaling reversible or non-reversible and if the non-reversible option is selected the non-reversible versions of the aforementioned arrays should be used, that is ISOGCRX, ISOGCRX-, ISOGCRY, ISOGCRY-, ISOGCRZ and ISOGCRZ-, instead of the ISOGCR keyword.

Example

```
--
--       DEFINE GRID BLOCK END-POINT ISOGCR DATA FOR ALL CELLS
--       (NX x NY x NZ = 300)
--
ISOGCR
         300*0.200                                                    /
```

The above example defines a constant critical gas saturation of 0.20 to all 300 cells in the model as defined by the DIMENS keyword in the RUNSPEC section.

8.3.128 ISOWCR – END-POINT SCALING OF GRID CELL CRITICAL OIL SATURATION TO WATER (IMBIB.)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

ISOWCR defines the imbibition critical oil saturation for all the cells in the model via an array when the end-point scaling option has been invoked via the ENDSCALE in the RUNSPEC section and the hysteresis model option has been activated on the SATOPTS keyword in the RUNSPEC section. The critical oil saturation with respect to water is defined as the maximum oil saturation for which the oil relative permeability is zero in a two-phase oil-water relative permeability table. The keyword can be used with all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	ISOWCR	ISOWCR is an array of real numbers assigning the critical oil saturation with respect to water values to each cell in the model. The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword. Repeat counts may be used, for example 30*0.30			Taken from cell allocated relative permeability table.
		dimensionless	dimensionless	dimensionless	

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) The keyword is terminated by a "/".
3) Note this the directional independent version of the critical gas saturation used with the end-point scaling option. If directional end-point scaling has been activated then the ISOWCRX± , ISOWCRX± and ISOWCRX± series of keywords should be used.

Table 8.68: ISOWCR Keyword Description

End-point scaling allows the entered relative permeability functions to be re-scaled based on the saturation end-points allocated to each cell by the ISWL, ISWCR, ISWU, ISGL, ISGCR, ISGU, ISOWCR, and ISOGCR saturation grid arrays for the saturation end-points, and the KRG, KROG, KROW and KRW relative permeability grid cell arrays for the relative permeability end-point data. In addition end-point scaling may be directional dependent in which case the directional dependent versions of the aforementioned arrays should be used, that is ISOWCRX, ISOWCRY and ISOWCRZ instead of ISOWCR. There is also the facility to make the directional end-point scaling reversible or non-reversible and if the non-reversible option is selected the non-reversible versions of the aforementioned arrays should be used, that is ISOWCRX, ISOWCRX-, ISOWCRY, ISOWCRY-, ISOWCRZ and ISOWCRZ-, instead of the ISOWCR keyword.

Example

```
--
--       DEFINE GRID BLOCK END-POINT ISOWCR DATA FOR ALL CELLS
--       (NX x NY x NZ = 300)
--
ISOWCR
         300*0.200                                                    /
```

The above example defines a constant critical gas saturation of 0.20 to all 300 cells in the model as defined by the DIMENS keyword in the RUNSPEC section

8.3.129 ISWCR – END-POINT SCALING OF GRID CELL CRITICAL WATER SATURATION (IMBIBITION)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

ISWCR defines the imbibition critical water saturation for all the cells in the model via an array when the end-point scaling option has been invoked via the ENDSCALE in the RUNSPEC section and the hysteresis model option has been activated on the SATOPTS keyword in the RUNSPEC section. The critical water saturation is defined as the maximum water saturation for which the water relative permeability is zero in a two-phase relative permeability table. The keyword can be used with all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	ISWCR	ISWCR is an array of real numbers assigning the critical water saturation values to each cell in the model. The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword. Repeat counts may be used, for example 30*0.20			Taken from cell allocated relative permeability table.
		dimensionless	dimensionless	dimensionless	

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) The keyword is terminated by a "/".
3) Note this the directional independent version of the critical water saturation used with the end-point scaling option. If directional end-point scaling has been activated then the ISWCRX± , ISWCRX± and ISWCRX± series of keywords should be used.

Table 8.69: ISWCR Keyword Description

End-point scaling allows the entered relative permeability functions to be re-scaled based on the saturation end-points allocated to each cell by the ISWL, ISWCR, ISWU, ISGL, ISGCR, ISGU, ISOWCR, and ISOGCR saturation grid arrays for the saturation end-points, and the KRG, KROG, KROW and KRW relative permeability grid cell arrays for the relative permeability end-point data. In addition end-point scaling may be directional dependent in which case the directional dependent versions of the aforementioned arrays should be used, that is ISWCRX, ISWCRY and ISWCRZ instead of ISWCR. There is also the facility to make the directional end-point scaling reversible or non-reversible and if the non-reversible option is selected the non-reversible versions of the aforementioned arrays should be used, that is ISWCRX, ISWCRX-, ISWCRY, ISWCRY-, ISWCRZ and ISWCRZ-, instead of the ISWCR keyword.

Example

```
--
--       DEFINE GRID BLOCK END-POINT ISWCR DATA FOR ALL CELLS
--       (NX x NY x NZ = 300)
--
ISWCR
  300*0.200                                                    /
```

The above example defines a constant critical water saturation of 0.20 to all 300 cells in the model as defined by the DIMENS keyword in the RUNSPEC section.

8.3.130 ISWL – END-POINT SCALING OF GRID CELL CONNATE WATER SATURATION (IMBIBITION)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

ISWL defines the imbibition connate water saturation for all the cells in the model via an array when the end-point scaling option has been invoked via the ENDSCALE in the RUNSPEC section and the hysteresis model option has been activated on the SATOPTS keyword in the RUNSPEC section. The connate water saturation is defined as the minimum water saturation in a two-phase water relative permeability table.

The keyword can be used with all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	ISWL	ISWL is an array of real numbers assigning the connate water saturation values to each cell in the model. The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword. Repeat counts may be used, for example 30*0.15			Taken from cell allocated relative permeability table.
		dimensionless	dimensionless	dimensionless	

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) The keyword is terminated by a "/".
3) Note this the directional independent version of the connate gas saturation array used with the end-point scaling option. If directional end-point scaling has been activated then the ISWLX± , ISWLY± and ISWZ± series of keywords should be used.

Table 8.70: ISWL Keyword Description

End-point scaling allows the entered relative permeability functions to be re-scaled based on the saturation end-points allocated to each cell by the ISWL, ISWCR, ISWU, ISGL, ISGCR, ISGU, ISOWCR, and ISOGCR saturation grid arrays for the saturation end-points, and the KRG, KROG, KROW and KRW relative permeability grid cell arrays for the relative permeability end-point data. In addition end-point scaling may be directional dependent in which case the directional dependent versions of the aforementioned arrays should be used, that is ISWLX, ISWLY and ISWLZ instead of ISWL. There is also the facility to make the directional end-point scaling reversible or non-reversible and if the non-reversible option is selected the non-reversible versions of the aforementioned arrays should be used, that is ISWLX, ISWLX-, ISWLY, ISWLY-, ISWLZ and ISWLZ-, instead of the ISWL keyword.

Example

```
--
--        DEFINE GRID BLOCK END-POINT ISWL DATA FOR ALL CELLS (NX x NY x NZ = 300)
--
ISWL
          300*0.150                                                              /
```

The above example defines a constant connate water saturation of 0.15 to all 300 cells in the model as defined by the DIMENS keyword in the RUNSPEC section.

8.3.131 ISWLPC – END-POINT SCALING GRID CELL CAPILLARY PRESSURE CONNATE WATER SATURATIONS (IMBIBITION)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

ISWLPC defines the imbibition connate water saturation for all the cells in the model via an array when the end-point scaling option has been invoked via the ENDSCALE in the RUNSPEC section and the hysteresis model option has been activated on the SATOPTS keyword in the RUNSPEC section. The connate water saturation is defined as the minimum water saturation in a two-phase water relative permeability table. The keyword only applies the scaling to the imbibition capillary pressures tables, unlike the ISWL keyword that applies the scaling to both the capillary pressure and relative permeability tables. The keyword can be used with all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	ISWLPC	ISWLPC is an array of real numbers assigning the connate water saturation values to each cell in the model. The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword. If ISWLPC is omitted from the input deck the values will be defaulted to those on the ISGL series of keywords. If the ISWL series of keywords are missing from the input deck then the values are taken from the cell allocated capillary pressure table. Repeat counts may be used, for example 30*0.15			Taken from SWL or from the cell allocated capillary pressure table.
		dimensionless	dimensionless	dimensionless	

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) The keyword is terminated by a "/".
3) Note this the directional independent version of the connate gas saturation array used with the end-point scaling option. If directional end-point scaling has been activated then the ISWLX± , ISWLY± and ISWZ± series of keywords should be used.

Table 8.71: ISWLPC Keyword Description

End-point scaling allows the entered relative permeability functions to be re-scaled based on the saturation end-points allocated to each cell by the ISWL, ISWCR, ISWU, ISGL, ISGCR, ISGU, ISOWCR, and ISOGCR saturation grid arrays for the saturation end-points, and the KRG, KROG, KROW and KRW relative permeability grid cell arrays for the relative permeability end-point data. In addition end-point scaling may be directional dependent in which case the directional dependent versions of the aforementioned arrays should be used, that is ISWLX, ISWLY and ISWLZ instead of ISWL or ISWLPC. There is also the facility to make the directional end-point scaling reversible or non-reversible and if the non-reversible option is selected the non-reversible versions of the aforementioned arrays should be used, that is ISWLX, ISWLX-, ISWLY, ISWLY-, ISWLZ and ISWLZ-, instead of the ISWL or ISWLPC keywords.

Missing Some Functionality - Use with Caution.

Example

```
--
--      DEFINE GRID BLOCK END-POINT ISWLPC DATA FOR ALL CELLS (
--      NX x NY x NZ = 300)
--
ISWLPC
         300*0.150                                                              /
```

The above example defines a constant connate water saturation of 0.15 to all 300 cells in the model as defined by the DIMENS keyword in the RUNSPEC section.

8.3.132 ISWU – END-POINT SCALING OF GRID CELL MAXIMUM WATER SATURATION (IMBIBITION)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

ISWU defines the imbibition maximum water saturation for all the cells in the model via an array when the end-point scaling option has been invoked via the ENDSCALE in the RUNSPEC section and the hysteresis model option has been activated on the SATOPTS keyword in the RUNSPEC section. The maximum water saturation is defined as the maximum water saturation in a two-phase water relative permeability table. The keyword can be used with all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	ISWU	ISWU is an array of real numbers assigning the maximum water saturation values to each cell in the model. The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword. Repeat counts may be used, for example 30*0.70			Taken from cell allocated relative permeability table.
		dimensionless	dimensionless	dimensionless	

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) The keyword is terminated by a "/".
3) Note this the directional independent version of the maximum water saturation array used with the end-point scaling option. If directional end-point scaling has been activated then the ISWUX± , ISWUX± and ISWU± series of keywords should be used.

Table 8.72: ISWU Keyword Description

End-point scaling allows the entered relative permeability functions to be re-scaled based on the saturation end-points allocated to each cell by the ISWL, ISWCR, ISWU, ISGL, ISGCR, ISGU, ISOWCR, and ISOGCR saturation grid arrays for the saturation end-points, and the KRG, KROG, KROW and KRW relative permeability grid cell arrays for the relative permeability end-point data. In addition end-point scaling may be directional dependent in which case the directional dependent versions of the aforementioned arrays should be used, that is ISWUX, ISWUY and ISWUZ instead of ISWU. There is also the facility to make the directional end-point scaling reversible or non-reversible and if the non-reversible option is selected the non-reversible versions of the aforementioned arrays should be used, that is ISWUX, ISWUX-, ISWUY, ISWUY-, ISWUZ and ISWUZ-, instead of the ISWU keyword.

Example

```
--
--        DEFINE GRID BLOCK END-POINT ISWU DATA FOR ALL CELLS (NX x NY x NZ = 300)
--
ISWU
          300*0.700                                                              /
```

The above example defines a constant connate gas saturation of 0.70 to all 300 cells in the model as defined by the DIMENS keyword in the RUNSPEC section.

8.3.133 KRG – END-POINT SCALING OF GRID CELL KRG(SGU) (DRAINAGE)

Description

KRG defines the scaling parameter at the maximum drainage gas relative permeability value (SGU), normally SGU is equal to 1.0 - Swc, for all the cells in the model via an array. The ENDSCALE keyword in the RUNSPEC section should be activated to enable end-point scaling and the use of this keyword. The SCALCERS keyword in the PROPS section defines the options used in the re-scaling process, the options are two point scaling and three point scaling.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	KRG	KRG is an array of positive real numbers which are greater than zero and less than or equal to 1.0, that are the assigned scaling KRG values for each cell in the model. Repeat counts may be used, for example 50*0.400. dimensionless			Taken from cell allocated relative permeability table.
		dimensionless	dimensionless	dimensionless	

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) The keyword is terminated by a "/".
3) Note this the directional independent version of the keyword used with the end-point scaling option. If directional end-point scaling has been activated then the KRGX±, KRGY± and KRGZ± series of the keywords should be used.

Table 8.73: KRG Keyword Description

For the two point scaling option and for the KRGR gas relative permeability array NOT present in the input deck the krg value for a grid block is scaled by:

$$k_{rg} = k_{rg\,TABLE}\left(\frac{KRG}{k_{rg\,TABLE-MAX}}\right) \quad (8.63)$$

Where:

krg = the resulting krg value for a grid cell.
KRG = the scaling gas relative permeability value from the KRG array for a given cell.
$k_{rg\,TABLE}$ = the gas relative permeability from a grid block's gas-oil table at the grid blocks gas saturation.
$k_{rg\,TABLE-MAX}$ = the maximum gas relative permeability from a grid block's gas-oil table, that is at the connate water saturation (Swc).

If the KRGR keyword is present in the input deck then the scaling matches the relative permeability at the critical saturation of the displacing phase.

If three point scaling option has been selected via the SCALECRS keyword in the PROPS section the critical displacing phase is defined as:

No	Phases Present	Critical Saturation
1	Gas-Oil	$S_{critical}$ = 1.0 – SOGCR - SWL
2	Gas-Oil-Water	$S_{critical}$ = 1.0 – SOGCR - SWL
3	Gas-Water	$S_{critical}$ = 1.0 – SWCR

Table 8.74: Critical Displacement Relationships

End-point scaling allows the entered relative permeability functions to be re-scaled based on the saturation end-points allocated to each cell by the SWL, SWCR, SWU, SGL, SGCR, SGU, SOWCR, and SOGCR saturation grid arrays for the saturation end-points, In addition end-point scaling may be directional dependent in which case the directional dependent versions of the aforementioned arrays should be used, that is SWUX, SWUY and SWUZ instead of SWU, There is also the facility to make the directional end-point scaling reversible or non-reversible and if the non-reversible option is selected the non-reversible versions of the aforementioned arrays should be used, that is SWUX, SWUX-, SWUY, SWUY-, SWUZ and SWUZ-, instead of the SWU keyword.

End-point scaling also allows the entered relative permeability functions to be scale on the relative permeability values using the KRG, KRGR, KRO, KRORG, KRORW, KRW and KRWR relative permeability grid cell arrays for the relative permeability end-point data. In addition end-point scaling may be directional dependent in which case the directional dependent versions of the aforementioned arrays should be used, that is KRGX, KRGY and KRGZ instead of KRG, There is also the facility to make the directional end-point scaling reversible or non-reversible and if the non-reversible option is selected the non-reversible versions of the aforementioned arrays should be used, that is KRGX, KRGX-, KRGY, KRGY-, KRGZ and KRGZ-, instead of the KRG keyword.

If the hysteresis model option has been activated on the SATOPTS keyword in the RUNSPEC section, then the equivalent imbibition arrays prefixed with the letter I, for example IKRG, can be used to scale KRG for the relative permeability imbibition tables.

Examples

The first example defines an input box for the whole grid and for layers one to three, for layer one KRG is set equal to 0.555, for layer two KRG equals 0.575, and for layer three KRG equals 0.600.

```
--
--       DEFINE INPUT BOX FOR EDITING INPUT ARRAYS (NX=100, NY=100)
--
--       ---------- BOX ---------
--       I1  I2   J1  J2   K1  K2
BOX
         1*  1*   1*  1*   1   3                          / DEFINE BOX AREA
--
--       SET KRG VALUES FOR THREE LAYERS IN THE MODEL
--
KRG
         10000*0.555  10000*0.575  10000*0.600              /
--
--       DEFINE END OF INPUT BOX EDITING OF INPUT ARRAYS
--
ENDBOX
```

The next example does exactly the same thing using the EQUALS keyword instead.

```
--
--        ARRAY       CONSTANT      ---------- BOX ---------
--                                  I1  I2   J1  J2   K1  K2
EQUALS
          KRG         0.5550         1*  1*   1*  1*   1   1  / KRG FOR LAYER 1
          KRG         0.5750         1*  1*   1*  1*   2   2  / KRG FOR LAYER 2
          KRG         0.6000         1*  1*   1*  1*   3   3  / KRG FOR LAYER 3
/
```

8.3.134 KRGR – END-POINT SCALING OF GRID CELL KRGR(1-SOGCR) (DRAINAGE)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

KRGR defines the scaling parameter at the relative permeability of gas at residual oil saturation (1 – SOGCR), or critical water saturation in a gas-water run (Swc), for all the cells in the model via an array. The ENDSCALE keyword in the RUNSPEC section should be activated to enable end-point scaling and the use of this keyword. The SCALCERS keyword in the PROPS section defines the options used in the re-scaling process, the options are two point scaling and three point scaling.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	KRGR	KRGR is an array of positive real numbers which are greater than zero and less than or equal to 1.0, that are the assigned scaling KRGR values for each cell in the model. In addition, for a given grid block KGRG should be less than KRG. Repeat counts may be used, for example 50*0.400.			Taken from cell allocated relative permeability table.
		dimensionless	dimensionless	dimensionless	

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) The keyword is terminated by a "/".
3) Note this the directional independent version of the keyword used with the end-point scaling option. If directional end-point scaling has been activated then the KRGRX± , KRGRY± and KRGRZ± series of the keywords should be used.

Table 8.75: KRGR Keyword Description

When the KRGR keyword is present in the input deck then the scaling matches the relative permeability at the critical saturation of the displacing phase (oil or water).

If three point scaling option has been selected via the SCALECRS keyword in the PROPS section the critical displacing phase is defined as:

No	Phases Present	Critical Saturation
1	Gas-Oil	$S_{critical}$ = 1.0 – SOGCR - SWL
2	Gas-Oil-Water	$S_{critical}$ = 1.0 – SOGCR - SWL
3	Gas-Water	$S_{critical}$ = 1.0 – SWCR

Table 8.76: Critical Displacement Relationships

End-point scaling allows the entered relative permeability functions to be re-scaled based on the saturation end-points allocated to each cell by the SWL, SWCR, SWU, SGL, SGCR, SGU, SOWCR, and SOGCR saturation grid arrays for the saturation end-points, In addition end-point scaling may be directional dependent in which case the directional dependent versions of the aforementioned arrays should be used, that is SWUX, SWUY and SWUZ instead of SWU, There is also the facility to make the directional end-point

scaling reversible or non-reversible and if the non-reversible option is selected the non-reversible versions of the aforementioned arrays should be used, that is SWUX, SWUX-, SWUY, SWUY-, SWUZ and SWUZ-, instead of the SWU keyword.

End-point scaling also allows the entered relative permeability functions to be scale on the relative permeability values using the KRG, KRGR, KRO, KRORG, KRORW, KRW and KRWR relative permeability grid cell arrays for the relative permeability end-point data. In addition end-point scaling may be directional dependent in which case the directional dependent versions of the aforementioned arrays should be used, that is KRGRX, KRGRY and KRGRZ instead of KRGR, There is also the facility to make the directional end-point scaling reversible or non-reversible and if the non-reversible option is selected the non-reversible versions of the aforementioned arrays should be used, that is KRGRX, KRGRX-, KRGRY, KRGRY-, KRGRZ and KRGRZ-, instead of the KRGR keyword.

If the hysteresis model option has been activated on the SATOPTS keyword in the RUNSPEC section, then the equivalent imbibition arrays suffixed with the letter I, for example IKRGR, can be used to define the KRGR for the relative permeability imbibition tables.

Examples

The first example defines an input box for the whole grid and for layers one to three, for layer one KRGR is set equal to 0.500, for layer two KRGR equals 0.570, and for layer three KRGR equals 0.580.

```
--
--       DEFINE INPUT BOX FOR EDITING INPUT ARRAYS (NX=100, NY=100)
--
--       ---------- BOX ---------
--       I1  I2   J1  J2   K1  K2
BOX
         1*  1*   1*  1*   1   3                          / DEFINE BOX AREA
--
--       SET KRGR VALUES FOR THREE LAYERS IN THE MODEL
--
KRGR
         10000*0.500  10000*0.570  10000*0.580              /
--
--       DEFINE END OF INPUT BOX EDITING OF INPUT ARRAYS
--
ENDBOX
```

The next example does exactly the same thing using the EQUALS keyword instead.

```
--
--       ARRAY       CONSTANT     ---------- BOX ---------
--                                I1  I2   J1  J2   K1  K2
EQUALS
         KRGR        0.5000       1*  1*   1*  1*    1   1  / KRGR FOR LAYER 1
         KRGR        0.5700       1*  1*   1*  1*    2   2  / KRGR FOR LAYER 2
         KRGR        0.5800       1*  1*   1*  1*    3   3  / KRGR FOR LAYER 3
/
```

8.3.135 KRO – END-POINT SCALING OF GRID CELL KRO(SWL) (DRAINAGE)

Description

KRO defines the scaling parameter for the drainage oil relative permeability value at the connate water saturation (SWL), for all the cells in the model via an array. The ENDSCALE keyword in the RUNSPEC section should be activated to enable end-point scaling and the use of this keyword. The SCALCERS keyword in the PROPS section defines the options used in the re-scaling process, the options are two point scaling and three point scaling.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	KRO	KRO is an array of positive real numbers which are greater than zero and less than or equal to 1.0, that are the assigned scaling KRO values for each cell in the model. Repeat counts may be used, for example 50*0.500.			Taken from cell allocated relative permeability table.
		dimensionless	dimensionless	dimensionless	

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) The keyword is terminated by a "/".
3) Note this the directional independent version of the keyword used with the end-point scaling option. If directional end-point scaling has been activated then the KROX±, KROY± and KROZ± series of the keywords should be used.

Table 8.77: KRO Keyword Description

For the two point scaling option and for the KRORW or KRORG oil relative permeability arrays NOT being present in the input deck the kro value for a grid block is scaled by:

$$k_{ro} = k_{ro\,TABLE}\left(\frac{KRO}{k_{ro\,TABLE-MAX}}\right) \quad (8.64)$$

Where:

kro = the resulting kro value for a grid cell.
KRO = the scaling oil relative permeability value from the KRO array for a given cell.
$k_{ro\,TABLE}$ = the oil relative permeability from a grid block's oil relative permeability table at the grid blocks oil saturation.
$k_{ro\,TABLE-MAX}$ = the maximum oil relative permeability from a grid block's oil relative table, that is at the critical water saturation (Swcr).

If the KRORW or KRORG keywords are present in the input deck then the scaling matches the relative permeability at the critical saturation of the displacing phase.

If three point scaling option has been selected via the SCALECRS keyword in the PROPS section the critical displacing phase is defined as:

No	Keywords Present	Critical Saturation
1	KRORW	$S_{critical}$ = 1.0 – SWCR - SGL
2	KRORG	$S_{critical}$ = 1.0 – SGCR - SWL

Table 8.78: Critical Displacement Relationships

End-point scaling allows the entered relative permeability functions to be re-scaled based on the saturation end-points allocated to each cell by the SWL, SWCR, SWU, SGL, SGCR, SGU, SOWCR, and SOGCR saturation grid arrays for the saturation end-points, In addition end-point scaling may be directional dependent in which case the directional dependent versions of the aforementioned arrays should be used, that is SWUX, SWUY and SWUZ instead of SWU, There is also the facility to make the directional end-point scaling reversible or non-reversible and if the non-reversible option is selected the non-reversible versions of the aforementioned arrays should be used, that is SWUX, SWUX-, SWUY, SWUY-, SWUZ and SWUZ-, instead of the SWU keyword.

End-point scaling also allows the entered relative permeability functions to be scale on the relative permeability values using the KRG, KRGR, KRO, KRORG, KRORW, KRW and KRWR relative permeability grid cell arrays for the relative permeability end-point data. In addition end-point scaling may be directional dependent in which case the directional dependent versions of the aforementioned arrays should be used, that is KROX, KROY and KROZ instead of KRO, There is also the facility to make the directional end-point scaling reversible or non-reversible and if the non-reversible option is selected the non-reversible versions of the aforementioned arrays should be used, that is KROX, KROX-, KROY, KROY-, KROZ and KROZ-, instead of the KRO keyword.

If the hysteresis model option has been activated on the SATOPTS keyword in the RUNSPEC section, then the equivalent imbibition arrays suffixed with the letter I, for example IKRO, can be used to define the KRO for the relative permeability imbibition tables.

Examples

The first example defines an input box for the whole grid and for layers one to three, for layer one KRO is set equal to 0.855, for layer two KRO equals 0.875, and for layer three KRO equals 0.900.

```
--
--       DEFINE INPUT BOX FOR EDITING INPUT ARRAYS (NX=100, NY=100)
--
--       ---------- BOX ---------
--       I1  I2   J1  J2   K1  K2
BOX
         1*  1*   1*  1*   1   3                          / DEFINE BOX AREA
--
--       SET KRO VALUES FOR THREE LAYERS IN THE MODEL
--
KRO
         10000*0.855  10000*0.875  10000*0.900              /
--
--       DEFINE END OF INPUT BOX EDITING OF INPUT ARRAYS
--
ENDBOX
```

The next example does exactly the same thing using the EQUALS keyword instead.

```
--
--        ARRAY       CONSTANT      ---------- BOX ---------
--                                  I1  I2   J1  J2   K1  K2
EQUALS
          KRO         0.8550        1*  1*   1*  1*    1   1  / KRO FOR LAYER 1
          KOG         0.8750        1*  1*   1*  1*    2   2  / KRO FOR LAYER 2
          KRO         0.9000        1*  1*   1*  1*    3   3  / KRO FOR LAYER 3
/
```

8.3.136 KRORG – END-POINT SCALING OF GRID CELL KRO(SGCR) (DRAINAGE)

Description

KRORG defines the scaling parameter for the drainage relative permeability of oil at the critical gas saturation (SGCR), for all the cells in the model via an array. The ENDSCALE keyword in the RUNSPEC section should be activated to enable end-point scaling and the use of this keyword. The SCALCERS keyword in the PROPS section defines the options used in the re-scaling process, the options are two point scaling and three point scaling.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	KRORG	KRORG is an array of positive real numbers which are greater than zero and less than or equal to 1.0, that are the assigned scaling KRORG values for each cell in the model. Repeat counts may be used, for example 50*0.850.			Taken from cell allocated relative permeability table.
		dimensionless	dimensionless	dimensionless	

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) The keyword is terminated by a "/".
3) Note this the directional independent version of the keyword used with the end-point scaling option. If directional end-point scaling has been activated then the KRORGX±, KRORGY± and KRORGZ± series of the keywords should be used.

Table 8.79: KRORG Keyword Description

When the KRORG keyword is present in the input deck then the scaling matches the relative permeability at the critical saturation of the displacing phase.

If three point scaling option has been selected via the SCALECRS keyword in the PROPS section the critical displacing phase is defined as:

No	Keywords Present	Critical Saturation
1	KRORW	$S_{critical}$ = 1.0 – SWCR - SGL
2	KRORG	$S_{critical}$ = 1.0 – SGCR - SWL

Table 8.80: Critical Displacement Relationships

End-point scaling allows the entered relative permeability functions to be re-scaled based on the saturation end-points allocated to each cell by the SWL, SWCR, SWU, SGL, SGCR, SGU, SOWCR, and SOGCR saturation grid arrays for the saturation end-points, In addition end-point scaling may be directional dependent in which case the directional dependent versions of the aforementioned arrays should be used, that is SWUX, SWUY and SWUZ instead of SWU, There is also the facility to make the directional end-point scaling reversible or non-reversible and if the non-reversible option is selected the non-reversible versions of the aforementioned arrays should be used, that is SWUX, SWUX-, SWUY, SWUY-, SWUZ and SWUZ-, instead of the SWU keyword.

End-point scaling also allows the entered relative permeability functions to be scale on the relative permeability values using the KRG, KRGR, KRORG, KRORGRG, KRORGRW, KRW and KRWR relative permeability grid cell arrays for the relative permeability end-point data. In addition end-point scaling may be directional dependent in which case the directional dependent versions of the aforementioned arrays should be used, that is KRORGX, KRORGY and KRORGZ instead of KRORG, There is also the facility to make the directional end-point scaling reversible or non-reversible and if the non-reversible option is selected the non-reversible versions of the aforementioned arrays should be used, that is KRORGX, KRORGX-, KRORGY, KRORGY-, KRORGZ and KRORGZ-, instead of the KRORG keyword.

If the hysteresis model option has been activated on the SATOPTS keyword in the RUNSPEC section, then the equivalent imbibition arrays suffixed with the letter I, for example IKRORG, can be used to define the KRORG for the relative permeability imbibition tables.

Examples

The first example defines an input box for the whole grid and for layers one to three, for layer one KRORG is set equal to 0.755, for layer two KRORG equals 0.775, and for layer three KRORG equals 0.800.

```
--
--       DEFINE INPUT BOX FOR EDITING INPUT ARRAYS (NX=100, NY=100)
--
--       ---------- BOX ---------
--       I1  I2   J1  J2   K1  K2
BOX
         1*  1*   1*  1*   1   3                          / DEFINE BOX AREA
--
--       SET KRORG VALUES FOR THREE LAYERS IN THE MODEL
--
KRORG
         10000*0.755  10000*0.775  10000*0.800               /
--
--       DEFINE END OF INPUT BOX EDITING OF INPUT ARRAYS
--
ENDBOX
```

The next example does exactly the same thing using the EQUALS keyword instead.

```
--
--       ARRAY       CONSTANT     ---------- BOX ---------
--                                I1  I2   J1  J2   K1  K2
EQUALS
         KRORG       0.7550       1*  1*   1*  1*    1   1  / KRORG FOR LAYER 1
         KRORG       0.7750       1*  1*   1*  1*    2   2  / KRORG FOR LAYER 2
         KRORG       0.8000       1*  1*   1*  1*    3   3  / KRORG FOR LAYER 3
/
```

8.3.137 KRORW – END-POINT SCALING OF GRID CELL KRO(SWCR) (DRAINAGE)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

KRORW defines the scaling parameter for the drainage relative permeability of oil at the critical water saturation (SWCR), for all the cells in the model via an array. The ENDSCALE keyword in the RUNSPEC section should be activated to enable end-point scaling and the use of this keyword. The SCALECRS keyword in the PROPS section defines the options used in the re-scaling process, the options are two point scaling and three point scaling.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	KRORW	KRORW is an array of positive real numbers which are greater than zero and less than or equal to 1.0, that are the assigned scaling KRORW values for each cell in the model. Repeat counts may be used, for example 50*0.850			Taken from cell allocated relative permeability table.
		dimensionless	dimensionless	dimensionless	

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) The keyword is terminated by a "/".
3) Note this the directional independent version of the keyword used with the end-point scaling option. If directional end-point scaling has been activated then the KRORWX±, KRORWY± and KRORWZ± series of the keywords should be used.

Table 8.81: KRORW Keyword Description

When the KRORW keyword is present in the input deck then the scaling matches the relative permeability at the critical saturation of the displacing phase.

If three point scaling option has been selected via the SCALECRS keyword in the PROPS section the critical displacing phase is defined as:

No	Keywords Present	Critical Saturation
1	KRORW	$S_{critical}$ = 1.0 – SWCR - SGL
2	KRORG	$S_{critical}$ = 1.0 – SGCR - SWL

Table 8.82: Critical Displacement Relationships

End-point scaling allows the entered relative permeability functions to be re-scaled based on the saturation end-points allocated to each cell by the SWL, SWCR, SWU, SGL, SGCR, SGU, SOWCR, and SOGCR saturation grid arrays for the saturation end-points, In addition end-point scaling may be directional dependent in which case the directional dependent versions of the aforementioned arrays should be used, that is SWUX, SWUY and SWUZ instead of SWU, There is also the facility to make the directional end-point scaling reversible or non-reversible and if the non-reversible option is selected the non-reversible versions of

the aforementioned arrays should be used, that is SWUX, SWUX-, SWUY, SWUY-, SWUZ and SWUZ-, instead of the SWU keyword.

End-point scaling also allows the entered relative permeability functions to be scale on the relative permeability values using the KRG, KRGR, KRORW, KRORWRG, KRORWRW, KRW and KRWR relative permeability grid cell arrays for the relative permeability end-point data. In addition end-point scaling may be directional dependent in which case the directional dependent versions of the aforementioned arrays should be used, that is KRORWX, KRORWY and KRORWZ instead of KRORW, There is also the facility to make the directional end-point scaling reversible or non-reversible and if the non-reversible option is selected the non-reversible versions of the aforementioned arrays should be used, that is KRORWX, KRORWX-, KRORWY, KRORWY-, KRORWZ and KRORWZ-, instead of the KRORW keyword.

If the hysteresis model option has been activated on the SATOPTS keyword in the RUNSPEC section, then the equivalent imbibition arrays suffixed with the letter I, for example IKRORW, can be used to define the KRORW for the relative permeability imbibition tables.

Examples

The first example defines an input box for the whole grid and for layers one to three, for layer one KRORW is set equal to 0.755, for layer two KRORW equals 0.775, and for layer three KRORW equals 0.800.

```
--
--       DEFINE INPUT BOX FOR EDITING INPUT ARRAYS (NX=100, NY=100)
--
--       ---------- BOX ---------
--       I1  I2   J1  J2   K1  K2
BOX
         1*  1*   1*  1*   1   3                          / DEFINE BOX AREA
--
--       SET KRORW VALUES FOR THREE LAYERS IN THE MODEL
--
KRORW
         10000*0.755  10000*0.775  10000*0.800              /
--
--       DEFINE END OF INPUT BOX EDITING OF INPUT ARRAYS
--
ENDBOX
```

The next example does exactly the same thing using the EQUALS keyword instead.

```
--
--       ARRAY       CONSTANT     ---------- BOX ---------
--                                I1  I2   J1  J2   K1  K2
EQUALS
         KRORW       0.7550       1*  1*   1*  1*    1   1  / KRORW FOR LAYER 1
         KRORW       0.7750       1*  1*   1*  1*    2   2  / KRORW FOR LAYER 2
         KRORW       0.8000       1*  1*   1*  1*    3   3  / KRORW FOR LAYER 3
/
```

8.3.138 KRW – END-POINT SCALING OF GRID CELL KRW(SW =1.0) (DRAINAGE)

Description

KRW defines the scaling parameter at the maximum drainage water relative permeability value (SWU), that is for Sw = 1.0, for all the cells in the model via an array. The ENDSCALE keyword in the RUNSPEC section should be activated to enable end-point scaling and the use of this keyword. The SCALCERS keyword in the PROPS section defines the options used in the re-scaling process, the options are two point scaling and three point scaling.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	KRW	KRW is an array of positive real numbers which are greater than zero and less than or equal to 1.0, that are the assigned scaling KRW values for each cell in the model. Repeat counts may be used, for example 50*1.000.			Taken from cell allocated relative permeability table.
		dimensionless	dimensionless	dimensionless	

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) The keyword is terminated by a "/".
3) Note this the directional independent version of the keyword used with the end-point scaling option. If directional end-point scaling has been activated then the KRWX±, KRWY± and KRWZ± series of the keywords should be used.

Table 8.83: KRW Keyword Description

For the two point scaling option and for the KRWR water relative permeability array NOT present in the input deck the krw value for a grid block is scaled by:

$$k_{rw} = k_{rw\,TABLE}\left(\frac{KRW}{k_{rw\,TABLE-MAX}}\right) \tag{8.65}$$

Where:

krw = the resulting KRW value for a grid cell.
KRW = the scaling water relative permeability value from the KRW array for a given cell.
$k_{rw\,TABLE}$ = the water relative permeability from a grid block's oil relative permeability table at the grid blocks water saturation.
$k_{rw\,TABLE-MAX}$ = the maximum water relative permeability from a grid block's water relative table, that is at the maximum water saturation.

If the KRWR keyword is present in the input deck then the scaling matches the relative permeability at the critical saturation of the displacing phase.

If three point scaling option has been selected via the SCALECRS keyword in the PROPS section the critical displacing phase is defined as:

No	Phases Present	Critical Saturation
1	Gas-Oil	$S_{critical}$ = 1.0 – SOWCR - SGL
2	Gas-Oil-Water	$S_{critical}$ = 1.0 – SOWCR - SGL
3	Gas-Water	$S_{critical}$ = 1.0 – SGCR

Table 8.84: Critical Displacement Relationships

End-point scaling allows the entered relative permeability functions to be re-scaled based on the saturation end-points allocated to each cell by the SWL, SWCR, SWU, SGL, SGCR, SGU, SOWCR, and SOGCR saturation grid arrays for the saturation end-points, In addition end-point scaling may be directional dependent in which case the directional dependent versions of the aforementioned arrays should be used, that is SWUX, SWUY and SWUZ instead of SWU, There is also the facility to make the directional end-point scaling reversible or non-reversible and if the non-reversible option is selected the non-reversible versions of the aforementioned arrays should be used, that is SWUX, SWUX-, SWUY, SWUY-, SWUZ and SWUZ-, instead of the SWU keyword.

End-point scaling also allows the entered relative permeability functions to be scale on the relative permeability values using the KRG, KRGR, KRW, KRORG, KRORW, KRW and KRWR relative permeability grid cell arrays for the relative permeability end-point data. In addition end-point scaling may be directional dependent in which case the directional dependent versions of the aforementioned arrays should be used, that is KRWX, KRWY and KRWZ instead of KRW, There is also the facility to make the directional end-point scaling reversible or non-reversible and if the non-reversible option is selected the non-reversible versions of the aforementioned arrays should be used, that is KRWX, KRWX-, KRWY, KRWY-, KRWZ and KRWZ-, instead of the KRW keyword.

If the hysteresis model option has been activated on the SATOPTS keyword in the RUNSPEC section, then the equivalent imbibition arrays suffixed with the letter I, for example IKRW, can be used to define the KRW for the relative permeability imbibition tables.

Examples

The first example defines an input box for the whole grid and for layers one to three, for layer one KRW is set equal to 0.855, for layer two KRW equals 0.875, and for layer three KRW equals 0.900.

```
--
--       DEFINE INPUT BOX FOR EDITING INPUT ARRAYS (NX=100, NY=100)
--
--       ---------- BOX ---------
--       I1  I2   J1  J2   K1  K2
BOX
         1*  1*   1*  1*   1   3                          / DEFINE BOX AREA
--
--       SET KRW VALUES FOR THREE LAYERS IN THE MODEL
--
KRW
         10000*0.855  10000*0.875  10000*0.900              /
--
--       DEFINE END OF INPUT BOX EDITING OF INPUT ARRAYS
--
ENDBOX
```

The next example does exactly the same thing using the EQUALS keyword instead.

```
--        ARRAY        CONSTANT      ---------- BOX ---------
--                                   I1  I2   J1  J2   K1  K2
EQUALS
          KRW          0.8550        1*  1*   1*  1*    1   1  / KRW FOR LAYER 1
          KRW          0.8750        1*  1*   1*  1*    2   2  / KRW FOR LAYER 2
          KRW          0.9000        1*  1*   1*  1*    3   3  / KRW FOR LAYER 3
/
```

8.3.139 KRWR – END-POINT SCALING OF GRID CELL KRWR(SOWCR) (DRAINAGE)

Description

KRWR defines the scaling parameter at the drainage critical oil to water saturation value (SOWCR), for the drainage water relative permeability curve, for all the cells in the model via an array. The ENDSCALE keyword in the RUNSPEC section should be activated to enable end-point scaling and the use of this keyword. The SCALCERS keyword in the PROPS section defines the options used in the re-scaling process, the options are two point scaling and three point scaling.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	KRWR	KRWR is an array of positive real numbers which are greater than zero and less than or equal to 1.0, that are the assigned scaling KRWR values for each cell in the model. Repeat counts may be used, for example 50*1.000.			Taken from cell allocated relative permeability table.
		dimensionless	dimensionless	dimensionless	

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) The keyword is terminated by a "/".
3) Note this the directional independent version of the keyword used with the end-point scaling option. If directional end-point scaling has been activated then the KRWRX±, KRWRY± and KRWRZ± series of the keywords should be used.

Table 8.85: KRWR Keyword Description

When the KRWR keyword is present in the input deck then the scaling matches the relative permeability at the critical saturation of the displacing phase.

If three point scaling option has been selected via the SCALECRS keyword in the PROPS section the critical displacing phase is defined as:

No	Phases Present	Critical Saturation
1	Gas-Oil	$S_{critical}$ = 1.0 – SOWCR - SGL
2	Gas-Oil-Water	$S_{critical}$ = 1.0 – SOWCR - SGL
3	Gas-Water	$S_{critical}$ = 1.0 – SGCR

Table 8.86: Critical Displacement Relationships

End-point scaling allows the entered relative permeability functions to be re-scaled based on the saturation end-points allocated to each cell by the SWL, SWCR, SWU, SGL, SGCR, SGU, SOWCR, and SOGCR saturation grid arrays for the saturation end-points, In addition end-point scaling may be directional dependent in which case the directional dependent versions of the aforementioned arrays should be used, that is SWUX, SWUY and SWUZ instead of SWU, There is also the facility to make the directional end-point scaling reversible or non-reversible and if the non-reversible option is selected the non-reversible versions of

the aforementioned arrays should be used, that is SWUX, SWUX-, SWUY, SWUY-, SWUZ and SWUZ-, instead of the SWU keyword.

End-point scaling also allows the entered relative permeability functions to be scale on the relative permeability values using the KRG, KRGR, KRWR, KRORG, KRORW, KRWR and KRWRR relative permeability grid cell arrays for the relative permeability end-point data. In addition end-point scaling may be directional dependent in which case the directional dependent versions of the aforementioned arrays should be used, that is KRWRX, KRWRY and KRWRZ instead of KRWR, There is also the facility to make the directional end-point scaling reversible or non-reversible and if the non-reversible option is selected the non-reversible versions of the aforementioned arrays should be used, that is KRWRX, KRWRX-, KRWRY, KRWRY-, KRWRZ and KRWRZ-, instead of the KRWR keyword.

If the hysteresis model option has been activated on the SATOPTS keyword in the RUNSPEC section, then the equivalent imbibition arrays suffixed with the letter I, for example IKRWR, can be used to define the KRWR for the relative permeability imbibition tables.

Examples

The first example defines an input box for the whole grid and for layers one to three, for layer one KRWR is set equal to 0.755, for layer two KRWR equals 0.775, and for layer three KRWR equals 0.800.

```
--
--       DEFINE INPUT BOX FOR EDITING INPUT ARRAYS (NX=100, NY=100)
--
--       ---------- BOX ---------
--       I1  I2   J1  J2   K1  K2
BOX
         1*  1*   1*  1*   1   3                          / DEFINE BOX AREA
--
--       SET KRWR VALUES FOR THREE LAYERS IN THE MODEL
--
KRWR
         10000*0.755  10000*0.775  10000*0.800               /
--
--       DEFINE END OF INPUT BOX EDITING OF INPUT ARRAYS
--
ENDBOX
```

The next example does exactly the same thing using the EQUALS keyword instead.

```
--
--       ARRAY       CONSTANT     ---------- BOX ---------
--                                I1  I2   J1  J2   K1  K2
EQUALS
         KRWR        0.7550       1*  1*   1*  1*    1   1  / KRWR FOR LAYER 1
         KRWR        0.7750       1*  1*   1*  1*    2   2  / KRWR FOR LAYER 2
         KRWR        0.8000       1*  1*   1*  1*    3   3  / KRWR FOR LAYER 3
/
```

8.3.140 LANGMPL – DEFINE LANGMUIR PRESSURE GRID CELL MULTIPLIER

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, LANGMPL, defines the coal bed methane Langmuir Adsorption[199] pressure multiplier for each grid block, for when the Coal Bed Methane option has been activated via the COAL keyword in the RUNSPEC section. The keyword applies the multiplier to the pressure values in a cell's allocated Langmuir table when calculating a cell's adsorption capacity. See the LANGMUIR keyword in the PROPS section for specifying the Langmuir tables for the model.

Note that if he Dual Porosity model has been activated by either the DUALPORO or the DUALPERM keywords in the RUNSPEC section, then LANGMPL applies to only the matrix grid block.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

[199] *Langmuir, Irving (June 1918). "The Adsorption of Gases on Plane Surface of Glass, Mica and Platinum". The Research Laboratory of the General Electric Company. 40 (9): 1361–1402. doi:10.1021/ja02242a004*

8.3.141 LANGMUIR – LANGMUIR ADSORPTION ISOTHERM TABLES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The LANGMUIR keyword defines the coal bed methane Langmuir Adsorption Isotherms[200] tables, for when the Coal Bed Methane option has been activated via the COAL keyword in the RUNSPEC section. See the COALNUM keyword in the GRID section for allocating the Langmuir tables to the grid blocks and also the LANGMPL keyword in the PROPS section for re-scaling the pressure values in the tables that are allocated to a cell.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

[200] *Langmuir, Irving (June 1918). "The Adsorption of Gases on Plane Surface of Glass, Mica and Platinum". The Research Laboratory of the General Electric Company. 40 (9): 1361–1402. doi:10.1021/ja02242a004*

8.3.142 LANGSOLV – LANGMUIR ADSORPTION ISOTHERM SOLVENT TABLES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The LANGMUIR keyword defines the coal bed methane Langmuir Adsorption Isotherms[201] Solvent tables, for when the Coal Bed Methane option has been activated via the COAL keyword and the Solvent phase has been declared by the SOLVENT keyword in the RUNSPEC section. See the COALNUM keyword in the GRID section for allocating the Langmuir solvent tables to the grid blocks, and also the LANGMUIR keyword in the PROPS section for defining the Langmuir Adsorption Isotherm tables. Keywords COALADS and COALPP, also in the PROPS section, are used to specify the relative adsorption data in runs containing the solvent phase.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

[201] *Langmuir, Irving (June 1918). "The Adsorption of Gases on Plane Surface of Glass, Mica and Platinum". The Research Laboratory of the General Electric Company. 40 (9): 1361–1402. doi:10.1021/ja02242a004*

8.3.143 LCUNIT – DEFINE LINEAR COMBINATION RATE AND VOLUME UNITS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The LCUNIT keyword defines the units for the Linear Combination facility which allows for a linear combination of oil, gas and water rates and volumes to be used as combination targets and constraints in controlling group and well production and injection data. See also the LINCOM in the SCHEDULE section that defines the actual linear combination equation.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.144 LKRO – END-POINT SCALING OF GRID CELL KRO(SWL) (LOW SALINITY AND OIL WET)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

LKRO defines the scaling parameter for the oil relative permeability value at the connate water saturation (SWL), for all the cells in the model via an array, for when the Low Salt option and the End-point Scaling options has been activated by the LOWSALT and the ENDSCALE keywords in the RUNSPEC section. The data is used to scale the oil relative permeability in the low salinity oil wet oil relative permeability saturation tables.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.145 LKRORG – END-POINT SCALING OF GRID CELL KRO(SGCR) (LOW SALINITY AND OIL WET)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

LKRORG defines the scaling parameter for the relative permeability of oil at the critical gas saturation (SGCR), for all the cells in the model via an array, for when the Low Salt option and the End-point Scaling options has been activated by the LOWSALT and the ENDSCALE keywords in the RUNSPEC section. The data is used to scale the oil relative permeability in the low salinity oil wet oil relative permeability saturation tables.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.146 LKRORW – END-POINT SCALING OF GRID CELL KRO(SWCR) (LOW SALINITY AND OIL WET)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

LKRORW defines the scaling parameter for the relative permeability of oil at the critical water saturation (SWCR), for all the cells in the model via an array, for when the Low Salt option and the End-point Scaling options has been activated by the LOWSALT and the ENDSCALE keywords in the RUNSPEC section. The data is used to scale the oil relative permeability in the low salinity oil wet oil relative permeability saturation tables.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.147 LKRW – END-POINT SCALING OF GRID CELL KRW(SW =1.0) (LOW SALINITY AND OIL WET)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

LKRW defines the scaling parameter at the maximum oil relative permeability value (SWU), that is for Sw = 1.0, for all the cells in the model via an array, for when the Low Salt option and the End-point Scaling options has been activated by the LOWSALT and the ENDSCALE keywords in the RUNSPEC section. The data is used to scale the water relative permeability in the low salinity oil wet water relative permeability saturation tables.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.148 LKRWR – END-POINT SCALING OF GRID CELL KRWR(SOWCR) (LOW SALINITY AND OIL WET)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

LKRWR defines the scaling parameter at the maximum oil relative permeability value (SWU), that is for Sw = 1.0, for all the cells in the model via an array, for when the Low Salt option and the End-point Scaling options has been activated by the LOWSALT and the ENDSCALE keywords in the RUNSPEC section. The data is used to scale the water relative permeability in the low salinity oil wet water relative permeability saturation tables.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.149 LPCW – END-POINT SCALING OF GRID CELL WATER CAPILLARY PRESSURE (LOW SALINITY AND OIL WET)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

LPCW defines the maximum oil-water pressure values for all the cells in the model via an array, for when the Low Salt option and the End-point Scaling options has been activated by the LOWSALT and the ENDSCALE keywords in the RUNSPEC section. The keyword re-scales the oil-water capillary pressure in the low salinity oil wet capillary saturation tables from a cell's assigned saturation function by the grid block's LPCW value.

The capillary pressure for a grid block is scaled by:

$$P_c = P_{c_{TABLE}} \left(\frac{LPCW}{P_{c_{TABLE-MAX}}} \right) \tag{8.66}$$

Where:

P_c = the resulting low salt oil wet water capillary pressure for a grid cell.

$LPCW$ = the maximum capillary pressure from the LPCW array for a given cell.

$P_{c_{TABLE}}$ = the capillary pressure in the low salt oil wet capillary pressure table allocated to the grid block.

$P_{c_{TABLE-MAX}}$ = the maximum capillary pressure in the low salt oil capillary pressure table allocated to the grid block (that is at the connate water saturation).

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.150 LSALTFNC - DEFINE LOW SALT WEIGHTING FACTORS VERSUS SALT CONCENTRATION FUNCTIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The LSALTFNC keyword defines the low salt weighting factors versus salt concentration functions for when the Low Salt option has been activated by the LOWSALT keyword in the RUNSPEC section. The tables are used to modify the oil and water relative permeability saturation end-points, as well as the water-oil capillary pressure end-points, for different salt concentrations within a grid cell.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.151 LSOGCR – END-POINT SCALING GRID CELL SOGCR (LOW SALINITY AND OIL WET)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

LSOGCR defines the critical oil saturation with respect to gas ("SOGCR"), for all the cells in the model via an array, for when the Low Salt option has been selected. The data is used to scale the oil saturation in the low salinity oil wet oil-gas relative permeability saturation tables. The ENDSCALE keyword in the RUNSPEC section should be activated to enable end-point scaling and the use of this keyword. In addition, the Low Salt option should be activated by the LOWSALT keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.152 LSOWCR – END-POINT SCALING GRID CELL SOWCR (LOW SALINITY AND OIL WET)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

LSOWCR defines the critical oil saturation with respect to water ("SOWCR"), for all the cells in the model via an array, for when the Low Salt option has been selected. The data is used to scale the oil saturation in the low salinity oil wet water-oil relative permeability saturation tables. The ENDSCALE keyword in the RUNSPEC section should be activated to enable end-point scaling and the use of this keyword. In addition, the Low Salt option should be activated by the LOWSALT keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.153 LSWCR – END-POINT SCALING GRID CELL SWCR (LOW SALINITY AND OIL WET)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

LSWCR defines the critical water saturation ("SWCR"), for all the cells in the model via an array, for when the Low Salt option has been selected. The data is used to scale the water saturation in the low salinity oil wet water-oil relative permeability saturation tables. The ENDSCALE keyword in the RUNSPEC section should be activated to enable end-point scaling and the use of this keyword. In addition, the Low Salt option should be activated by the LOWSALT keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.154 LSWL – END-POINT SCALING GRID CELL SWL (LOW SALINITY AND OIL WET)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

LSWL defines the connate water saturation ("SWL"), for all the cells in the model via an array, for when the Low Salt option has been selected. The data is used to scale the water saturation in the low salinity oil wet water-oil relative permeability saturation tables, as well as the associated capillary pressure tables. The ENDSCALE keyword in the RUNSPEC section should be activated to enable end-point scaling and the use of this keyword. In addition, the Low Salt option should be activated by the LOWSALT keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.155 LSWLPC – END-POINT SCALING GRID CELL SWLPC (LOW SALINITY AND OIL WET)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

LSWLPC defines the capillary pressure connate water saturation ("SWLPC"), for all the cells in the model via an array, for when the Low Salt option has been selected. The data is used to scale the water saturation in the low salinity oil wet water-oil capillary pressure tables. The ENDSCALE keyword in the RUNSPEC section should be activated to enable end-point scaling and the use of this keyword. In addition, the Low Salt option should be activated by the LOWSALT keyword in the RUNSPEC section.

Note the keyword only applies the scaling to the capillary pressures tables, unlike the LSWL keyword that applies the scaling to both the capillary pressure and relative permeability tables.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.156 LSWU – END-POINT SCALING GRID CELL SWU (LOW SALINITY AND OIL WET)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

LSWU defines the maximum water saturation("SWU"), for all the cells in the model via an array, for when the Low Salt option has been selected. The data is used to scale the water saturation in the low salinity oil wet water-oil relative permeability saturation tables, as well as the associated capillary pressure tables. The ENDSCALE keyword in the RUNSPEC section should be activated to enable end-point scaling and the use of this keyword. In addition, the Low Salt option should be activated by the LOWSALT keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.157 LWKRO – END-POINT SCALING OF GRID CELL KRO(SWL) (LOW SALINITY AND WATER WET)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

LWKRO defines the scaling parameter for the oil relative permeability value at the connate water saturation (SWL), for all the cells in the model via an array, for when the Low Salt and Surfactant Wettability options have been selected. The data is used to scale the oil relative permeability in the low salinity water wet oil relative permeability saturation tables. The ENDSCALE keyword in the RUNSPEC section should be activated to enable end-point scaling and the use of this keyword. In addition the Low Salt option should be activated by the LOWSALT keyword in the RUNSPEC section and the Surfactant Wettability option activated by the SURFACT or SURFACTW keywords, which are also in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.158 LWKRORG – END-POINT SCALING OF GRID CELL KRO(SGCR) (LOW SALINITY AND WATER WET)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

LWKRORG defines the scaling parameter for the relative permeability of oil at the critical gas saturation (SGCR), for all the cells in the model via an array, for when the Low Salt and Surfactant Wettability options have been selected. The data is used to scale the oil relative permeability in the low salinity water wet oil relative permeability saturation tables. The ENDSCALE keyword in the RUNSPEC section should be activated to enable end-point scaling and the use of this keyword. In addition the Low Salt option should be activated by the LOWSALT keyword in the RUNSPEC section and the Surfactant Wettability option activated by the SURFACT or SURFACTW keywords, which are also in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.159 LWKRORW – END-POINT SCALING OF GRID CELL KRO(SWCR) (LOW SALINITY AND WATER WET)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

LWKRORW defines the scaling parameter for the relative permeability of oil at the critical water saturation (SWCR), for all the cells in the model via an array, for when the Low Salt and Surfactant Wettability options have been selected. The data is used to scale the oil relative permeability in the low salinity water wet oil relative permeability saturation tables. The ENDSCALE keyword in the RUNSPEC section should be activated to enable end-point scaling and the use of this keyword. In addition the Low Salt option should be activated by the LOWSALT keyword in the RUNSPEC section and the Surfactant Wettability option activated by the SURFACT or SURFACTW keywords, which are also in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.160 LWKRW – END-POINT SCALING OF GRID CELL KRW(SW =1.0) (LOW SALINITY AND WATER WET)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

LWKRW defines the scaling parameter at the maximum water relative permeability value (SWU), that is for Sw = 1.0, for all the cells in the model via an array, for when the Low Salt and Surfactant Wettability options have been selected. The data is used to scale the water relative permeability in the low salinity water wet water relative permeability saturation tables. The ENDSCALE keyword in the RUNSPEC section should be activated to enable end-point scaling and the use of this keyword. In addition the Low Salt option should be activated by the LOWSALT keyword in the RUNSPEC section and the Surfactant Wettability option activated by the SURFACT or SURFACTW keywords, which are also in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.161 LWKRWR – END-POINT SCALING OF GRID CELL KRWR(SW=1.0) (LOW SALINITY AND WATER WET)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

LWKRWR defines the scaling parameter at the critical oil to water saturation value (SOWCR), for the water relative permeability curve, for all the cells in the model via an array, and for when the Low Salt and Surfactant Wettability options have been selected. The data is used to scale the water relative permeability in the low salinity water wet water relative permeability saturation tables. The ENDSCALE keyword in the RUNSPEC section should be activated to enable end-point scaling and the use of this keyword. In addition the Low Salt option should be activated by the LOWSALT keyword in the RUNSPEC section and the Surfactant Wettability option activated by the SURFACT or SURFACTW keywords, which are also in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.162 LWPCW – END-POINT SCALING OF GRID CELL WATER CAPILLARY PRESSURE (LOW SALINITY AND WATER WET)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

LWPCW defines the maximum water-oil pressure values for all the cells in the model via an array, for when the Low Salt and Surfactant Wettability options have been selected. The ENDSCALE keyword in the RUNSPEC section should be activated to enable end-point scaling and the use of this keyword. In addition the Low Salt option should be activated by the LOWSALT keyword in the RUNSPEC section and the Surfactant Wettability option activated by the SURFACT or SURFACTW keywords, which are also in the RUNSPEC section. The keyword re-scales the oil-water capillary pressure in the low salinity water wet capillary saturation tables from the cell's assigned saturation function by the grid block's LWPCW value.

The capillary pressure for a grid block is scaled by:

$$P_c = P_{c_{TABLE}} \left(\frac{HWPCW}{P_{c_{TABLE-MAX}}} \right) \tag{8.67}$$

Where:

P_c = the resulting high salinity water wet water capillary pressure for a grid cell.

$HWPCW$ = the maximum capillary pressure from the HWPCW array for a given cell.

$P_{c_{TABLE}}$ = the capillary pressure in the high salinity water wet water capillary pressure table allocated to the grid block.

$P_{c_{TABLE-MAX}}$ = the maximum capillary pressure in the high salinity water wet water capillary pressure table allocated to the grid block (that is at the connate water saturation).

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.163 LWSOGCR – END-POINT SCALING GRID CELL SOGCR (LOW SALINITY AND WATER WET)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

LWSOGCR defines the critical oil saturation with respect to gas ("SOGCR"), for all the cells in the model via an array, for when the Low Salt and Surfactant Wettability options have been selected. The data is used to scale the oil saturation in the low salinity water wet oil-gas relative permeability saturation tables. The ENDSCALE keyword in the RUNSPEC section should be activated to enable end-point scaling and the use of this keyword. In addition, the Low Salt option should be activated by the LOWSALT keyword in the RUNSPEC section and the Surfactant Wettability option activated by the SURFACT or SURFACTW keywords, which are also in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.164 LWSOWCR – END-POINT SCALING GRID CELL SOWCR (LOW SALINITY AND WATER WET)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

LWSOWCR defines the critical oil saturation with respect to water ("SOWCR"), for all the cells in the model via an array, for when the Low Salt and Surfactant Wettability options have been selected. The data is used to scale the oil saturation in the low salinity water wet water-oil relative permeability saturation tables. The ENDSCALE keyword in the RUNSPEC section should be activated to enable end-point scaling and the use of this keyword. In addition, the Low Salt option should be activated by the LOWSALT keyword in the RUNSPEC section and the Surfactant Wettability option activated by the SURFACT or SURFACTW keywords, which are also in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.165 LWSWCR – END-POINT SCALING GRID CELL SWCR (LOW SALINITY AND WATER WET)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

LWSWCR defines the critical water saturation ("SWCR"), for all the cells in the model via an array, for when the Low Salt and Surfactant Wettability options have been selected. The data is used to scale the water saturation in the low salinity water wet water-oil relative permeability saturation tables. The ENDSCALE keyword in the RUNSPEC section should be activated to enable end-point scaling and the use of this keyword. In addition, the Low Salt option should be activated by the LOWSALT keyword in the RUNSPEC section and the Surfactant Wettability option activated by the SURFACT or SURFACTW keywords, which are also in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.166 LWSWL – END-POINT SCALING GRID CELL SWL (LOW SALINITY AND WATER WET)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

LWSWL defines the connate water saturation ("SWL"), for all the cells in the model via an array, for when the Low Salt and Surfactant Wettability options have been selected. The data is used to scale the water saturation in the low salinity water wet water-oil relative permeability saturation tables, as well as the associated capillary pressure tables. The ENDSCALE keyword in the RUNSPEC section should be activated to enable end-point scaling and the use of this keyword. In addition, the Low Salt option should be activated by the LOWSALT keyword in the RUNSPEC section and the Surfactant Wettability option activated by the SURFACT or SURFACTW keywords, which are also in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.167 LWSWLPC – END-POINT SCALING GRID CELL SWLPC (LOW SALINITY AND WATER WET)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

LWSWLPC defines the capillary pressure connate water saturation ("SWLPC"), for all the cells in the model via an array, for when the Low Salt and Surfactant Wettability options have been selected. The data is used to scale the water saturation in the low salinity water wet water-oil capillary pressure tables. The ENDSCALE keyword in the RUNSPEC section should be activated to enable end-point scaling and the use of this keyword. In addition, the Low Salt option should be activated by the LOWSALT keyword in the RUNSPEC section and the Surfactant Wettability option activated by the SURFACT or SURFACTW keywords, which are also in the RUNSPEC section.

Note the keyword only applies the scaling to the capillary pressures tables, unlike the LWSWL keyword that applies the scaling to both the capillary pressure and relative permeability tables.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.168 LWSWU – END-POINT SCALING GRID CELL SWU (LOW SALINITY AND WATER WET)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

LWSWU defines the maximum water saturation("SWU"), for all the cells in the model via an array, for when the Low Salt and Surfactant Wettability options have been selected. The data is used to scale the water saturation in the low salinity water wet water-oil relative permeability saturation tables, as well as the associated capillary pressure tables. The ENDSCALE keyword in the RUNSPEC section should be activated to enable end-point scaling and the use of this keyword. In addition, the Low Salt option should be activated by the LOWSALT keyword in the RUNSPEC section and the Surfactant Wettability option activated by the SURFACT or SURFACTW keywords, which are also in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.169 MASSFLOW – DEFINE RIVER MASS FLOW VERSUS TIME TABLES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The MASSFLOW keyword defines the upstream river mass flow versus time tables for rivers, for when the River option has been activated via the RIVRDIMS keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.170 MAXVALUE – SETS A MAXIMUM VALUE FOR AN ARRAY ELEMENT

The MAXVALUE keyword sets a maximum value for the specified array or part of an array. The constant can be real or integer depending on the array type; however, the arrays that can be operated on is dependent on which section the MAXVALUE keyword is being used.

See *MAXVALUE – Sets a Maximum Value for an Array Element* in the GRID section for a full description.

8.3.171 MESSAGE – OUTPUT USER MESSAGE

The MESSAGE keyword outputs a user message to the terminal, as well as to the print (*.PRT) and debug (*.DBG) files. Note this is different to the MESSAGES keyword, that defines OPM Flows message print limits and stop limits generated by the simulator.

See *MESSAGE – Output User Message* in the GLOBAL section for a full description.

8.3.172 MESSAGES – DEFINE MESSAGE PRINT LIMITS AND STOP LIMITS

The MESSAGES keyword defines the print and stops levels for various messages. The "print limits" set the maximum number of messages that will be printed, after which no more messages will be printed and the "stop limits" terminate the run when these limits are exceeded. There are six levels of message that increase in severity from informative all the way to programming errors, as outlined in Error: Reference source not found.

See *MESSAGES – Define Message Print Limits and Stop Limits* in the GLOBAL section for a full description.

8.3.173 MINVALUE – SET A MINIMUM VALUE FOR AN ARRAY ELEMENT

The MINVALUE keyword sets a minimum value for the specified array or part of an array. The constant can be real or integer depending on the array type; however, the arrays that can be operated on is dependent on which section the MINVALUE keyword is being used.

See *MINVALUE – Set a Minimum Value for an Array Element* in the GRID section for a full description.

8.3.174 MICPPARA – DEFINE MICROBIALLY INDUCED CALCITE PRECIPITATION PARAMETERS

Description

The MICPPARA keyword defines the model parameters for when the MICP model has been activated via the MICP keyword in the RUNSPEC section. In addition to the MICP keyword the water phase should be made active via the WATER keyword in the RUNSPEC section. See Landa-Marbán et al[202] and [203] for further information on the model.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	DENBIO	A real positive value that defines the density of the biofilm.			
		lb/rft^3 2.18	kg/rm^3 35.0	gm/rcc 0.035	Defined
2	DENCAL	A real positive value that sets the calcite density.			
		lb/rft^3 169.18	kg/rm^3 2710.0	gm/rcc 2.710	Defined
3	DETRAT	DETRAT is a real positive value that stipulates the biofilm detachment rate.			
		feet/cP 8.53 x 10^{-13}	m/cP 2.6 x 10^{-1}3	cm/cP 2.6 x 10^{-1}1	Defined
4	CRIPOR	CRIPOR is a real positive value that defines the critical porosity., the value of porosity that equates to the minimum permeability MINPER.			
		dimensionless	dimensionless	dimensionless	0.10
5	FITFAC	A real positive value that declares the exponent in the porosity-permeability relationship. $$K = k_o\left[\left(\frac{\phi-\phi_{crit}}{\phi_0-\phi_{crit}}\right)^{\alpha} + k_{min}\right]\frac{k_o}{k_o+k_{min}}$$ Where α is FITFAC in the above equation. See Landa-Marbán et al202 and [203] for details.			
		dimensionless	dimensionless	dimensionless	3.0
6	HVEOXY	HVEOXY is a real positive value that stipulates the half-velocity coefficient for oxygen.			Defined

[202] *Landa-Marbán, D., Tveit, S., Kumar, K., Gasda, S.E., 2021. Practical approaches to study microbially induced calcite precipitation at the eld scale. Int. J. Greenh. Gas Control 106, 103256. https://doi.org/10.1016/j.ijggc.2021.103256.*

[203] *Landa-Marbán, D., Kumar, K., Tveit, S., Gasda, S.E., 2021. Numerical studies of CO2 leakage remediation by micp-based plugging technology. In: Røkke, N.A. and Knuutila, H.K. (Eds) Short Papers from the 11th International Trondheim CCS conference, ISBN: 978-82-536-1714-5, 284-290.*

No.	Name	Description			Default
		Field	Metric	Laboratory	
		lb/rft^3 1.25 x 10^{-6}	kg/rm^3 2.0 x 10^{-5}	gm/rcc 2.0 x 10^{-8}	
7	HVEURE	A real positive value that defines the half-velocity coefficient for urea.			
		lb/rft^3 1.33	kg/rm^3 21.3	gm/rcc 0.213	Defined
8	MGRRAT	MGRRAT is a real positive value that sets the maximum specific growth rate.			
		1/days 3.6	1/days 3.6	1/hours 0.15	Defined
9	MOXCON	A real positive value that stipulates the maximum injected oxygen concentration.			
		lb/stb 2.5 x 10^{-3}	kg/sm^3 0.04	gm/scc 4.0 x 10^{-5}	Defined
10	MURCON	MURCON is a real positive value that defines the maximum injected urea concentration.			
		lb/stb 3.75	kg/sm^3 60.0	gm/scc 0.060	Defined
11	MURUTI	A real positive value that sets the maximum rate of urea utilization.			
		1/days 1391	1/days 1391	1/hours 57.96	Defined
12	MIARAT	MIARAT is a real positive value that declares the microbial attachment rate.			
		1/days 0.074	1/days 0.074	1/hours 0.003	Defined
13	MIDRAT	A real value that defines the microbial death rate.			
		1/days 0.028	1/days 0.0028	1/hours 0.001	Defined
14	MINPER	MINPER is a real positive value that defines the minimum permeability.			
		mD	mD	mD	1.013 x 10^{5}
15	OXCFAC	A real value that stipulates the oxygen consumption factor.			
		dimensionless	dimensionless	dimensionless	0.5
16	TOLCLO	TOLCLO is a real positive value that defines the tolerance to be used in the calculation of the clogging termination criteria.			
		dimensionless	dimensionless	dimensionless	1.0 x 10^{-4}

No.	Name	Description			Default
		Field	Metric	Laboratory	
17	YGRCOE	A real value that sets the yield growth coefficient.			
		dimensionless	dimensionless	dimensionless	0.5
Notes: 1) The keyword is terminated by a "/".					

Table 8.87: MICPPARA Keyword Description

Note

1) This is an OPM Flow specific keyword.
2) Unlike some of the other PROPS keywords that allocate fluid and rock properties to different regions of the model using the PVTNUM and SATNUM keywords in the REGIONS section, the MICPPARA keyword parameters are applied to all the cells in the model.

Example

```
--
--       DEFINE MICROBIALLY INDUCED CALCITE PRECIPITATION PARAMETERS
--
--       DEN     DEN     DET     CRI     FIT     HVE     HVE     MGR
--       BIOF    CAL     RAT     POR     FAC     OXY     URE     RAT
--       ------  ------  ------  ------  ------  ------  ------  ------  ------
--       MOX     MUR     MUR     MIA     MID     MIN     OXC     TOL     YGR
--       CON     CON     UTI     RAT     RAT     PER     FAC     CLO     COE
--       ------  ------  ------  ------  ------  ------  ------  ------  ------
MICPPARA
         35.     2710.   2.6E-13 0.1     3.      2E-5    21.3    3.6
         0.04    60.     1391.   0.074   0.028  1.013E-5 0.5     1E-4    0.5    /
```

The above example defines the 17 model parameters for the MICP model in the metric system.

8.3.175 MISC – DEFINE SOLVENT MISCIBILITY-IMMISCIBILITY TRANSFORM FUNCTIONS

Description

MISC defines the transformation between the miscible and immiscible relative permeability models, for when the MISCIBLE and SOLVENT keywords in the RUNSPEC section have been activated. The keyword can only be used with the MISCIBLE option and for when the oil, water, gas and solvent phases are active in the model.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SSOL	A columnar vector of real monotonically increasing down the column values starting from zero and terminating at one, that defines the solvent fraction with respect to the solvent and gas saturation, and is defined by: $\frac{S_s}{(S_g + S_s)}$ Where Sg is the gas saturation and Ss is the solvent saturation. Note that the first entry in the columnar vector should be zero and the last entry should be one to fully define the solvent fraction range.			
		dimensionless	dimensionless	dimensionless	None
2	MISC	A columnar vector of real equal or increasing down the column values that are greater than or equal to zero and less then one, that define the corresponding miscibility for the corresponding solvent fraction SSOL. The first entry in the columnar vector should be zero and the last entry should be one to fully define the miscible-immiscible relationship.			
		dimensionless	dimensionless	dimensionless	None

Notes:

1) The keyword is followed by NTMISC tables as declared on the MISCIBLE keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NSMISC rows as declared on the MISCIBLE keyword in the RUNSPEC section.
3) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.88: MISC Keyword Description

Example

```
--
--       SOLVENT MISCIBILITY-IMMISCIBLITY TRANSFORM TABLE
--
MISC
--       SSOL       MISC
--       FRAC       FRAC
--       -------    --------
          0.0000     0.0000
          0.2000     0.2500
          0.5000     0.7500
          1.0000     1.0000                                  / TABLE NO. 01

--       SSOL       MISC
--       FRAC       FRAC
--       -------    --------
          0.0000     0.0000
          0.3000     0.2500
          0.6000     1.0000
          1.0000     1.0000                                  / TABLE NO. 02
```

The above example defines two solvent miscible-immiscible transform tables assuming NTMISC equals two and NSMISC is greater than or equal to four on the MISCIBLE keyword in the RUNSPEC section.

8.3.176 MLANG – DEFINE LANGMUIR MAXIMUM GAS CONCENTRATION FOR ALL GRID CELLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, MLANG, defines the coal bed methane Langmuir Adsorption[204] maximum gas concentration for each grid cell used to scale the Langmuir isotherm table allocated to the cell, for when the Coal Bed Methane option has been activated via the COAL keyword in the RUNSPEC section. See the LANGMUIR keyword in the PROPS section for specifying the Langmuir tables for the model.

Note that if he Dual Porosity model has been activated by either the DUALPORO or the DUALPERM keywords in the RUNSPEC section, then MLANG applies to only the matrix grid block.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

[204] *Langmuir, Irving (June 1918). "The Adsorption of Gases on Plane Surface of Glass, Mica and Platinum". The Research Laboratory of the General Electric Company. 40 (9): 1361–1402. doi:10.1021/ja02242a004*

8.3.177 MLANGSLV – DEFINE LANGMUIR MAXIMUM SOLVENT CONCENTRATION FOR ALL GRID CELLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, MLANGSLV, defines the coal bed methane Langmuir Adsorption[205] maximum solvent concentration for each grid cell used to scale the Langmuir isotherm solvent table allocated to the cell, for when the Coal Bed Methane option has been activated via the COAL keyword in the RUNSPEC section. In addition, the Solvent phase must have been declared by the SOLVENT keyword in the RUNSPEC section. See the COALNUM keyword in the GRID section for allocating the Langmuir solvent tables to the grid blocks, and also the LANGMUIR keyword in the PROPS section for defining the Langmuir Adsorption Isotherm tables. Keywords COALADS and COALPP, also in the PROPS section, are used to specify the relative adsorption data in runs containing the solvent phase.

Note that if he Dual Porosity model has been activated by either the DUALPORO or the DUALPERM keywords in the RUNSPEC section, then MLANGSLV applies to only the matrix grid block.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

[205] *Langmuir, Irving (June 1918). "The Adsorption of Gases on Plane Surface of Glass, Mica and Platinum". The Research Laboratory of the General Electric Company. 40 (9): 1361–1402. doi:10.1021/ja02242a004*

8.3.178 MSFN – MISCIBLE NORMALIZED RELATIVE PERMEABILITY TABLES

Description

The MSFN keyword defines the miscible normalized relative permeability tables for when the MISCIBLE and or SOLVENT options have been activated in the RUNSPEC section using the respective keyword. The MISCIBLE keyword invokes a three component formulation (oil, water and solvent gas or an oil, water and solvent oil). Whereas the SOLVENT keyword results in a four component model (oil, water and gas plus a solvent). This keyword should only be used if the MISCIBLE and or SOLVENT options have been activated.

No.	Name	Description	Default
1	SGAS	A columnar vector of real monotonically increasing down the column values starting from zero and terminating at one, that defines the gas plus solvent saturation.	None
2	KRSG	A columnar vector of real values that are either equal or increasing down the column and that are greater than or equal to zero and less than or equal to one that defines the gas plus solvent relative permeability multiplier.	None
3	KRO	A columnar vector of real values that are either equal or increasing down the column and that are greater than or equal to zero and less than or equal to one that defines the oil relative permeability multiplier.	None

Notes:

1) The keyword is followed by NTSFUN tables as declared on the TABDIMS keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NSSFUN rows as declared on the TABDIMS keyword in the RUNSPEC section.
3) Each table is terminated by "/" and there is no "/" terminator for the keyword.

Table 8.89: MSFN Keyword Description

Example

```
--
--        MISCIBLE NORMALIZED RELATIVE PERMEABILITY TABLES
--
MSFN
--        SGAS        KRSG          KRO
--        FRAC
--        --------   --------      --------
           0.0000     0.0000        1.0000
           1.0000     1.0000        0.0000                    / TABLE NO. 01

--        SGAS        KRSG          KRO
--        FRAC
--        --------   --------      --------
           0.0000     0.0000        1.0000
           0.2000     0.2000        0.8000
           0.4000     0.3000        0.7000
           0.6000     0.4000        0.6000
           0.8000     0.5000        0.4000
           1.0000     1.0000        0.0000                    / TABLE NO. 02
```

The above example defines two MSN tables for use the MISCIBLE and SOLVENT options.

8.3.179 MULTIPLY – MULTIPLY A SPECIFIED ARRAY BY A CONSTANT

The MULTIPLY keyword multiplies a specified array or part of an array by a constant. The constant can be real or integer depending on the array type; however, the arrays that can be operated on is dependent on which section the keyword is being used.

See *MULTIPLY – Multiply a Specified Array by a Constant* in the GRID section for a full description.

8.3.180 MULTIREG – MULTIPLY AN ARRAY BY A CONSTANT BASED ON A REGION NUMBER

The MULTIREG keyword multiplies an array or part of an array by a constant for cells with a specific region number. The region number array can be FLUXNUM, MULTNUM or OPERNUM and these arrays must be defined and be available before the MULTIREG keyword is read by the simulator. The constant can be real or integer depending on the property array type; however, the property arrays that can be operated on is dependent on which section the MULTIREG keyword is being used.

See *MULTIREG – Multiply an Array by a Constant based on a Region Number* in the GRID section for a full description.

8.3.181 NCOMPS – CONFIRM NUMBER OF COMPOSITIONAL COMPONENTS

Description

This keyword confirms the maximum number of compositional components in the model, as such it should have the same number of components as that declared via the COMPS keyword in the RUNSPEC section. The keyword should only be used if the CO2STORE keyword and either the GASWAT or the GAS and WATER keywords in the RUNSPEC section, have also be activated for the gas-water two component model.

Note

This is an OPM Flow keyword used with OPM Flow's CO2STORE and GASWAT keywords in the RUNSPEC section, and should not be confused with the more general version of the NCOMPS keyword used in the commercial compositional simulator.

Secondly, although OPM Flow parses the keyword, the simulator currently ignores the data for this keyword.

No.	Name	Description	Default
1	NCOMPS	A positive integer defining the maximum number of compositional components in the model. Secondly the number of components must be the same as that enter via the COMPS keyword in the RUNSPEC section. Only the default value of two is currently supported by OPM Flow.	2
Notes: 1) The keyword is terminated by a "/".			

Table 8.90: NCOMPS Keyword Description

Example

The following example defines how to confirm a two component formulation to be used with the CO2STORE and GASWAT options.

```
--
--        CONFIRM NUMBER OF COMPOSITIONAL COMPONENTS (OPM FLOW KEYWORD)
--
NCOMPS
         2                                                                  /
```

8.3.182 NOECHO – DEACTIVATE ECHOING OF USER INPUT FILES TO THE PRINT FILE

Turns off echoing of all the input files to the print file. Note by default echoing of the inputs files is active. but can subsequently be switched off by the NOECHO activation keyword.

See *NOECHO – Deactivate Echoing of User Input Files to the Print File* in the GLOBAL section for a full description.

8.3.183 NOWARN – DEACTIVATE WARNING MESSAGES

Turns off warning messages to be printed to the print file; note that this keyword is deactivated by default and can subsequently be switched off by the WARN activation keyword. The warning messages may be turned on and off using keywords WARN and NOWARN.

See *NOWARN – Deactivate Warning Messages* in the GLOBAL section for a full description.

8.3.184 NOWARNEP – DEACTIVATE END-POINT SCALING WARNING MESSAGES

Description

The NOWARNEP keyword deactivates the writing out of warning messages associated with checking the consistency of saturation table end-points; however error messages are still reported by the simulator.

Hence, OPM Flow ignores this keyword but it is documented here for completeness.

There is no data required for this keyword and there is no terminating "/" for this keyword.

Example

```
--
--        DEACTIVATE END-POINT SCALING WARNING MESSAGES
--
NOWARNEP
```

The above example switches off the writing out of warning messages associated with checking the consistency of saturation table end-points;

8.3.185 OILDENT – DEFINE OIL DENSITY TEMPERATURE COEFFICIENTS

Description

OILDENT defines the oil density as a function of temperature coefficients for when OPM Flow's thermal option has been activated by the THERMAL keyword in the RUNSPEC. Note this is an OPM Flow keyword used with OPM Flow's black-oil thermal model that is not available in the commercial simulator's black-oil thermal formulation.

This keyword can only be used if OPM Flow's thermal option has been activated by the THERMAL keyword in the RUNSPEC section. Note this is different to the commercial simulator that uses the TEMP keyword in the RUNSPEC section to activate the black-oil thermal model.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	TEMP	TEMP is a real positive value greater than zero that defines the absolute reference temperature used with TEXP1 and TEXP2 to estimate the change in oil density with respect to temperature.			
		°R 527.67	K 293.15	K 293.15	Defined
2	TEXP1	TEXP1 is a real positive value greater than zero that defines the oil thermal expansion coefficient of the first order.			
		1/°R 1.67×10^{-4}	1/K 3.0×10^{-4}	1/K 3.0×10^{-4}	Defined
3	TEXP2	TEXP2 is a real positive value greater than zero that defines the oil thermal expansion coefficient of the second order.			
		1/°R2 9.26×10^{-7}	1/K2 3.0×10^{-6}	1/K2 3.0×10^{-6}	Defined

Notes:

1) The keyword is followed by NTPVT records as declared on the TABDIMS keyword in the RUNSPEC section.
2) Each data set is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.91: OILDENT Keyword Description

The oil density at a given pressure and temperature is calculated from its value at surface conditions and the oil shrinkage factor (the reciprocal of the oil formation volume factor) as shown in the following equation:

$$\rho_o(p,T) = \rho_o(p_s,T_s)\, b_o(p,T) \tag{8.68}$$

Where the temperature dependence of the oil shrinkage factor relative to its value at the reference temperature is calculated as shown in the following equation:

$$b_o(p,T)=\frac{b_o(p,T_{ref})}{1+c_1(T-T_{ref})+c_2(T-T_{ref})^2} \tag{8.69}$$

Where:

ρ_o = oil density
b_o = oil shrinkage factor
p = pressure
T = temperature
c_1, c_2 = thermal expansion coefficients to first and second order
s = subscript indicating surface conditions
ref = subscript indicating reference conditions

Example

The following example shows the OILDENT keyword using the default values, for when the thermal option has been activated by the THERMAL keyword in the RUNSPEC section and for when NTPVT on the TABDIMS keyword in the RUNSPEC section is set to two.

```
--
--       OIL DENSITY TEMPERATURE COEFFICIENTS (OPM FLOW THERMAL KEYWORD)
--
--       OIL         DENSITY   DENSITY
--       TEMP        COEFF1    COEFF2
--       --------    -------   -------
OILDENT
         1*          1*        1*                          / TABLE NO. 01
         1*          1*        1*                          / TABLE NO. 02
```

There is no terminating "/" for this keyword.

8.3.186 OILJT – DEFINE OIL JOULE-THOMSON COEFFICIENT

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

OILJT activates the oil Joule-Thomson effect[206] in temperature calculations, and defines the oil Joule-Thomson Coefficient ("JTC") at a given reference pressure, for when OPM Flow's thermal option has been activated by the THERMAL keyword in the RUNSPEC.

Note

This is an OPM Flow keyword used with OPM Flow's black-oil thermal model, that is not available in the commercial simulator's black-oil thermal formulation.

This keyword can only be used if OPM Flow's thermal option has been activated by the THERMAL keyword in the RUNSPEC section. Note this is different to the commercial simulator that uses the TEMP keyword in the RUNSPEC section to activate the black-oil thermal model, and does not include the Joule-Thomson effect in temperature calculations.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	PRESS	A real positive value that defines the reference pressure for the corresponding Joule-Thomson Coefficient, OILJTC.			
		psia	barsa	atma	None
2	OILJTC	OILJTC is a real positive or negative value that defines the oil phase Joule-Thomson Coefficient. If the value is defaulted (1*) or set to 0, then OILJTC is internally calculated using the thermal oil density data on the OILDENT keyword in the PROPS section. If a non-zero value is specified, then the OILJTC is assumed to be constant and equal to that value.			
		oF/psia	°C/barsa	°C/atma	0

Notes:

1) The keyword is followed by NTPVT records as declared on the TABDIMS keyword in the RUNSPEC section.
2) Each data set is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.92: OILJT Keyword Description

The Joule–Thomson effect is when a real gas, as oppose to an ideal gas, expands, resulting in the temperature of the gas dropping[207]. For liquids the effect is the opposite, that is the internal energy is transferred to kinetic energy with a corresponding increase in temperature as velocity increases.

Thermodynamically, the Joule–Thomson coefficient is defined as the isenthalpic[208] change in temperature in a fluid caused by a unitary pressure drop, as shown in the following equation:

[206] *The Joule–Thomson coefficient is defined as the change in temperature with respect to an increase in pressure at constant enthalpy.*

[207] *Natural Gas Engineering (McGraw-Hill chemical engineering series), Donald L. Katz, Robert I Lee, McGraw-Hill Education, 1990 (ISBN 0071007776, 9780071007771).*

$$\eta = \left(\frac{\partial T}{\partial P}\right) \tag{8.70}$$

Which can also express as[209]:

$$\eta = (T\alpha - 1)\frac{1}{(\rho C_p)} - \left(\frac{g}{C_p}\frac{dp}{dz}\right)^{-1} \tag{8.71}$$

Setting the gravity term, g, to zero we have:

$$\eta = (T\alpha - 1)\frac{1}{(\rho C_b)} \tag{8.72}$$

Where:

η	=	Joule–Thomson coefficient (oC/Pa),
α	=	thermal expansivity at constant pressure (1/oC),
C_p	=	specific heat at constant pressure (J/kg oC),
g	=	gravitational acceleration (m/s2)
P	=	pressure (Pa),
T	=	temperature (oC), and
z	=	height (m).

Example

The following example shows the OILJT keyword for when the thermal option has been activated by the THERMAL keyword in the RUNSPEC section, and for when NTPVT on the TABDIMS keyword in the RUNSPEC section is set equal to two.

```
--
--       OIL JOULE-THOMSON COEFFICIENT (OPM FLOW EXTENSION KEYWORD)
--
--       REF         OIL
--       PRESS       JTC
--       --------    -------
OILJT
         20.0        1*                  / TABLE NO. 01
         20.0        -0.20               / TABLE NO. 02
```

Here the first entry is defaulted, and the simulator will therefore calculate the oil JTC internally using the data on the OILDENT keyword in the PROPS section.

There is no terminating "/" for this keyword.

[208] *An isenthalpic process or isoenthalpic process, is a process that proceeds without any change in enthalpy, H; or specific enthalpy, h.*

[209] *Pippard, A.B.: Elements of Classical Thermodynamics: For Advanced Students of Physics. Cambridge University Press, Cambridge, UK (1957)*

8.3.187 OILVISCT – DEFINE OIL VISCOSITY VERSUS TEMPERATURE FUNCTIONS

Description

OILVISCT defines the oil viscosity as a function of temperature for when OPM Flow's thermal option has been activated by the THERMAL keyword in the RUNSPEC section. The reference pressure and solution gas-oil ratio of the oil for this table is given by the VISCREF keyword in the PROPS section. Note this is an OPM Flow keyword used with OPM Flow's black-oil thermal model that is not available in the commercial simulator's black-oil thermal formulation.

This keyword can only be used if OPM Flow's thermal option has been activated by the THERMAL keyword in the RUNSPEC section. Note this is different to the commercial simulator that uses the TEMP keyword in the RUNSPEC section to activate the black-oil thermal model.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	TEMP	A columnar vector of real monotonically increasing down the column values that defines the temperature values.			
		°F	°C	°C	None
2	VIS	A columnar vector of real increasing down the column values that defines the oil viscosity for the corresponding temperature values (TEMP). VIS should be given at the reference pressure and solution gas-oil ratio as defined by PRESS and RS variables on the VISCREF keyword.			
		cP	cP	cP	None

Notes:

1) The keyword is followed by NTPVT tables as declared on the TABDIMS keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NPPVT rows as declared on the TABDIMS keyword in the RUNSPEC section.
3) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.93: OILVISCT Keyword Description

There is no terminating "/" for this keyword.

Example

The following example shows the OILVISCT keyword for when the thermal option has been activated by the THERMAL keyword in the RUNSPEC section and for when NTPVT on the TABDIMS keyword in the RUNSPEC section is set equal to one.

```
--
--       OIL VISCOSITY VERSUS TEMPERATURE TABLES (OPM FLOW EXTENSION KEYWORD)
--
--       OIL         OIL
--       TEMP        VISC
--       --------    -------
OILVISCT
           100.0     0.600
           110.0     0.650
           120.0     0.680
           150.0     0.720
           165.0     0.725                                  / TABLE NO. 01
```

8.3.188 OPERATE – DEFINE MATHEMATICAL OPERATIONS ON ARRAYS

This keyword, OPERATE, defines mathematical operations on property arrays (NTG, PORO etc.) and optionally using another property array as input to the function. The keyword allows for various mathematical functions and their associated variables to be defined and applied to the selected array data. Input constants can be real or integer depending on the array type; however, the arrays that can be operated on is dependent on which section the keyword is being used.

See *OPERATE – Define Mathematical Operations on Arrays* in the GRID section for a full description.

8.3.189 OPERATER – DEFINE MATHEMATICAL OPERATIONS ON ARRAYS BY REGION

This keyword defines the mathematical operations on arrays for specific regions in the commercial simulator and is currently not supported by OPM Flow. However, similar functionality is provided by the ADD and MULTIPLY keywords.

See *OPERATER – Define Mathematical Operations on Arrays by Region* in the GRID section for a full description.

8.3.190 OVERBURD – DEFINE ROCK OVERBURDEN PRESSURE VERSUS DEPTH TABLES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The OVERBURD keyword defines the overburden pressures versus depth relationship to be applied for when the rock compaction option has been invoked by the ROCKCOMP keyword in the RUNSPEC section.

The rock compaction pore volume and transmissibility multipliers, entered via the ROCKTAB, ROCK2D and ROCK2DTR keywords, are applied to the pore pressure, unless the OVERBURD keyword is included in the input deck. When the OVERBURD keyword is present the multipliers are applied to the effective pore volume pressure, that is $P_{(effective)} = P_{(Pressure)} - P_{(overburden)}$. If the keyword is not present in the input deck then the overburden pressure is set to zero.

This keyword should only be used if compaction option has been enabled.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	DEPTH	A columnar vector of real monotonically increasing down the column values that defines the depth for corresponding overburden pressure parameter PRESS.			
		feet	m	cm	None
2	PRESS	A columnar vector of real monotonically increasing down the column values that defines the corresponding overburden pressure for the given DEPTH.			
		psia	bars	atm	None

Notes:

1) The keyword is followed by NTROCC tables as declared on the ROCKCOMP keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NPPVT rows as declared on the TABDIMS keyword in the RUNSPEC section.
3) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.94: OVERBURD Keyword Description

See also the ROCKTAB, ROCK2D, ROCK2DTR, and ROCKWNOD keywords in the PROPS section.

Examples

The example below defines three overburden tables, assuming NTROCC is equal to three on the ROCKCOMP keyword and NPPVT is greater than or equal to four on the TABDIMS keyword.

```
--
--        OVERBURDEN PRESSURE VERSUS DEPTH TABLES
--
OVERBURD
--        DEPTH     OVERBURDEN
--        FEET      PRESSURE
--        ------    ----------
          1000.0     300.000
          2000.0     600.000
          3000.0     900.000
          4000.0    1200.000                                  / TABLE N0. 01
--        DEPTH     OVERBURDEN
--        FEET      PRESSURE
--        ------    ----------
          1000.0     200.000
          2000.0     400.000
          3000.0     800.000
          4000.0    1000.000                                  / TABLE N0. 02
--        DEPTH     OVERBURDEN
--        FEET      PRESSURE
--        ------    ----------
          1000.0     400.000
          2000.0     800.000
          3000.0    1100.000
          4000.0    1500.000                                  / TABLE N0. 03
```

Note that there must be exactly NTROCC tables entered for this keyword, otherwise an error will occur.

```
--
--        ROCK COMPACTION TABLES
--
ROCKTAB
--        PRESS     PORV      TX(YZ)
--                  MULT      MULT
--        ------    ------    ------
          1000.0    0.9600    0.9650
          1500.0    0.9800    0.9850
          3000.0    0.9900    0.9950
          4500.0    1.0000    1.0000
          4750.0    1.0100    1.0100                          / TABLE NO. 01
--        PRESS     PORV      TX(YZ)
--                  MULT      MULT
--        ------    ------    ------
          1000.0    0.9600    0.9650
          1500.0    0.9800    0.9850
          3000.0    0.9900    0.9950
          4500.0    1.0000    1.0000
          4750.0    1.0100    1.0100                          / TABLE NO. 02
--        PRESS     PORV      TX(YZ)
--                  MULT      MULT
--        ------    ------    ------
          1000.0    0.9600    0.9650
          2000.0    0.9800    0.9850
          3000.0    0.9900    0.9950
          4000.0    1.0100    1.0100                          / TABLE NO. 03
```

Here ROCKTAB tables one and two are identical.

8.3.191 PCG – END-POINT SCALING OF GRID CELL MAXIMUM GAS CAPILLARY PRESSURE (DRAINAGE)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

PCG defines the maximum drainage gas-oil capillary pressure values for all the cells in the model via an array. The ENDSCALE keyword in the RUNSPEC section should be activated to enable end-point scaling and the use of this keyword. The keyword can be used with all grid types.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped. See also the IPCG keyword for the equivalent imbibition functionality.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	PCG	PCG is an array of positive real numbers assigning the maximum drainage gas-oil capillary pressure values for each cell in the model. Repeat counts may be used, for example 30*100.0.			
		psia	bars	atm	None

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.

2) If the HYSTER on the SATOPTS keyword in the RUNSPEC section has been activated to invoke hysteresis then PCG scales the drainage curve and IPCG scales the imbibition curve.

3) The keyword is terminated by a "/".

Table 8.95: PCG Keyword Description

The capillary pressure for a grid block is scaled by:

$$P_c = P_{c_{TABLE}} \left(\frac{PCG}{P_{c_{TABLE-MAX}}} \right) \quad (8.73)$$

Where:

P_c = the resulting drainage gas-oil capillary pressure for a grid cell.
PCG = the maximum capillary pressure from the PCG array for a given cell.
$P_{c_{TABLE}}$ = the capillary pressure in the drainage capillary pressure table allocated to the grid block.
$P_{c_{TABLE-MAX}}$ = the maximum capillary pressure in the drainage capillary pressure table allocated to the grid block at $S_g = 1 - S_{wco}$.

Example

```
--
--       DEFINE GRID BLOCK PCG DATA FOR ALL CELLS (BASED ON NX x NY x NZ = 300)
--
PCG
         100*50.0  100*75.0  100*125.0                                          /
```

The above example defines the PCW for 300 cells in the model as defined by the DIMENS keyword in the RUNSPEC section.

8.3.192 PCG32D – GAS-OIL CAPILLARY PRESSURE VERSUS OIL AND WATER SATURATION TABLES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, PCG32D, enables the gas-oil capillary pressure data to be entered as a function of both oil and water saturations. The keyword should be used in conjunction with the SGF32D keyword in the PROPS section. See also the PCW32D keyword in the PROPS section that provides similar functionality for the water-oil capillary pressure data.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.193 PCW – END-POINT SCALING OF GRID CELL WATER CAPILLARY PRESSURE (DRAINAGE)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

PCW defines the maximum drainage water-oil or water-gas capillary pressure values for all the cells in the model via an array. The ENDSCALE keyword in the RUNSPEC section should be activated to enable end-point scaling and the use of this keyword. The keyword can be used with all grid types.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	PCW	PCW is an array of positive real numbers assigning the maximum drainage water capillary pressure values for each cell in the model. Repeat counts may be used, for example 30*100.0.			
		psia	bars	atm	None

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.

2) If the HYSTER on the SATOPTS keyword in the RUNSPEC section has been activated to invoke hysteresis then PCW scales the drainage curve and IPCW scales the imbibition curve.

3) The keyword is terminated by a "/".

Table 8.96: PCW Keyword Description

See also the IPCW keyword for the equivalent imbibition functionality.

The capillary pressure for a grid block is scaled by:

$$P_c = P_{c_{TABLE}} \left(\frac{PCW}{P_{c_{TABLE-MAX}}} \right) \quad (8.74)$$

Where:

P_c = the resulting drainage water capillary pressure for a grid cell.

PCW = the maximum capillary pressure from the PCW array for a given cell.

$P_{c_{TABLE}}$ = the capillary pressure in the drainage capillary pressure table allocated to the grid block.

$P_{c_{TABLE-MAX}}$ = the maximum capillary pressure in the drainage capillary pressure table allocated to the grid block (that is at the connate water saturation).

Example

```
--
--        DEFINE GRID BLOCK PCW DATA FOR ALL CELLS (BASED ON NX x NY x NZ = 300)
--
PCW
          100*50.0  100*75.0  100*125.0                                          /
```

The above example defines the PCW for 300 cells in the model as defined by the DIMENS keyword in the RUNSPEC section.

8.3.194 PCW32D – WATER-OIL CAPILLARY PRESSURE VERSUS OIL AND GAS SATURATION TABLES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, PCW32D, enables the water-oil capillary pressure data to be entered as a function of both oil and gas saturations. The keyword should be used in conjunction with the SWF32D keyword in the PROPS section. See also the PCG32D keyword in the PROPS section that provides similar functionality for the gas-oil capillary pressure data.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.195 PECOEFS – DEFINE PETRO-ELASTIC MODEL COEFFICIENTS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The PECOEFS keyword defines the Petro-Elastic model coefficients.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.196 PEGTAB – PETRO-ELASTIC PRESSURE SHEAR MODULUS TABLE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The PEGTAB series of keywords define a series of coefficients of a polynomial function used in the calculation of the shear modulus in the petro-elastic model. The series of keywords consist of: PEGTAB0 PEGTAB1, PEGTAB2, PEGTAB3, PEGTAB4, PEGTAB5, PEGTAB6, and PEGTAB7.

This series of keywords are ignored by OPM Flow and have no effect on the simulation.

8.3.197 PEKTAB – PETRO-ELASTIC PRESSURE BULK MODULUS TABLE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The PEKTAB series of keywords define a series of coefficients of a polynomial function used in the calculation of the bulk modulus in the petro-elastic model. The series of keywords consist of: PEKTAB0 PEKTAB1, PEKTAB2, PEKTAB3, PEKTAB4, PEKTAB5, PEKTAB6, and PEKTAB7.

This series of keywords are ignored by OPM Flow and have no effect on the simulation.

8.3.198 PERMFACT – PERMEABILITY MULTIPLICATION FACTOR AS A FUNCTION OF POROSITY CHANGE

Description

PERMFACT defines the permeability multiplication factor due to a change in porosity. Currently the keyword is used in conjunction with OPM Flow's Salt Precipitation model, in which the pore space is reduced due to salt precipitating in the pore space, causing a reduction in porosity and an associated reduction in permeability.

Note

This is an OPM Flow specific keyword for the simulator's Salt Precipitation model that is activated by the BRINE and PRECSALT keywords and declaring that vaporized water is present in the run via the VAPWAT keyword. All three keywords are in the RUNSPEC section.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	POROFAC	A real monotonically increasing positive columnar vector that defines the porosity ($\frac{\phi}{\phi_0}$) factor for the corresponding PERMFAC vector. In the simulator's Salt Precipitation model, the maximum value of Φ is Φ0, implying a maximum value of one for POROFAC.			
		dimensionless	dimensionless	dimensionless	None
2	PERMFAC	A real positive monotonically increasing columnar vector that defines the permeability (k) multiplier associated with POROFAC and used to scale a grid block's permeability due to the reduction in pore volume caused by salt precipitation. Where: $PERMFAC = m(\phi)$ $with\ k = m(\phi)k_0$			
		dimensionless	dimensionless	dimensionless	None

Notes:

1) The keyword is followed by NTPVT tables as declared on the TABDIMS keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NPPVT rows as declared on the TABDIMS keyword in the RUNSPEC section.
3) There must be same number of entries for each column.
4) Each table is terminated by a single "/" and the keyword is terminated by a "/".

Table 8.97: PERMFACT Keyword Description

The porosity reduction is a function of the volume fraction of salt (ss) precipitated out of the vaporized water phase, that is:

$$\phi = (1 - s_s)\,\phi_o \tag{8.75}$$

The porosity and associated permeability factor data can be calculated using a permeability-porosity

relationship, for example:

$$\frac{k}{k_o} = \left(\frac{\phi - \phi_c}{\phi_o - \phi_c} \right)^{\gamma} \tag{8.76}$$

Where:

k_o and ϕ_o	=	the initial permeability and porosity,
k and ϕ	=	the actual permeability and porosity that changes due to precipitation or dissolution of the salt,
ϕ_c	=	the residual porosity at which permeability is zero, and
γ	=	a positive exponent.

See also Kozeny-Carment (extended)[210], [211] and [212] and Verma-Pruess[213] for additional functional forms that can be used to derive the tabulated data that can be entered via the PERMFACT keyword.

Example

The example below defines two PERMFACT tables assuming NTPVT equals two and NPPVT is greater than or equal to five on the TABDIMS keyword in the RUNSPEC section.

```
--
--       PERMEABILITY FACTOR REDUCTION DUE TO SALT PRECIPITATION
--       (OPM FLOW KEYWORD)
--
PERMFACT
--       PORO       PERM
--       FACTOR     FACTOR
--       -------    --------
         0.00        0.0000
         0.25        0.0625
         0.50        0.2500
         0.75        0.5625
         1.00        1.0000                    / TABLE NO. 01
--       -------    --------
         0.00        0.0000
         0.25        0.0625
         0.50        0.2500
         0.75        0.5625
         1.00        1.0000                    / TABLE NO. 02
/
```

Both tables use equation (8.76) with Φc equal to zero and ɣ equal to 2.0. Note that PERMFACT changes the permeability in all three dimensions, that is the PERMX, PERMY and PERMZ arrays are all modified.

Finally, notice that the terminating "/" for this keyword.

[210] *J. Kozeny, "Ueber kapillare Leitung des Wassers im Boden." Sitzungsber Akad. Wiss., Wien, 136(2a): 271-306, 1927.*

[211] *P.C. Carman, "Fluid flow through granular beds." Transactions, Institution of Chemical Engineers, London, 15: 150-166, 1937.*

[212] *P.C. Carman, "Flow of gases through porous media." Butterworths, London, 1956.*

[213] *Verma, A., & Pruess, K. (1988). Thermohydrological conditions and silica redistribution near high-level nuclear wastes emplaced in saturated geological formations. Journal of Geophysical Research, 93, 1159–1173.*

8.3.199 PLMIXPAR – DEFINE THE POLYMER TODD-LONGSTAFF MIXING PARAMETERS

Description

The PLMIXPAR keyword defines the Todd-Longstaff[214] mixing parameters for when the polymer option has been activated by the POLYMER keyword in the RUNSPEC section. This keyword must be present in the input deck if the POLYMER keyword has been activated.

Note that this keyword is used only for the polymer option, if the MISCIBLE keyword in the RUNSPEC section has been invoked then in addition the TLMIXPAR keyword is also required to define the Todd-Longstaff mixing parameters for the MISCIBLE option.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	PLMVIS	A real positive value that is greater than or equal to zero and less than or equal to one, that defines the viscosity Todd-Longstaff mixing parameter for each polymer region.			
		dimensionless	dimensionless	dimensionless	None

Notes:

1) The keyword is followed by NPLMIX tables as declared on the REGDIMS keyword in the RUNSPEC section, for each rock region. There should be only one row per table.

2) Each entry is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.98: PLMIXPAR Keyword Description

Example

```
--
--       POLYMER TODD-LONGSTAFF MIXING PARAMETERS
--
PLMIXPAR
--       PLM
--       VISCOS
--       -------
         0.3500                                    / TABLE NO. 01
         0.2500                                    / TABLE NO. 02
         0.6500                                    / TABLE NO. 03
```

The above example defines three polymer Todd-Longstaff mixing parameter data sets, based on the NPLMIX variable on the REGDIMS keyword in the RUNSPEC section being equal to three.

[214] *Todd, M. and Longstaff, W. "The Development, Testing and Application of a Numerical Simulator for Predicting Miscible Flood Performance," paper SPE 3484, Journal of Canadian Petroleum Technology (1972) 24, No. 7, 874-882.*

8.3.200 PLYADS - DEFINE POLYMER ROCK ADSORPTION TABLES

Description

The PLYADS keyword defines the rock polymer adsorption tables for when the polymer option has been activated by the POLYMER keyword in the RUNSPEC section. Alternatively, the functions can be entered via the PLYADSS keyword in the PROPS section for when salt sensitivity is to be considered.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	POLCON	A columnar vector of real monotonically increasing down the column values that defines the polymer concentration in the solution surrounding the rock. The first entry should be zero to define a no polymer concentration.			
		lb/stb	kg/sm^3	gm/scc	None
2	POLRATIO	A columnar vector of real increasing down the column values that defines the mass of adsorbed polymer per unit mass of rock. The first entry should be zero to define a zero ratio of polymer concentration.			
		lb/lb	kg/kg	gm/gm	None

Notes:

1) The keyword is followed by NTSFUN tables as declared on the TABDIMS keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NSSFUN rows as declared on the TABDIMS keyword in the RUNSPEC section.
3) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.99: PLYADS Keyword Description

See also the PLYADSS keyword in the PROPS section to also define rock polymer adsorption tables when the polymer concentration is a function of salinity.

Example

```
--
--       POLYMER ROCK ADSORPTION TABLE
--
PLYADS
--       POLYMER    POLYMER
--       POLCON     POLRATIO
--       -------    --------
             0.0     0.00000
             2.0     0.00003
             4.0     0.00005
             6.0     0.00007
             8.0     0.00009
            10.0     0.00011
            12.0     0.00012
            14.0     0.00015                              / TABLE NO. 01
--       POLYMER    POLYMER
--       POLCON     POLRATIO
--       -------    --------
             0.0     0.00000
             3.0     0.00004
             5.0     0.00006
             7.0     0.00008
             8.0     0.00009
            10.0     0.00011                              / TABLE NO. 02
```

The above example defines two polymer rock adsorption tables assuming NTSFUN equals two and NSSFUN is greater than or equal to eight on the TABDIMS keyword in the RUNSPEC section.

There is no terminating "/" for this keyword.

8.3.201 PLYADSS - DEFINE POLYMER ROCK ADSORPTION WITH SALT DEPENDENCE TABLES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The PLYADSS keyword defines the rock polymer adsorption tables for when the polymer and the salt options has been activated by the POLYMER and BRINE keywords in the RUNSPEC section.

Note that the BRINE option is not currently supported by OPM Flow; however, the polymer rock adsorption functions without salt dependence may be entered via the PLYADS keyword in the PROPS section, for when salt sensitivity is not to be considered.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	POLCON	A columnar vector of real monotonically increasing down the column values that defines the polymer concentration in the solution surrounding the rock. The first entry should be zero to define a no polymer and no salt concentration data set. POLCON should only be given for the first entry of the POLCON/POLRATIO set and skipped until another POLCON/POLRATIO table is entered.			
		lb/stb	kg/sm^3	gm/scc	None
2	POLRATIO	A columnar vector of real increasing down the column values that defines the mass of adsorbed polymer per unit mass of rock of the saturated concentration of polymer adsorbed by the rock for a given POLCON and the salt concentration given by SALTCON on the ADSALNOD keyword in the PROPS section. The first table data set entry should be zero to define a no polymer and no salt concentration data set. Subsequent POLRATIO values define the POLCON/POLRATIO combinations for a given salt concentration as listed (and in the same order) by the SALTCON variable on the ADSALNOD keyword in the PROPS section. Each POLCON/POLRATIO/SALT data sets should be terminated by a "/"			
		lb/lb	kg/kg	gm/gm	None

Notes:

1) The keyword is followed by NTSFUN tables as declared on the TABDIMS keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NSSFUN rows as declared on the TABDIMS keyword in the RUNSPEC section.
3) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.100: PLYADSS Keyword Description

See also the PLYADS keyword in the PROPS section to also define rock polymer adsorption tables when the polymer concentration is not a function of salinity.

Example

```
--
--       SETS SALT CONCENTRATION FOR POLYMER SOLUTION ADSORPTION
--       VIA SATNUM ARRAY ALLOCATION
--
--       SALT
--
ADSALNOD
         1.0
         5.0
         10.5
         25.0      / SATNUM TABLE NO. 01
--
--       POLYMER ROCK ADSORPTION WITH SALT DEPENDENCY TABLE
--
PLYADSS
--       POLYMER    POLYMER
--       POLCON     POLRATIO
--       -------    --------
             0.0     0.00000
                     0.00000
                     0.00000
                     0.00000                              / TABLE NO. 01
--       POLYMER    POLYMER
--       POLCON     POLRATIO
--       -------    --------
             1.0     0.00002
                     0.00003
                     0.00004
                     0.00005                              / TABLE NO. 02

--       POLYMER    POLYMER
--       POLCON     POLRATIO
--       -------    --------
             2.0     0.00003
                     0.00004
                     0.00005
                     0.00006                              / TABLE NO. 03
--       POLYMER    POLYMER
--       POLCON     POLRATIO
--       -------    --------
             3.0     0.00004
                     0.00005
                     0.00006
                     0.00007                              / TABLE NO. 04
```

The above example defines four polymer rock adsorption tables for four salt concentration on the ADSALNOD keyword, assuming NTSFUN equals one and NSSFUN is greater than or equal to four on the TABDIMS keyword in the RUNSPEC section.

There is no terminating "/" for this keyword.

8.3.202 PLYATEMP – DEFINE POLYMER ADSORPTION TABLE TEMPERATURE

Description

This keyword defines the polymer adsorption temperature for subsequent polymer adsorption tables entered via the PLYADS and PLYADSS keywords in the PROPS section. The Polymer option must have been activated by the POLYMER keyword in the RUNSPEC section and the Thermal option invoked by the THERMAL keyword, also in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	PLYATEMP	Single real positive value that defines polymer adsorption temperature for subsequent polymer adsorption tables.			
		°F	°C	°C	None
Notes: 1) The keyword is terminated by a "/".					

Table 8.101: PLYATEMP Keyword Description

Example

The example shows how to enter the polymer adsorption data using the PLYADS keyword for two different temperatures.

```
--
--        RESERVOIR
--        TEMPERATURE
--        -----------
PLYATEMP
          60.0                                          / TEMPERATURE
--
--        POLYMER ROCK ADSORPTION TABLE
--
PLYADS
--        POLYMER    POLYMER
--        POLCON     POLRATIO
--        -------    --------
              0.0     0.00000
              2.0     0.00003
              4.0     0.00005
              6.0     0.00007
              8.0     0.00009
             10.0     0.00011
             12.0     0.00012
             14.0     0.00015                          / TABLE NO. 01
```

```
--
--        POLYMER    POLYMER
--        POLCON     POLRATIO
--        -------    --------
              0.0     0.00000
              3.0     0.00004
              5.0     0.00006
              7.0     0.00008
              8.0     0.00009
             10.0     0.00011                                  / TABLE NO. 02
--
--        RESERVOIR
--        TEMPERATURE
--        -----------
PLYATEMP
          120.0                                               / TEMPERATURE
--
--        POLYMER ROCK ADSORPTION TABLE
--
PLYADS
--        POLYMER    POLYMER
--        POLCON     POLRATIO
--        -------    --------
              0.0     0.00000
              2.0     0.00003
              4.0     0.00005
              6.0     0.00007
              8.0     0.00009
             10.0     0.00011
             12.0     0.00012
             14.0     0.00015                                  / TABLE NO. 01
--
--        POLYMER    POLYMER
--        POLCON     POLRATIO
--        -------    --------
              0.0     0.00000
              3.0     0.00004
              5.0     0.00006
              7.0     0.00008
              8.0     0.00009
             10.0     0.00011                                  / TABLE NO. 02
```

Here the first PLYATEMP keyword defines the temperature to be 60 oF for the subsequent two polymer rock adsorption tables, assuming NTSFUN equals four and NSSFUN is greater than or equal to eight on the TABDIMS keyword in the RUNSPEC section. The next PLYATEMP keyword defines the temperature to be 120 oF for the subsequent two polymer rock adsorption tables.

8.3.203 PLYCAMAX - DEFINE POLYMER-ROCK MAXIMUM ADSORPTION BY CELL

Description

The PLYCAMAX keyword defines the maximum polymer-rock adsorption value used in the calculation of the resistance factor for the water phase by individual grid block, for when the Polymer option has been activated by the POLYMER keyword in the RUNSPEC section. See the POLMAX parameter on the PLYROCK keyword in the PROPS section for setting the property for the whole grid.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.204 PLYDHFLF - DEFINE POLYMER THERMAL DEGRADATION HALF-LIFE TABLES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The PLYDHFLF keyword defines the polymer thermal degradation half-life with respect to temperature functions for when the polymer option has been activated by the POLYMER keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	TEMP	A columnar vector of real monotonically increasing down the column values that defines the polymer thermal degradation temperature.			
		°F	°C	°C	None
2	POLHFLF	A columnar vector of real values that defines the corresponding polymer half-life.			
		days	days	hours	None

Notes:

1) The keyword is followed by NTPVT tables as declared on the TABDIMS keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NPPVT rows as declared on the TABDIMS keyword in the RUNSPEC section.
3) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.102: PLYDHFLF Keyword Description

Example

```
--
--       POLYMER THERMAL DEGRADATION HALF-LIFE TABLE
--
PLYDHFLF
--       POLYMER    POLYMER
--       TEMP       HALF-LIFE
--       -------    ---------
             0.0     365.000
            40.0     200.000
            80.0     150.000
           120.0     100.000                              / TABLE NO. 01
--       POLYMER    POLYMER
--       TEMP       HALF-LIFE
--       -------    --------
             0.0     365.000
            50.0     175.000
            75.0     140.000
           100.0     120.000
           125.0      90.000
           150.0      85.000                              / TABLE NO. 02
```

The example defines two polymer thermal degradation half-life tables, based on the NTPVT variable on the TABDIMS keyword in the RUNSPEC section being equal to two and NPPVT variable on the same keyword being greater than or equal to six.

8.3.205 PLYESAL – DEFINE POLYMER EFFECTIVE SALINITY COEFFICIENT

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, PLYESAL, defines the polymer effective salinity coefficient as well as enabling the effective salinity calculation for polymer adsorption. The keyword should only be used if the BRINE keyword has been declared to activate the brine phase, the ECLMC keyword to enable the Multi-Component Brine model, and the POLYMER keyword has been used to activate the polymer phase. All three keywords are in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.206 PLYKRRF – DEFINE POLYMER ROCK PERMEABILITY REDUCTION BY CELL

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The PLYKRRF keyword defines the polymer rock permeability reduction factor to the water phase by individual cell, for when the Polymer option has been activated by the POLYMER keyword in the RUNSPEC section. PLYKRRF should consist of an array of real positive values that are greater than or equal to one. See the PERMFAC parameter on the PLYROCK keyword in the PROPS section for setting the property for the whole grid.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.207 PLYMAX - DEFINE POLYMER-SALT VISCOSITY MIXING CONCENTRATIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The PLYMAX keyword defines maximum polymer and salt concentrations that are to be used in the mixing parameter calculation of the fluid component viscosities, for when the Polymer option has been activated by the POLYMER keyword in the RUNSPEC section.

Note that If the BRINE option has not be activated by the BRINE keyword in the RUNSPEC section, then the salt concentrations in the second column are ignored.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	POLCON	A real value that defines the polymer concentration in the solution which is used to calculate maximum polymer fluid component viscosity.			
		lb/stb	kg/sm^3	gm/scc	None
2	SALTCON	A real value that defines the salt concentration in the solution which is used to calculate maximum polymer fluid component viscosity. Note that If the BRINE option has not been activated by the BRINE keyword in the RUNSPEC section, then this variable is ignored; however, there should still be dummy entries in this case. This variable is ignored as the BRINE and POLYMER combination is not implemented in OPM Flow.			
		lb/stb	kg/sm^3	gm/scc	None

Notes:

1) The keyword is followed by NPLMIX tables as declared on the REGDIMS keyword in the RUNSPEC section, for each rock region. There should be only one row per table.
2) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.103: PLYMAX Keyword Description

Note

Currently, combining the BRINE and POLYMER models is not implemented in OPM Flow, and therefore SALTCON parameter on the PLYMAX keyword is ignored.

Example

```
--
--       POLYMER-SALT VISCOSITY MIXING CONCENTRATIONS
--
PLYMAX
--       POLYMER    SALT
--       POLCON     SALTCON
--       -------    --------
         0.0100      0.0500                              / TABLE NO. 01
         0.0075      0.0400                              / TABLE NO. 02
         0.0050      0.0300                              / TABLE NO. 03
```

The above example defines three polymer-salt viscosity mixing concentrations, based on the NPLMIX variable on the REGDIMS keyword in the RUNSPEC section being equal to three.

8.3.208 PLYMWINJ – POLYMER MOLECULAR WEIGHT MODEL THROUGHPUT AND VELOCITY TABLE

Description

This keyword, PLYMWINJ, describes the relationship of the injected polymer molecular weight as a function of polymer throughput and polymer velocity, for the simulator's Polymer Molecular Weight Transport option, that uses the polymer molecular weight in calculating the polymer viscosity. The table is a two dimensional table that relates the polymer throughput values and velocity values to derive the resulting molecular weight of the injected polymer, which is then used via the PLYVHM keyword in the PROPS section, to derive the polymer molecular weight scaled viscosity.

This keyword should only be used if the POLYMER and POLYMW keywords in the RUNSPEC section are also activated.

Note

This is an OPM Flow specific keyword that employs an alternative polymer flood model based on a Polymer Molecular Weight Transport equation, that is not available in the commercial simulator.

The model has been tested using metric units; however, using either field or laboratory units with the option should be considered experimental.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1-1	PLYMWNUM	A positive integer value greater than zero and less than or equal to the NTPMWINJ variable, as defined on the PINTDIMS keyword in the RUNSPEC section, that defines the PLYMWINJ Polymer Molecular Weight Model throughput and velocity table number.			None
2-1	THRUPUT	A real positive monotonically increasing vector, that defines the polymer throughput values. The first entry should be zero to define a no throughput data set, and each vector record should be on a separate line terminated by a "/".			None
		feet^3/feet^2	m^3/m^2	cm^3/cm^2	
3-1	VELOCITY	A real positive monotonically increasing vector, that defines the polymer velocity values. The first entry should be zero to define a no velocity data set, and each vector record should be on a separate line terminated by a "/".			None
		feet/day	m/day	cm/hour	

No.	Name	Description			Default
		Field	Metric	Laboratory	
4-1	POLYMW	A series of real positive vectors representing the polymer molecular weights for all combinations of the polymer throughput values (THRUPUT) and velocity values (VELOCITY), organized as a series of vectors POLYMW(THRUPUT,VELOCITY). Thus, the first vector represents the molecular weights of the first THRUPUT value and each entry in the vector is the corresponding molecular weight of the associated VELOCITY vector. Each vector should be on a separate line and should be terminated by a "/". Thus, if THRUPUT has three entries and VELOCITY has four, then there should be three vectors, with each vector containing four elements representing molecular weight values, as a function of THRUPUT and VELOCITY.			None
		lb/lb-M	kg/kg-M	gm/gm-M	

Notes:

1) The keyword should be terminated by a "/", but may be repeated up to NTPMWINJ times, as per the PINTDIMS keyword in the RUNSPEC section, to allow for the input of multiple tables.

Table 8.104: PLYMWINJ Keyword Description

Unlike other PROPS section table keywords, that enable multiple tables following the keyword to be entered, the PLYMWINJ keyword requires that the keyword itself must be repeated for each table, as is shown in the example on the following page.

See also the WPMITAB keyword in the SCHEDULE section, that assigns the PLYMWINJ tables to the water injection wells, as well as the SKPRWAT, SKPRPOLY, and PLYVMH keywords, in the PROPS section, that are the additional property keywords required for the Polymer Molecular Weight Transport option.

The WSKPTAB keyword in the SCHEDULE section may be used to assign the SKPRPOLY and SKPRWAT tables to water injections wells, that enable the calculation of the wellbore skin pressure based on the fluids being injected.

Note that the standard polymer property data keywords: PLYROCK, PLYADS, PLYMAX, etc., are still required to fully describe the polymer fluid.

Example

Given NTPMWINJ equals two on the PINTDIMS keyword in the RUNSPEC section, then two PLYWINJ tables are required to be entered:

```
--
--        POLYMER MOLECULAR WEIGHT MODEL THROUGHPUT AND VELOCITY TABLE
--        (OPM FLOW PROPS KEYWORD)
--
PLYMWINJ
          1                              / TABLE NUMBER
--
--        THROUGHPUT VALUES
--
          0.0     200.0   400.0          /
--
--        VELOCITY VALUES
--
          0.0     50.0    80.0    100.0 /
--
--        POLYMW VALUES
--
          20.0    19.0    18.0    16.0  / POLYMW(OUTPUT=1, VELOCITY=1 TO N)
          20.0    16.0    14.0    12.0  / POLYMW(OUTPUT=2, VELOCITY=1 TO N)
          20.0    12.0     8.0     4.0  / POLYMW(OUTPUT=3, VELOCITY=1 TO N)
/
PLYMWINJ
          2                              / TABLE NUMBER
--
--        THROUGHPUT VALUES
--
          0.0     200.0   400.0          /
--
--        VELOCITY VALUES
--
          0.0     50.0    70.0    100.0 /
--
--        POLYMW VALUES
--
          20.0    19.0    18.0    16.0  / POLYMW(OUTPUT=1, VELOCITY=1 TO N)
          20.0    16.0    14.0    12.0  / POLYMW(OUTPUT=2, VELOCITY=1 TO N)
          20.0    12.0     8.0     4.0  / POLYMW(OUTPUT=3, VELOCITY=1 TO N)
/
```

As mentioned previously, the PLYMWINJ keyword requires that the keyword itself must be repeated for each table, as is shown in the above example.

8.3.209 PLYRMDEN - DEFINE POLYMER MODEL IN SITU ROCK DENSITY

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The PLYRMDEN keyword defines the in situ rock density at reservoir conditions by individual cell, for when the Polymer option has been activated by the POLYMER keyword in the RUNSPEC section. PLYRMDEN should consist of an array of real positive values. See the DENSITY parameter on the PLYROCK keyword in the PROPS section for setting the property for the whole grid.

8.3.210 PLYROCK - DEFINE POLYMER-ROCK PROPERTIES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The PLYROCK keyword defines rock properties for when the Polymer option has been activated by the POLYMER keyword in the RUNSPEC section.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	PSPACE	A real positive value that is greater than or equal to zero and less the maximum water saturation and less than one, that defines available pore space for this rock type.			
		dimensionless	dimensionless	dimensionless	None
2	PERMFAC	A real positive value that is greater than or equal to one that defines decrease in the rock permeability to the water phase when the maximum amount of polymer has been adsorbed.			
		dimensionless	dimensionless	dimensionless	None
3	DENSITY	A real value that defines the rock in-situ density, that is at reservoir conditions.			
		lb/rb	kg/rm^3	gm/rcc	None
4	ADINDX	A positive integer of 1 or 2 that defines the polymer desorption option. 1) then polymer desorption may occur by retracing the polymer adsorption isotherm when the local polymer concentration in the solution decreases. 2) then no polymer desorption may occur.			
		dimensionless 1	dimensionless 1	dimensionless 1	Defined
5	POLMAX	A real positive non-zero value that defines the maximum polymer adsorption to be used in the calculation of the resistance factor for the water phase.			
		lb/lb	kg/kg	gm/gm	None

Notes:

1) The keyword is followed by NTSFUN tables as declared on the TABDIMS keyword in the RUNSPEC section, for each polymer flooding region. There should be only one row per table.
2) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.105: PLYROCK Keyword Description

Example

```
--
--       POLYMER-ROCK PROPERTIES
--
PLYROCK
--       PORE      PERM      INSITU    DESORP  MAX
--       SPACE     FACTOR    DENSITY   OPTN    POLY
--       ------   --------   -------   ------  -------
         0.1200    1.7500    1800.0      1     0.00012     / TABLE NO. 01
         0.1300    1.8500    1980.0      2     0.00015     / TABLE NO. 02
         0.1500    1.9500    2005.0      1     0.00014     / TABLE NO. 03
```

The above example defines three polymer-rock tables, based on the NTSFUN variable on the TABDIMS keyword in the RUNSPEC section being equal to three.

There is no terminating "/" for this keyword.

8.3.211 PLYSHEAR – ACTIVATE AND DEFINE POLYMER SHEARING PARAMETERS

Description

The PLYSHEAR keyword activates and defines the polymer shear thinning-thickening option for when the polymer option has been activated by the POLYMER keyword in the RUNSPEC section.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	VELOCITY	A columnar vector of real monotonically increasing down the column values that defines the water-polymer flow velocity. The VELOCITY value for the first row in the table should be zero.			
		feet/day	m/day	cm/hour	None
2	VISFAC	A columnar vector of real values that defines a factor that scales the effective water and polymer viscosities for when shear thinning-thickening of the polymer occurs. Normally VISFAC value for the first row in the table should be one.			
		dimensionless	dimensionless	dimensionless	None

Notes:

1) The keyword is followed by NTPVT tables as declared on the TABDIMS keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NPPVT rows as declared on the TABDIMS keyword in the RUNSPEC section.
3) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.106: PLYSHEAR Keyword Description

Example

```
--
--       ACTIVATE AND DEFINE POLYMER SHEARING PARAMETERS
--
PLYSHEAR
--       WAT-POLY    VISCOSITY
--       VELOCITY    FACTOR
--       --------    ---------
             0.0       1.000
             1.0       0.900
             3.0       0.800
             6.0       0.700                          / TABLE NO. 01
--       WAT-POLY    VISCOSITY
--       VELOCITY    FACTOR
--       --------    ---------
             0.0       1.000
             1.0       0.900
             2.0       0.800
             4.0       0.750
             6.0       0.700
             8.0       0.650                          / TABLE NO. 02
```

The above example activates the polymer shear thinning-thickening option and defines two polymer shear thinning-thickening tables, based on the NTPVT variable on the TABDIMS keyword in the RUNSPEC section being equal to two and NPPVT variable on the same keyword being greater than or equal to six.

8.3.212 PLYSHLOG - ACTIVATE AND DEFINE THE POLYMER SHEARING LOGARITHMIC PARAMETERS

Description

This keyword activates and defines the parameters for the logarithm-based polymer shear thinning/thickening option.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1-1	POLCON	A real positive value that defines the reference polymer concentration for the VELOCITY and VISFAC data for this keyword.			
		lb/stb	kg/sm^3	gm/scc	None
1-2	SALTCON	A real positive value that defines the reference salt concentration for the VELOCITY and VISFAC data for this keyword. Note that If the BRINE option has not been activated by the BRINE keyword in the RUNSPEC section, then this variable is ignored. This variable is ignored as the BRINE option is not implemented in OPM Flow.			
		lb/stb	kg/sm^3	gm/scc	None
1-3	TEMP	A real positive value defines the reference polymer temperature for the VELOCITY and VISFAC data for this keyword. Note that If the TEMP option has not been activated by the TEMP keyword in the RUNSPEC section, then this variable is ignored. This variable is ignored as the TEMP and POLYMER options combination is not implemented in OPM Flow.			
		°F	°C	°C	None
1-4	/	Record terminated by a "/"			Not Applicable
2-1	VELOCITY	A columnar vector of real monotonically increasing down the column values that defines the water-polymer flow velocity for the reference conditions of POLCON, SALTCON and TEMP. The VELOCITY value for the first row in the table should be a very small value that is greater than zero and less than 1 x 10-6.			
		feet/day	m/day	cm/hour	None
2-2	VISFAC	A columnar vector of real positive values that define the dimensionless shear effect multiplier for the given VELOCITY entry for the reference conditions of POLCON, SALTCON and TEMP. Normally VISFAC value for the first row in the table should be one.			
		dimensionless	dimensionless	dimensionless	None
1-4	/	Record terminated by a "/"			Not Applicable

No.	Name	Description			Default
		Field	Metric	Laboratory	

Notes:

1) The keyword is followed by NTPVT tables as declared on the TABDIMS keyword in the RUNSPEC section.
2) Each table must be entered with two records, with entries 1-1, 1-2, 1-3 and 1-4 representing record number one and 2-1, 2-2 and 2-3 representing record number two in the "No." column in this table.
3) Each of the records are terminated by a "/" and is explicitly shown in the above rows.
4) For record number two a minimum of two rows and a maximum of NPPVT rows, as declared on the TABDIMS keyword in the RUNSPEC section, are required.
5) There is no "/" terminator for the keyword.

Table 8.107: PLYSHLOG Keyword Description

See the PLYSHEAR keyword for the alternative polymer shear thinning/thickening option that is also implemented in OPM Flow.

Example

The following example show how to enter two PLYSHLOG tables given that the NTPVT variable on the TABDIMS keyword in the RUNSPEC section is set equal to two.

```
--
--        POLYMER SHEARING LOGARITHMIC PARAMETERS
--
PLYSHLOG
--        REF          REF         REF
--        POLCON       SALTCON     TEMP
--        --------     -------     ----
          0.5
/
--
--        VELOCITY     VISFAC
--        --------     -------
          0.0000001     1.00
          0.000001      1.10
          0.0001        1.30
          0.001         1.47
          0.01          1.67
          0.1           2.00
          1.0           2.20
          10.0          2.30
          100.0         2.40
          1000.0        2.40
                                                    / TABLE NO. 01
--        REF          REF         REF
--        POLCON       SALTCON     TEMP
--        --------     -------     ----
          0.5
/
```

```
--
--        VELOCITY    VISFAC
--        --------    -------
          0.0000001    1.00
          0.000001     1.10
          0.0001       1.35
          0.001        1.57
          0.01         1.87
          0.1          2.20
          1.0          2.40
          10.0         2.60
          100.0        2.65
          1000.0       2.65
                                    / TABLE NO. 02
```

The example activates the polymer logarithmic shear thinning-thickening option and defines two polymer shear thinning-thickening tables, based on the NTPVT variable on the TABDIMS keyword in the RUNSPEC section being equal to two and NPPVT variable on the same keyword being greater than or equal to ten.

8.3.213 PLYTRRF – DEFINE POLYMER ROCK PERMEABILITY REDUCTION VERSUS TEMPERATURE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The PLYTRRF keyword defines the polymer rock permeability reduction factor to the water phase as a function of temperature, for when the Polymer option has been activated by the POLYMER keyword in the RUNSPEC section. See the PLYTRRF keyword for the options on how this data is used in the polymer model and the PERMFAC parameter on the PLYROCK keyword for setting the property for the whole grid for a constant temperature. Both keywords are in the PROPS section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.214 PLYTRRFA – DEFINE POLYMER ROCK PERMEABILITY REDUCTION VERSUS TEMPERATURE OPTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The PLYTRRFA keyword defines the how the polymer rock permeability reduction factor to the water phase as a function of temperature data, entered via the PLYTRRA keyword in the PROPS section, should be used. This keyword should only be used if the Polymer option has been activated by the POLYMER keyword in the RUNSPEC section. See the PERMFAC parameter on the PLYROCK keyword in the PROPS section for setting the property for the whole grid for a constant temperature.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.215 PLYVISC – DEFINE POLYMER VISCOSITY SCALING FACTORS

Description

PLYSVISC defines the polymer viscosity scaling factors used to determine the relationship of pure water viscosity with respect to increasing polymer concentration within a grid block. The polymer option must be activated by the POLYMER keyword in the RUNSPEC section in order to use this keyword.

The BRINE option in the RUNSPEC should be deactivated if this keyword is to be used.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	POLCON	A columnar vector of real monotonically increasing down the column values that defines the polymer concentration in the solution surrounding the rock. The first entry should be zero to define a no polymer concentration.			
		lb/stb	kg/sm^3	gm/scc	None
2	VISFAC	A columnar vector of real increasing or equal values that defines a factor that scales the effective viscosity of the solution for the given POLCON entry. Normally VISFAC value for the first row in the table should be one.			
		dimensionless	dimensionless	dimensionless	None

Notes:

1) The keyword is followed by NTPVT tables as declared on the TABDIMS keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NPPVT rows as declared on the TABDIMS keyword in the RUNSPEC section.
3) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.108: PLYVISC Keyword Description

Example

```
--
--       POLYMER VISCOSITY SCALING FACTOR TABLES
--
PLYVISC
--       POLYMER     VISCOSITY
--       POLCON      VISFAC
--       --------    ---------
          0.0000       1.000
          0.0002      10.000
          0.0004      20.000
          0.0008      40.000                          / TABLE NO. 01
```

```
--       POLYMER     VISCOSITY
--       POLCON      VISFAC
--       --------    ---------
          0.0000       1.000
          0.0003      10.000
          0.0005      20.000
          0.0007      40.000
          0.0009      45.000
          0.0011      55.000                          / TABLE NO. 02
```

The example defines two polymer viscosity scaling factor tables, based on the NTPVT variable on the TABDIMS keyword in the RUNSPEC section being equal to two and NPPVT variable on the same keyword being greater than or equal to six.

8.3.216 PLYVISCS – DEFINE POLYMER-SALT VISCOSITY SCALING FACTORS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

PLYSVISCS defines the polymer-salt viscosity scaling factor tables applied to pure water that are used to determine the viscosity of a polymer-salt mixture with respect to increasing polymer concentration within a grid block. The polymer option must be activated by the POLYMER keyword, as well as the brine phase declared by the BRINE keyword in the RUNSPEC section in order to use this keyword. However the ECLM keyword in the RUNSPEC must not be used with this keyword.

See also the PLYVSCST keyword in the PROPS section to enter polymer viscosity scaling factor data that is dependent on both salt and reservoir temperature.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.217 PLYVISCT – DEFINE POLYMER-TEMPERATURE VISCOSITY SCALING FACTORS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

PLYSVISCT defines the polymer-temperature viscosity scaling factor tables applied to pure water that are used to determine the viscosity of the polymer at a given temperature with respect to increasing polymer concentration within a grid block. Both the polymer option must be activated by the POLYMER keyword and the temperature option invoked by the TEMP keyword in the RUNSPEC section in order to use this keyword. However the BRINE keyword in the RUNSPEC must not be used with this keyword, that is the salt sensitivity options should be deactivated.

See also the PLYVSCST keyword in the PROPS section to enter polymer viscosity scaling factor data that is dependent on both salt and reservoir temperature.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.218 PLYVMH - POLYMER MOLECULAR WEIGHT MODEL POLYMER VISCOSITY CONSTANTS

Description

This keyword, PLYVMH, defines the constants used to calculate viscosity of the polymer solution as a function of the polymer molecular weight and the polymer concentration, for the simulator's Polymer Molecular Weight Transport option, that uses the polymer molecular weight in calculating the polymer viscosity. The keyword consists of a series of row vectors, which each vector having four elements, that define the constants used in calculating the polymer viscosity

This keyword should only be used if the POLYMER and POLYMW keywords in the RUNSPEC section are also activated.

Note

This is an OPM Flow specific keyword that employs an alternative polymer flood model based on a Polymer Molecular Weight Transport equation, that is not available in the commercial simulator.

The model has been tested using metric units; however, using either field or laboratory units with the option should be considered experimental.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	MHK	The Mark-Houwink K polymer specific constant in the Mark-Houwink equation, see equation (8.81).			None
			ml/g		
2	MHA	The Mark-Houwink exponent parameter a, in the Mark-Houwink equation, see equation (8.81).			None
		dimensionless	dimensionless	dimensionless	
3	GAMMA	The γ intrinsic water-polymer viscosity constant in the Huggins equation, see equation (8.83).			None
4	KAPPA	The κ intrinsic water-polymer viscosity constant in the Huggins equation, see equation (8.83).			None

Notes:

1) The keyword is followed by NPLYVHM records as declared on the PINTDIMS keyword in the RUNSPEC section, with each record terminated by a "/" and the keyword itself should be terminated by a "/".

Table 8.109: PLYVMH Keyword Description

The high molecular weight of polymers greatly increase the viscosity of the injected water in which they are dissolved, thus adjusting the mobility ration of the displacing phase. The increase in viscosity is caused by strong internal friction between the randomly coiled and swollen macro molecules and the surrounding

water molecules. And is dependent on both the nature of the polymer and the injected water acting as the solvent. There are several formulations of viscosity associated with polymer rheology, namely:

Relative viscosity is defined as:

$$\eta_{rel} = \frac{\eta}{\eta_s} \tag{8.77}$$

where:

η_{rel} = is the relative viscosity,
η = is the viscosity of the solution, and
η_s = is the viscosity of the solvent (injected water).

Specific viscosity is defined as:

$$\begin{aligned} \eta_{\text{sp}} &= \frac{(\eta - \eta_s)}{\eta_s} \\ &= \frac{\eta}{\eta_s} - 1 \\ &= \eta_{rel} - 1 \end{aligned} \tag{8.78}$$

where:

η_{sp} = is the specific water-polymer viscosity, and
η_s = is the viscosity of the solvent (injected water).

Reduced Specific Viscosity is given by:

$$\begin{aligned} \eta_{\text{red}} &= \frac{\eta_{\text{sp}}}{C_p} \\ &= \frac{(\eta_{rel} - 1)}{C_p} \end{aligned} \tag{8.79}$$

where:

η_{red} = is the reduced specific water-polymer viscosity, and
C_p = is the polymer concentration.

Finally, the Intrinsic Velocity, which is defined as a measure for the internal friction in polymer solutions at the limit of zero polymer concentration. Thus, this quantity describes the effect of completely separated polymer chains on the solution viscosity, and is defined as:

$$\begin{aligned} [\eta] &= \lim_{C_p \to 0} \frac{\eta_{\text{sp}}}{C_p} \\ &= \lim_{C_p \to 0} \frac{\eta - \eta_0}{\eta_0 C_p} \end{aligned} \tag{8.80}$$

where:

$[\eta]$ = is the intrinsic water-polymer viscosity and describes the increase in viscosity arising from an individual polymer chain and is a measure of the polymers' thickening power, and
η_o = zero-shear viscosity.

For any given solvent pair, the intrinsic viscosity increases as the molecular weight of the polymer increases; here, the *Mark-Houwink equation*, also known as the Mark-Houwink-Staudinger equation[215], [216], and [217] is used to calculate [η], that is:

$$[\eta] = K \cdot M_w^a \tag{8.81}$$

where:

K = is the polymer specific constant in the Mark-Houwink equation, the MHK parameter in Table 8.109,
M_w = is the polymer molecular weight, and
a = the exponent constant in the Mark-Houwink equation, the MHA parameter in Table 8.109.

The Mark-Houwink parameters can be determined from a double logarithmic plot of intrinsic viscosity versus molecular weight which yields straight lines, that is:

$$\ln([\eta]) = \ln(K) + a \times \ln(M_w) \tag{8.82}$$

OPM Flow uses a form of the Huggins[218] equation to calculate the polymer apparent viscosity, as shown below:

$$\frac{\eta_0}{\eta_s} = 1 + \gamma\left(X + \kappa X^2\right) \quad \text{where} \quad X = [\eta] C_p \tag{8.83}$$

where:

γ = a user defined constant, GAMMA in Table 8.109, and
κ = a user defined constant, KAPPA in Table 8.109.

Which can be used to calculate the zero-shear viscosity, η_0, based on the quadratic function in equation (8.83) time polymer concentration(C_p) and polymer intrinsic viscosity described in equation (8.81).

In terms of the keyword units, given the intrinsic viscosity in ml/g, polymer concentration in kg/m3, and the molecular weight as kg/kg-M, then equation (8.81) becomes:

$$[\eta] = K\left(M_w \cdot 1.0\times10^{-3}\right)^a \cdot 1.0\times10^{-3} \tag{8.84}$$

and equation (8.83) becomes:

$$\frac{\eta_0}{\eta_s} = 1 + \gamma\left(X + \kappa X^2\right) \quad \text{where} \quad X = 1.0\times10^{-6}[\eta] C_p \tag{8.85}$$

Note that the model does not account for non-Newtonian flow; the apparent viscosity is simply set equal to the zero-shear viscosity, and that the model only considers the full mixing between the polymer and water.

[215] *H. Mark, in R. Saenger, Der feste Koerper, Hirzel, Leipzig, 1938.*

[216] *R. Houwink , J. Prakt. Chem., Vol. 157, Issue 1-3, p. 15 (1940).*

[217] *H. Staudinger, Die Hochmolekulare Organischen Verbindungen, Julius Springer, Berlin 1932.*

[218] *Huggins, M. L. 1942. The viscosity of dilute solutions of long-chain molecules. IV. Dependence on concentration.» Journal of the American Chemical Society 64 (11): 2716-2718.*

See also the PLYMWINJ keyword in the PROPS section, that describes the relationship of the injected polymer molecular weight as a function of polymer throughput and polymer velocity. Note that the standard polymer property data keywords: PLYROCK, PLYADS, PLYMAX, etc., are still required to fully describe the polymer fluid.

Example

Given NPLYVMH equals two on the PINTDIMS keyword in the RUNSPEC section, then:

```
--
--       POLYMER MOLECULAR WEIGHT MODEL POLYMER VISCOSITY CONSTANTS
--       (OPM FLOW PROPS KEYWORD)
--
--       MHK     MHA     VISC    VISC
--       CONST   EXPON   GAMMA   KAPPA
PLYVMH
         0.02    0.50    0.40    0.60                    /
         0.03    0.51    0.42    0.63                    /
/
```

Two sets of data should be entered with the PLYVMH keyword, as shown above.

8.3.219 PLYVSCST – DEFINE POLYMER-SALT-TEMPERATURE VISCOSITY SCALING FACTORS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

PLYVSCST defines the polymer-salt-temperature viscosity scaling factor tables applied to pure water that are used to determine the viscosity of the polymer at a given salt concentration and for a given temperature, with respect to increasing polymer concentration within a grid block. Both the polymer option must be activated by the POLYMER keyword and the temperature option invoked by the TEMP keyword in the RUNSPEC section in order to use this keyword. In addition, the BRINE keyword in the RUNSPEC must also be invoked. The keyword is used in conjunction with the SALTNODE keyword to define the various salt concentrations and the TEMPNODE keyword to define the various reservoir temperatures. Both keywords are in the PROPS section.

See also the PLYVISCS keyword in the PROPS section to enter polymer viscosity scaling factor data that is dependent just salt concentration and the PLYVISCT keyword in the PROPS section to enter polymer viscosity scaling factor data that is dependent just on reservoir temperature.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.220 PMAX – MAXIMUM AND MINIMUM PRESSURE FOR TOTAL COMPRESSIBILITY CHECK

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The PMAX keyword defines the maximum and minimum pressures expected to be encountered during the run. The data is used to perform the PVT total compressibility check that ensures that the total compressibility of a mixture of oil-gas, for when the gas-oil ratio is increasing for an oil, or the condensate gas ratio is increasing for a gas condensate, is positive respect to pressure. The total compressibility check is used to ensure that the entered oil and gas PVT data is consistent. If the check fails for given oil-gas mixture at a given pressure, resulting in a negative total compressibility, then this will result in numerical instabilities in the run causing this simulator difficulties in converging to a solution.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.221 PMISC – DEFINE MISCIBILITY VERSUS PRESSURE TABLES

Description

PMISC defines the transition between immiscible and miscible displacement as a function of oil pressure tables, for when the MISCIBLE keyword in the RUNSPEC section has be activated. If this keyword is absent from the input deck and MISCIBLE keyword in the RUNSPEC keyword has been activated, then miscibility is independent of the oil phase pressure.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	PRESS	A columnar vector of real monotonically increasing down the column values that defines the oil phase pressure.			
		psia	barsa	atma	None
2	MISC	A columnar vector of real equal or increasing down the column values that defines the corresponding miscibility factor. MISC is a scaling that should lie be zero and one, where zero means no miscibility and one means full miscibility.			
		dimensionless	dimensionless	dimensionless	None

Notes:

1) The keyword is followed by NTMISC tables as declared on the MISCIBLE keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NSMISC rows as declared on the MISCIBLE keyword in the RUNSPEC section.
3) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.110: PMISC Keyword Description

Example

```
--
--       MISCIBILITY VERSUS PRESSURE TABLES
--
PMISC
--       OIL        MISCIBILE
--       PRESS      FACTOR
--       -------    ---------
          1000.0       0.000
          2000.0       0.250
          3000.0       1.000
          4000.0       1.000                          / TABLE NO. 01
--       OIL        MISCIBILE
--       PRESS      FACTOR
--       -------    ---------
          1500.0       0.000
          2000.0       0.000
          2500.0       0.250
          3000.0       0.350
          3500.0       1.000
          4000.0       1.000                          / TABLE NO. 02
```

The above example defines two miscibility versus pressure tables assuming NTMISC equals two and NSMISC is greater than or equal to six on the MISCIBLE keyword in the RUNSPEC section.

8.3.222 PPCWMAX – DEFINE SWATINIT CALCULATED CAPILLARY PRESSURE CONSTRAINTS

Description

The PPCWMAX keyword defines the maximum capillary pressure allowed when scaling the capillary pressure tables to match the inputted SWATINIT array. This is primary used for when the SWATINIT array has values of water saturation above the connate water saturation significantly outside than capillary pressure transition zone, that is high on the structure. In this case OPM Flow may generate large values for the capillary pressure which may result in numerical converge problems. This keyword sets the maximum allowable calculated capillary pressure and how the water saturation should be treated when the limit is exceeded.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	PCWO	A columnar vector of real values that defines the maximum allowable capillary pressure for each SATNUM region. The default value of infinity means there is no limit applied.			
		psia	barsa	atma	Infinity
2	OPTN	A columnar vector of character strings that should be set to: 1) NO: To ignore the SWATINIT value for the offending cell for when PCWO is exceeded. In this cases the capillary pressure for the block is set to the maximum (PCWO) and the water saturation is re-calculated based on PCWO. 2) YES: To set the SWATINIT value to the connate water saturation for the offending cell for when PCWO is exceeded. In this case the capillary pressure is set to the maximum value of the appropriate SATNUM table and the initial water saturation is calculated to be consistent with the tables maximum capillary pressure. This results in the capillary pressures not being re-scale for the offending cell.			No

Notes:

1) The keyword is followed by NTSFUN rows as declared on the TABDIMS keyword in the RUNSPEC section.
2) Each row show contain two values representing PCWO and OPTN values.
3) Each row is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.111: PPCWMAX Keyword Description

Note

Using this keyword to limit the re-scaled grid block capillary pressure values will effect the fluids in-place when the simulator has to re-calculate values due to the capillary pressure limit being exceeded.

In addition, the high grid block capillary pressures may be indicative of an inconsistency between the tabular SATNUM capillary pressure values and the provided SWATINIT array water saturations. This inconsistency may be a result of the SWATINIT array being derived using a saturation height function, as is customary in static modeling software, and the numerical models tabulated capillary pressure. Rather than resetting the maximum calculated capillary pressure using the PPCWMAX keyword, it may be more appropriate to investigate the reason for the high capillary pressures values first, prior to applying the keyword.

Example

```
--
--       SET MAXIMUM PC FOR SWATINIT INITIALIZATION
--       MAX       MATCH
--       PC        SWATINIT
--       --------  --------
PPCWMAX
         100.0     YES                                    / TABLE NO 01
         125.0     YES                                    / TABLE NO 02
         135.0     YES                                    / TABLE NO 03
```

The above example sets the maximum capillary pressure for three saturation regions to 100, 125 and 135 with SWATINIT reset to the connate water saturation for when the capillary pressure limit is exceeded.

8.3.223 PROPS - DEFINE THE START OF THE PROPS SECTION OF KEYWORDS

Description

The PROPS activation keyword marks the end of the EDIT section and the start of the PROPS section that defines the key fluid and rock property data for the simulator

There is no data required for this keyword.

Example

```
-- ==========================================================================
--
-- PROPS SECTION
--
-- ==========================================================================
PROPS
```

The above example marks the end of the EDIT section and the start of the PROPS section in the OPM Flow data input file.

8.3.224 PVCDO - OIL PVT PROPERTIES FOR DEAD OIL (CONSTANT COMPRESSIBILITY)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

PVCDO defines the oil PVT properties for dead oil[219] with constant compressibility. If the oil has a constant and uniform dissolved gas concentration, Gas-Oil Ratio ("GOR"), and if the reservoir pressure never drops below the saturation pressure (bubble point pressure), then the model can be run more efficiently by omitting the GAS and DISGAS keywords from the RUNSPEC section, treating the oil as a dead oil, and defining a constant Rs (GOR) value with keyword RSCONST or RSCONSTT in the PROPS section. This results in the model being run as a dead oil problem with no active gas phase. However, OPM Flow takes into account the constant Rs in the calculations and reporting.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	PRESS	PRESS is a real positive value defining the oil reference pressure for the other parameters for this data set.			
		psia	barsa	atma	None
2	OFVF	OFVF is a real positive value defining the oil formation volume factor (Bo) at the reference pressure.			
		rb/stb	rm^3/sm^3	rcc/scc	None
3	OCOMP	OCOMP is a real positive value defining the oil compressibility (Co) at the oil reference pressure and is defined as: $$C_o = -\frac{1}{B_o}\left(\frac{dB_o}{dP}\right)$$			
		1/psia	1/barsa	1/atma	None
4	OVISC	OVISC is a real positive value defining the oil viscosity (μo) at the oil reference pressure.			
		cP	cP	cP	None
5	OVISCOMP	OVISCOMP is a real positive value defining the oil viscosibility (μoc) at the oil reference pressure and is defined as: $$\mu_{oc} = \frac{1}{\mu_o}\left(\frac{d\mu_o}{dP}\right)$$			
		1/psia	1/barsa	1/atma	None

Notes:

1) The keyword is followed by NTPVT tables as declared on the TABDIMS keyword in the RUNSPEC section.
2) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.112: PVCDO Keyword Description

[219] *"Dead" oil is oil that it contains no dissolved gas or a relatively thick oil or residue that has lost its volatile components.*

See also the RSCONST and RSCONSTT keywords to define the constant Rs for dead oil and PVDO as an alternative keyword to enter the dead oil properties.

Example

```
--
--       OIL PVT TABLE FOR DEAD WITH CONSTANT COMPRESSIBILITY
--
PVCDO
--       REF PRES  BO         CO        VISC     VISC
--       PSIA      RB/STB     1/PSIA    CPOISE   GRAD
--       --------  --------   -------   ------   ------
          3840.0    1.080     1.5E-6     1.750    0.0       / TABLE NO. 01
          3840.0    1.100     1.5E-6     1.050    0.0       / TABLE NO. 02
          3840.0    1.120     1.6E-6     0.950    0.0       / TABLE NO. 03
          3840.0    1.140     1.7E-6     0.850    0.0       / TABLE NO. 04
          3840.0    1.160     1.7E-6     0.800    0.0       / TABLE NO. 05
```

The above example defines five dead oil PVT tables with constant compressibility and viscosity, and assumes that NTPVT equals five on the TABDIMS keyword in the RUNSPEC section.

There is no terminating "/" for this keyword.

8.3.225 PVCO - OIL PVT PROPERTIES FOR LIVE OIL

Description

PVCO defines the oil PVT properties for live[220] and the keyword should only be used if the there is both oil and gas phases in the model. This keyword should be used when the DISGAS keyword has be declared in the RUNSPEC section indicating that dissolved gas (more commonly referred to as solution gas) is present in the oil. The keyword may be used for oil-water and oil-water-gas input decks. This is an alternative keyword to the PVTO keyword in the PROPS section that also enables entering live oil PVT data. Here, the PVCO keyword assumes that for the undersaturated oil with a given Gas-Oil Ratio ("GOR" or "Rs"), the oil compressibility is independent of the pressure. Hence, is not necessary to enter the undersaturated oil formation volume factor versus pressure data. Similarly, the viscosity of the same type of oil is assumed to have a pressure independent "viscosibility" derivative, and therefore it is not necessary to enter undersaturated viscosity versus pressure data.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped; however, the PVTO keyword in the PROPS section may be used to enter live oil PVT data instead.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	PRESS	PRESS is a real columnar vector of real monotonically increasing down the column values that defines the oil phase saturation pressure (bubble-point pressure), that defines the RS, oil formation volume factor and the oil viscosity at PRESS.			
		psia	barsa	atma	None
2	RS	RS is a real monotonically increasing down the column values that defines the saturated gas-oil ratio ("GOR") or Rs, for the given value of PRESS.			
		Mscf/stb	sm^3/sm^3	scc/scc	1*
3	OFVF	OFVF is a real positive value defining the oil saturated formation volume factor (Bo) at the saturation pressure PRESS.			
		rb/stb	rm^3/sm^3	rcc/scc	None
4	OVISC	OVISC is a real positive value defining the oil viscosity (μo) at the oil saturated reference pressure, PRESS.			
		cP	cP	cP	1*
5	OCOMP	OCOMP is a real positive value defining the oil compressibility (Co) at the saturated oil reference pressure and is defined as: $$C_o = -\frac{1}{B_o}\left(\frac{dB_o}{dP}\right)$$			
		1/psia	1/barsa	1/atma	1*

[220] *"Live" oil is oil that contains gas in solution, which is normally the case for most conventional oil reservoirs. However, for oil reservoirs classified as heavy oil reservoirs, the in situ dissolved gas may be negligible and oil would then be classified as gas-free oil which is commonly referred to as "dead" oil.*

No.	Name	Description			Default
		Field	Metric	Laboratory	
6	OVISCOMP	OVISCOMP is a real positive value defining the oil viiscosibility (μoc) at the saturated oil reference pressure with the given RS, where (μoc) is defined as: $$\mu_{oc} = -\frac{1}{\mu_o}\left(\frac{d\mu_o}{dP}\right)$$			
		1/psia	1/barsa	1/atma	1*

Notes:

1) The keyword is followed by NTPVT tables as declared on the TABDIMS keyword in the RUNSPEC section.
2) Items (2) to (6) may be defaulted, in which case linear interpolation will be used to calculate the missing values. In addition, OVISCOMP, item (6),may be completely defaulted, which sets this data to zero.
3) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.113: PVCO Keyword Description

Example

```
--
--        OIL PVT TABLE FOR LIVE OIL
--
PVCO
--        PSAT       RS         BO        VISC     OIL       OIL
--        PSIA       MSCF/STB   RB/STB    CPOISE   COMPRES   VISCOS
--        --------   --------   -------   ------   -------   ------
             14.7     0.0010    1.05340   1.7230   3.0E-5    1*
            500.0     0.0890    1.08890   1.1670   1*        1*
           1000.0     0.2060    1.13850   0.8570   1*        1*
           1500.0     0.3360    1.19640   0.6840   1*        1*
           2000.0     0.4750    1.26110   0.5750   1*        1*
           2500.0     0.6220    1.33160   0.5000   1*        1*
           3000.0     0.7750    1.40740   0.4450   1*        1*
           3500.0     0.9330    1.48790   0.4020   1*        1*
           4000.0     1.0960    1.57280   0.3680   1*        1*
           4258.0     1.1800    1.61760   0.3530   1*        1*
           4500.0     1.2630    1.66190   0.3400   1*        1*
           5000.0     1.4340    1.75480   0.3170   1*        1*
           5500.0     1.6060    1.85020   0.2980   1*        1*     / TABLE NO. 01
--
--        PSAT       RS         BO        VISC     OIL       OIL
--        PSIA       MSCF/STB   RB/STB    CPOISE   COMPRES   VISCOS
--        --------   --------   -------   ------   -------   ------
             14.7     0.0010    1.05340   1.7230   3.0E-5    1*
            500.0     0.0890    1.08890   1.1670   1*        1*
           1000.0     0.2060    1.13850   0.8570   1*        1*
           1500.0     0.3360    1.19640   0.6840   1*        1*
           2000.0     0.4750    1.26110   0.5750   1*        1*
           2500.0     0.6220    1.33160   0.5000   1*        1*
           3000.0     0.7750    1.40740   0.4450   1*        1*
           3500.0     0.9330    1.48790   0.4020   1*        1*
           4000.0     1.0960    1.57280   0.3680   1*        1*
           4258.0     1.1800    1.61760   0.3530   1*        1*
           4500.0     1.2630    1.66190   0.3400   1*        1*
           5000.0     1.4340    1.75480   0.3170   1*        1*
           5500.0     1.6060    1.85020   0.2980   1*        1*     / TABLE NO. 02
```

The example defines two live oil PVT tables with constant compressibility above the saturation pressure, and assumes that NTPVT equals two on the TABDIMS keyword in the RUNSPEC section.

8.3.226 PVDG - GAS PVT PROPERTIES FOR DRY GAS

Description

PVDG defines the gas PVT properties for dry gas[221]. If the gas has a constant and uniform vaporized oil concentration, Condensate-Gas Ratio ("CGR"), and if the reservoir pressure never drops below the saturation pressure (dew point pressure), then the model can be run more efficiently by omitting the OIL and VAPOIL keywords from the RUNSPEC section, treating the gas as a dry gas, and defining a constant Rv (CGR) value with keyword RVCONST or RVCONSTT in the PROPS section. This results in the model being run with as a dry gas problem with no active oil (condensate) phase. However, OPM Flow takes into account the constant Rv in the calculations and reporting.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	PRESS	A columnar vector of real monotonically increasing down the column values that defines the gas phase pressure.			
		psia	barsa	atma	None
2	GFVF	A columnar vector of real decreasing down the column values that defines the corresponding gas phase formation volume factor.			
		rb/Mscf	rm^3/sm^3	rcc/scc	None
3	GVISC	A columnar vector of real increasing down the column values that defines the corresponding gas phase viscosity.			
		cP	cP	cP	None

Notes:

1) The keyword is followed by NTPVT tables as declared on the TABDIMS keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NPPVT rows as declared on the TABDIMS keyword in the RUNSPEC section.
3) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.114: PVDG Keyword Description

Note that provided the first table has been entered, subsequent tables may be defaulted, in this case the prior table is copied to the current table. See the second example for an illustration on how to use this feature.

See also the RVCONST and RVCONSTT keywords to define the constant Rv for dry gas.

[221] *Natural gas that occurs in the absence of condensate or liquid hydrocarbons, or gas that had condensable hydrocarbons removed, is called dry gas. It is primarily methane with some intermediates. The hydrocarbon mixture is solely gas in the reservoir and there is no liquid (condensate surface liquid) formed either in the reservoir or at surface. The term dry indicates that the gas does not contain heavier hydrocarbons to form liquids at the surface conditions. Dry gas typically has GOR's greater than 100,000 scf/stb or 18,000 Sm3/m3.*

Example

The first example below defines two dry gas PVT tables assuming NTPVT equals two and NPPVT is greater than or equal to 22 on the TABDIMS keyword in the RUNSPEC section.

```
--
--        GAS PVT TABLE FOR DRY GAS
--
PVDG
--        PRES      BG          VISC
--        PSIA      RB/MSCF     CPOISE
--        ------    --------    ------
            14.7   197.8092    0.0129
            50.0    65.9364    0.0130
           100.0    31.6495    0.0130
           230.0    13.8813    0.0131
           460.0     6.8210    0.0132
           690.0     4.4703    0.0135
           920.0     3.2968    0.0138
          1150.0     2.6113    0.0141
          1380.0     2.1560    0.0145
          1610.0     1.8316    0.0150
          1840.0     1.5952    0.0155
          2070.0     1.4129    0.0161
          2300.0     1.2700    0.0167
          2372.0     1.2305    0.0169
          2530.0     1.1551    0.0174
          2760.0     1.0621    0.0181
          2990.0     0.9841    0.0189
          3220.0     0.9190    0.0196
          3450.0     0.8638    0.0204
          4500.0     0.6910    0.0242
          6000.0     0.5616    0.0293                    / TABLE N0. 01
--
--        PRES      BG          VISC
--        PSIA      RB/MSCF     CPOISE
--        ------    --------    ------
            14.7   265.0126    0.0133
            50.0    66.2531    0.0133
           100.0    33.1266    0.0133
           230.0    14.4552    0.0134
           460.0     7.0357    0.0136
           690.0     4.6493    0.0138
           920.0     3.4417    0.0140
          1150.0     2.7227    0.0144
          1380.0     2.2522    0.0147
          1610.0     1.9158    0.0151
          1840.0     1.6702    0.0156
          2070.0     1.4805    0.0162
          2300.0     1.3317    0.0167
          2372.0     1.2927    0.0169
          2530.0     1.2119    0.0173
          2760.0     1.1135    0.0180
          2990.0     1.0325    0.0187
          3220.0     0.9637    0.0194
          3450.0     0.9055    0.0201
          4500.0     0.7228    0.0236
          6000.0     0.5837    0.0285                    / TABLE N0. 02
```

The second example defines four dry gas PVT tables assuming NTPVT equals four and NPPVT is greater than or equal to 22 on the TABDIMS keyword in the RUNSPEC section. Here table two defaults to table one, and table four defaults to table three.

```
--
--        GAS PVT TABLE FOR DRY GAS
--
PVDG
--        PRES      BG           VISC
--        PSIA      RB/MSCF      CPOISE
--        ------    --------     ------
            14.7    197.8092     0.0129
            50.0     65.9364     0.0130
           100.0     31.6495     0.0130
           230.0     13.8813     0.0131
           460.0      6.8210     0.0132
           690.0      4.4703     0.0135
           920.0      3.2968     0.0138
          1150.0      2.6113     0.0141
          1380.0      2.1560     0.0145
          1610.0      1.8316     0.0150
          1840.0      1.5952     0.0155
          2070.0      1.4129     0.0161
          2300.0      1.2700     0.0167
          2372.0      1.2305     0.0169
          2530.0      1.1551     0.0174
          2760.0      1.0621     0.0181
          2990.0      0.9841     0.0189
          3220.0      0.9190     0.0196
          3450.0      0.8638     0.0204
          4500.0      0.6910     0.0242
          6000.0      0.5616     0.0293                    / TABLE N0. 01
                                                           / TABLE N0. 02
--
--        PRES      BG           VISC
--        PSIA      RB/MSCF      CPOISE
--        ------    --------     ------
            14.7    265.0126     0.0133
            50.0     66.2531     0.0133
           100.0     33.1266     0.0133
           230.0     14.4552     0.0134
           460.0      7.0357     0.0136
           690.0      4.6493     0.0138
           920.0      3.4417     0.0140
          1150.0      2.7227     0.0144
          1380.0      2.2522     0.0147
          1610.0      1.9158     0.0151
          1840.0      1.6702     0.0156
          2070.0      1.4805     0.0162
          2300.0      1.3317     0.0167
          2372.0      1.2927     0.0169
          2530.0      1.2119     0.0173
          2760.0      1.1135     0.0180
          2990.0      1.0325     0.0187
          3220.0      0.9637     0.0194
          3450.0      0.9055     0.0201
          4500.0      0.7228     0.0236
          6000.0      0.5837     0.0285                    / TABLE N0. 03
                                                           / TABLE N0. 04
```

There is no terminating "/" for this keyword.

8.3.227 PVDO – OIL PVT PROPERTIES FOR DEAD OIL

Description

PVDO defines the oil PVT properties for dead oil[222]. If the oil has a constant and uniform dissolved gas concentration, Gas-Oil Ratio ("GOR"), and if the reservoir pressure never drops below the saturation pressure (bubble point pressure), then the model can be run more efficiently by omitting the GAS and DISGAS keywords from the RUNSPEC section, treating the oil as a dead oil, and defining a constant Rs (GOR) value with keyword RSCONST or RSCONSTT in the PROPS section. This results in the model being run as a dead oil problem with no active gas phase. However, OPM Flow takes into account the constant Rs in the calculations and reporting.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	PRESS	A columnar vector of real monotonically increasing down the column values that defines the oil phase pressure.			
		psia	barsa	atma	None
2	OFVF	A columnar vector of real decreasing down the column values that defines the corresponding oil phase formation volume factor.			
		rb/stb	rm^3/sm^3	rcc/scc	None
3	OVISC	A columnar vector of real increasing down the column values that defines the corresponding oil phase viscosity.			
		cP	cP	cP	None

Notes:

1) The keyword is followed by NTPVT tables as declared on the TABDIMS keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NPPVT rows as declared on the TABDIMS keyword in the RUNSPEC section.
3) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.115: PVDO Keyword Description

Note that provided the first table has been entered, subsequent tables may be defaulted, in this case the prior table is copied to the current table. See the second example for an illustration on how to use this feature.

See also the RSCONST and RSCONSTT keywords to define the constant Rs for dead oil and PVCDO as an alternative keyword to enter the dead oil properties.

[222] *"Dead" oil is oil that it contains no dissolved gas or a relatively thick oil or residue that has lost its volatile components.*

Example

The example below defines two dead oil PVT tables with variable viscosity and compressibility with respect to pressure, and assumes that NTPVT equals two and NPPVT is greater than or equal to eight on the TABDIMS keyword in the RUNSPEC section.

```
--
--      OIL PVT TABLE FOR DEAD OIL
--
PVDO
--      PSAT       BO         VISC
--      PSIA       RB/STB     CPOISE
--      --------   -------    ------
           400     1.0102      1.16
          1200     1.0040      1.164
          2000     0.9960      1.167
          2800     0.9880      1.172
          3600     0.9802      1.177
          4400     0.9724      1.181
          5200     0.9646      1.185
          5600     0.9607      1.19                    / TABLE NO. 01
--      --------   -------    ------
           800     1.0255      1.14
          1600     1.0172      1.14
          2400     1.0091      1.14
          3200     1.0011      1.14
          4000     0.9931      1.14
          4800     0.9852      1.14
          5600     0.9774      1.14                    / TABLE NO. 02
```

The second example defines four dead oil PVT tables with variable viscosity and compressibility with respect to pressure, and assumes that NTPVT equals four and NPPVT is greater than or equal to eight on the TABDIMS keyword in the RUNSPEC section. Here table two defaults to table one, and table four defaults to table three.

```
--
--      OIL PVT TABLE FOR DEAD OIL
--
PVDO
--      PSAT       BO         VISC
--      PSIA       RB/STB     CPOISE
--      --------   -------    ------
           400     1.0102      1.16
          1200     1.0040      1.164
          2000     0.9960      1.167
          2800     0.9880      1.172
          3600     0.9802      1.177
          4400     0.9724      1.181
          5200     0.9646      1.185
          5600     0.9607      1.19                    / TABLE NO. 01
                                                       / TABLE NO. 02
           800     1.0255      1.14
          1600     1.0172      1.14
          2400     1.0091      1.14
          3200     1.0011      1.14
          4000     0.9931      1.14
          4800     0.9852      1.14
          5600     0.9774      1.14                    / TABLE NO. 03
                                                       / TABLE NO. 04
```

There is no terminating "/" for this keyword.

8.3.228 PVDS - SOLVENT PVT PROPERTIES FOR THE SOLVENT MODEL

Description

PVDS defines the solvent PVT properties for use with SOLVENT option. The solvent is treated as an additional dry gas phase within the model. This keyword should only be used if the SOLVENT model has been invoked in the RUNSPEC section.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	PRESS	A columnar vector of real monotonically increasing down the column values that defines the solvent phase pressure.			
		psia	barsa	atma	None
2	GFVF	A columnar vector of real decreasing down the column values that defines the corresponding solvent phase formation volume factor.			
		rb/Mscf	rm^3/sm^3	rcc/scc	None
3	GVISC	A columnar vector of real increasing down the column values that defines the corresponding solvent phase viscosity.			
		cP	cP	cP	None

Notes:

1) The keyword is followed by NTPVT tables as declared on the TABDIMS keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NPPVT rows as declared on the TABDIMS keyword in the RUNSPEC section.
3) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.116: PVDS Keyword Description

Example

```
--
--       GAS SOLVENT PVT TABLE
--
PVDS
--       PRES     BG          VISC
--       PSIA     RB/MSCF     CPOISE
--       ------   --------    ------
          700.0     4.4703    0.0135
          920.0     3.2968    0.0138
         1150.0     2.6113    0.0141
         1380.0     2.1560    0.0145
         1610.0     1.8316    0.0150
         1840.0     1.5952    0.0155
         2070.0     1.4129    0.0161
         2300.0     1.2700    0.0167
         2372.0     1.2305    0.0169
         2530.0     1.1551    0.0174
         2760.0     1.0621    0.0181
         2990.0     0.9841    0.0189
         3220.0     0.9190    0.0196
         3450.0     0.8638    0.0204
         4500.0     0.6910    0.0242
         6000.0     0.5616    0.0293                    / TABLE N0. 01
--
--       PRES     BG          VISC
--       PSIA     RB/MSCF     CPOISE
--       ------   --------    ------
          700.0     4.6493    0.0138
          920.0     3.4417    0.0140
         1150.0     2.7227    0.0144
         1380.0     2.2522    0.0147
         1610.0     1.9158    0.0151
         1840.0     1.6702    0.0156
         2070.0     1.4805    0.0162
         2300.0     1.3317    0.0167
         2372.0     1.2927    0.0169
         2530.0     1.2119    0.0173
         2760.0     1.1135    0.0180
         2990.0     1.0325    0.0187
         3220.0     0.9637    0.0194
         3450.0     0.9055    0.0201
         4500.0     0.7228    0.0236
         6000.0     0.5837    0.0285                    / TABLE N0. 02
```

The above example defines two solvent PVT tables assuming NTPVT equals two and NPPVT is greater than or equal to 16 on the TABDIMS keyword in the RUNSPEC section.

There is no terminating "/" for this keyword.

8.3.229 PVTG - GAS PVT PROPERTIES FOR WET GAS WITH VAPORIZED OIL

Description

PVTG defines the gas PVT properties for wet gas[223]. This keyword should be used when the VAPOIL keyword has be declared in the RUNSPEC section indicating that that vaporized oil (more commonly referred to as condensate) is present in the wet gas phase. The keyword may be used for gas-water and oil-water-gas input decks that contain the oil and gas phases.

No.	Name		Description			Default
			Field	Metric	Laboratory	
1	PRESS		A real monotonically increasing down the column vector that defines the gas phase pressure, associated with the saturated condensate-gas ratio ("CGR") or Rv, the gas formation volume factor and the gas viscosity for the corresponding pressure for the stated saturated RVS. For a given PRESS the variability of the gas formation volume factor and the gas viscosity with respect to the under-saturated Rv is optionally included as a sub table under RVU, FVFU and VISU columns, that is it is not necessary to repeat PRESS for each sub table entry. However, each sub table must be terminated by a "/". The under saturated Rv entries are optional, except for perhaps the last PRESS entry to define the PVT properties above the initial saturation pressure.			
			psia	barsa	atma	None
2	RVS	RVU	A columnar vector of real positive number for both the saturated (RVS) and under saturated (RVU) Rv sub table entries. The RVS entry on the main table is the saturated CGR at the pressure indicated by PRESS and may be increasing or decreasing in value as PRESS varies. Subsequent under-saturated Rv for a sub table at the given PRESS, as defined by RVU, are monotonically decreasing for entries in a given sub table.			
			stb/Mscf	sm^3/sm^3	scc/scc	None
3	FVFS	FVFU	A columnar vector of real decreasing down the column values that defines the corresponding gas phase formation volume factor for a given pressure (PRESS) and for a given Rv (either RVS or RVU).			
			rb/Mscf	rm^3/sm^3	rcc/scc	None
4	VISS	VISU	VISS a columnar vector of real increasing down the column values that defines the corresponding gas phase viscosity for a given pressure (PRESS) and for a given RVS. VISU a columnar vector of real decreasing from VISS down the column values that defines the corresponding gas phase viscosity for a given pressure (PRESS) and for a given RVU.			None

[223] *Natural gas that contains significant heavy hydrocarbons such as propane, butane and other liquid hydrocarbons is known as wet gas or rich gas. The general rule of thumb is if the gas contains less methane (typically less than 85% methane) and more ethane, and other more complex hydrocarbons, it is labeled as wet gas. Wet gas normally has GOR's less than 100,000 scf/stb or 18,000 Sm3/m3, with the condensate having a gravity greater than 50 oAPI.*

No.	Name		Description			Default
			Field	Metric	Laboratory	
			cP	cP	cP	

Notes:

1) The keyword is followed by NTPVT tables as declared on the TABDIMS keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NPPVT rows and NRPVT Rv entries as declared on the TABDIMS keyword in the RUNSPEC section.
3) Apart from the PRESS data there must be same number of entries for each column.
4) Each sub table defining the under saturated gas properties must be terminated by "/".
5) Each main table is terminated by a single "/" on a separate line and there is no "/" terminator for the keyword.

Table 8.117: PVTG Keyword Description

Note that provided the first table has been entered, subsequent tables may be defaulted, in this case the prior table is copied to the current table. See the second example for an illustration on how to use this feature.

Note

If the VAPWAT keyword in the RUNSPEC section is also present in the input deck, then the PVTG keyword in the PROPS section should be used to define the gas properties as function of pressure and RV, assuming water-saturated gas. Also, in this case, the PVTGW keyword, also in the PROPS section, should also be in the input deck. In this case, PVTGW defines the gas properties as function of pressure and RVW, assuming oil-saturated gas.

Example

The first example defines two wet gas PVT tables assuming NTPVT equals two, NPPVT is greater than or equal to eight, and NRPVT greater than or equal to two on the TABDIMS keyword in the RUNSPEC section.

```
--
--       GAS PVT TABLE FOR WET GAS WITH VAPORIZED OIL
--
PVTG
--       PRES       RV           BG           VISC
--       BARSA    SM^3/SM^3    RM^3/SM^3       CPOISE
--      ------    ---------    -------      ------
           20     0.000132      0.042340      0.01344
                  0             0.042310      0.01389      /
           40     0.000124      0.020460      0.01420
                  0             0.020430      0.01450      /
           60     0.000126      0.013280      0.01526
                  0             0.013250      0.01532      /
           80     0.000135      0.009770      0.01660
                  0             0.009730      0.01634      /
          100     0.000149      0.007730      0.01818
                  0             0.007690      0.01752      /
          120     0.000163      0.006426      0.01994
                  0             0.006405      0.01883      /
          140     0.000191      0.005541      0.02181
                  0             0.005553      0.02021      /
          160     0.000225      0.004919      0.02370
                  0             0.004952      0.02163      /
                                                           / TABLE NO. 1
--       PRES       RV           BG           VISC
--       BARSA    SM^3/SM^3    RM^3/SM^3       CPOISE
--      ------    ---------    -------      ------
           20     0.000132      0.042340      0.01344
           40     0.000124      0.020460      0.01420
           60     0.000126      0.013280      0.01526
           80     0.000135      0.009770      0.01660
          100     0.000149      0.007730      0.01818
          120     0.000163      0.006426      0.01994
          140     0.000191      0.005541      0.02181
          160     0.000225      0.004919      0.02370
                  0             0.004952      0.02163      /
                                                           / TABLE NO. 2
```

The second example defines four wet gas PVT tables assuming NTPVT equals four, NPPVT is greater than or equal to eight, and NRPVT greater than or equal to two on the TABDIMS keyword in the RUNSPEC section. Here table two defaults to table one, and table four defaults to table three.

```
--
--        GAS PVT TABLE FOR WET GAS WITH VAPORIZED OIL
--
PVTG
--        PRES        RV           BG           VISC
--        BARSA     SM^3/SM^3     RM^3/SM^3       CPOISE
--       ------     ---------     -------       ------
            20      0.000132      0.042340      0.01344
                    0             0.042310      0.01389      /
            40      0.000124      0.020460      0.01420
                    0             0.020430      0.01450      /
            60      0.000126      0.013280      0.01526
                    0             0.013250      0.01532      /
            80     0.000135      0.009770      0.01660
                    0             0.009730      0.01634      /
           100      0.000149      0.007730      0.01818
                    0             0.007690      0.01752      /
           120      0.000163      0.006426      0.01994
                    0             0.006405      0.01883      /
           140      0.000191      0.005541      0.02181
                    0             0.005553      0.02021      /
           160      0.000225      0.004919      0.02370
                    0             0.004952      0.02163      /
                                                             / TABLE NO. 1
                                                             / TABLE NO. 2
--        PRES        RV           BG           VISC
--        BARSA     SM^3/SM^3     RM^3/SM^3       CPOISE
--       ------     ---------     -------       ------
            20      0.000132      0.042340      0.01344
            40      0.000124      0.020460      0.01420
            60      0.000126      0.013280      0.01526
            80     0.000135      0.009770      0.01660
           100      0.000149      0.007730      0.01818
           120      0.000163      0.006426      0.01994
           140      0.000191      0.005541      0.02181
           160      0.000225      0.004919      0.02370
                    0             0.004952      0.02163      /
                                                             / TABLE NO. 3
                                                             / TABLE NO. 4
```

Notice that in both examples there is no terminating "/" for this keyword only for a table and a sub table.

8.3.230 PVTGW - GAS PVT PROPERTIES FOR DRY GAS WITH VAPORIZED WATER

Description

PVTGW defines the gas PVT properties for dry gas[224] with vaporized water. This keyword should be used when the VAPWAT keyword has be declared in the RUNSPEC section indicating that that vaporized water is present in the dry gas phase. The keyword may be used for gas-water and oil-water-gas input decks that contain the dry gas and vaporized water phases.

Note

This is an OPM Flow specific keyword for the simulator's Water Vaporization Model that is activated by declaring that vaporized water is present in the run.

No.	Name		Description			Default
			Field	Metric	Laboratory	
1	PRESS		A real monotonically increasing down the column vector that defines the gas phase pressure, associated with the corresponding saturated water-gas ratio ("WGR") or Rw, the gas formation volume factor, and the gas viscosity for the stated saturated RWS. For a given PRESS the variability of the gas formation volume factor and the gas viscosity with respect to the under-saturated Rw is optionally included as a sub table under RWU, FVFU and VISU columns, that is it is not necessary to repeat PRESS for each sub table entry. However, each sub table must be terminated by a "/". The under saturated Rw entries are optional, except for perhaps the last PRESS entry to define the PVT properties above the initial saturation pressure.			
			psia	barsa	atma	None
2	RWS	RWU	A columnar vector of real positive numbers for both the saturated (RWS) and under saturated (RWU) Rw sub table entries. The RWS entry on the main table is the saturated WGR at the pressure indicated by PRESS and may be increasing or decreasing in value as PRESS varies. Subsequent under-saturated Rw for a sub table at the given PRESS, as defined by RWU, are monotonically decreasing for entries in a given sub table.			
			stb/Mscf	sm^3/sm^3	scc/scc	None
3	FVFS	FVFU	A columnar vector of real decreasing down the column values that defines the corresponding gas phase formation volume factor for a given pressure (PRESS) and for a given Rw (either RWS or RWU).			None

[224] *Natural gas that occurs in the absence of condensate or liquid hydrocarbons, or gas that had condensable hydrocarbons removed, is called dry gas. It is primarily methane with some intermediates. The hydrocarbon mixture is solely gas in the reservoir and there is no liquid (condensate surface liquid) formed either in the reservoir or at surface. The term dry indicates that the gas does not contain heavier hydrocarbons to form liquids at the surface conditions. Dry gas typically has GOR's greater than 100,000 scf/stb or 18,000 Sm3/m3.*

No.	Name		Description			Default
			Field	Metric	Laboratory	
			rb/Mscf	rm^3/sm^3	rcc/scc	
4	VISS	VISU	VISS a columnar vector of real increasing down the column values that defines the corresponding gas phase viscosity for a given pressure (PRESS) and for a given RWS. VISU a columnar vector of real decreasing from VISS down the column values that defines the corresponding gas phase viscosity for a given pressure (PRESS) and for a given RWU.			
			cP	cP	cP	None

Notes:

1) The keyword is followed by NTPVT tables as declared on the TABDIMS keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NPPVT rows and NRPVT Rw entries as declared on the TABDIMS keyword in the RUNSPEC section.
3) Apart from the PRES data there must be same number of entries for each column.
4) Each sub table defining the under saturated gas properties must be terminated by "/".
5) Each main table is terminated by a single "/" on a separate line and there is no "/" terminator for the keyword.

Table 8.118: PVTGW Keyword Description

Note that provided the first table has been entered, subsequent tables may be defaulted, in this case the prior table is copied to the current table.

Note

If both the VAPWAT and VAPOIL keywords have been declared in the RUNSPEC section indicating that both vaporized water and vaporized oil are present in the wet gas, then the PVTGW keyword should be used along with the PVTG keyword in the PROPS section to fully define the wet gas PVT properties. The PVTGW keyword should be used to define the gas properties as a function of pressure and water-gas ratio (RVW), assuming oil-saturated gas. The PVTG keyword should be used to define the gas properties as a function of pressure and oil-gas ratio (RV), assuming water-saturated gas.

Alternatively, the PVTGWO keyword in the PROPS section may be used instead of the PVTGW and PVTG keywords to fully define the wet gas PVT properties.

See also the PVTG keyword in the PROPS section that defines the wet gas PVT for when vaporized oil is present in the gas phase. Alternatively, the PVTGWO keyword in the PROPS section may be utilized instead of PVTG and PVTGW to fully define the wet gas PVT properties, for when both vaporized oil and water are present in the gas phase.

Example

```
--
--       GAS PVT TABLE FOR DRY GAS WITH VAPORIZED WATER (OPM FLOW KEYWORD)
--
PVTGW
--       PRES       RW           BG           VISC
--    PSIA   SM^3/SM^3   RM^3/SM^3    CPOISE
--      ------    ---------    -------       ------
          300     0.000479      0.042340      0.01344
                  0             0.042310      0.01389      /
          600     0.000469      0.020460      0.01420
                  0             0.020430      0.01450      /
          900     0.000403      0.013280      0.01526
                  0             0.013250      0.01532      /
         1200     0.000354      0.009770      0.01660
                  0             0.009730      0.01634      /
         1500     0.000272      0.007730      0.01818
                  0             0.007690      0.01752      /
         1800     0.000225      0.006426      0.01994
                  0             0.006405      0.01883      /
         2100     0.000191      0.005541      0.02181
                  0             0.005553      0.02021      /
         2400     0.000163      0.004919      0.02370
                  0             0.004952      0.02163      /
                                                           / TABLE NO. 1
--       PRES       RW           BG           VISC
--    PSIA   SM^3/SM^3   RM^3/SM^3    CPOISE
--      ------    ---------    -------       ------
          300     0.000479      0.042340      0.01344
          600     0.000469      0.020460      0.01420
          900     0.000403      0.013280      0.01526
         1200     0.000354      0.009770      0.01660
         1500     0.000272      0.007730      0.01818
         1800     0.000225      0.006426      0.01994
         2100     0.000191      0.005541      0.02181
         2400     0.000163      0.004919      0.02370      /
                                                           / TABLE NO. 2
```

The above example defines two dry gas PVT tables assuming NTPVT equals two and NPPVT is greater than or equal to eight on the TABDIMS keyword in the RUNSPEC section.

Notice that there is no terminating "/" for this keyword only for a table and a sub table.

8.3.231 PVTGWO - GAS PVT PROPERTIES FOR WET GAS WITH VAPORIZED WATER AND OIL

Description

PVTGWO defines the gas PVT properties for wet gas[225] with vaporized water and oil. This keyword should be used when the VAPOIL and VAPWAT keywords have been declared in the RUNSPEC section indicating that vaporized oil and water are present in the wet gas phase. The keyword may be used for oil-water-gas input decks that contain the wet gas with vaporized oil and water phases.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

Note

This is an OPM Flow specific keyword for the simulator's Water Vaporization Model that is activated by declaring that vaporized water is present in the run.

No.	Name		Description			Default
			Field	Metric	Laboratory	
1	PRESS		A real monotonically increasing down the column vector that defines the gas phase pressure, associated with the saturated water-gas ratio ("WGR") or Rw, the saturated condensate-gas ratio ("CGR") or Rv, the gas formation volume factor, and the gas viscosity for the corresponding pressure for the stated saturated RWS. For a given PRESS the variability of the gas formation volume factor and the gas viscosity with respect to the under-saturated Rw and Rv is optionally included as a sub table under RWU, RVU, FVFU and VISU columns, that is it is not necessary to repeat PRESS for each sub table entry. However, each sub table must be terminated by a "/". The under saturated Rw and Rv entries are optional, except for perhaps the last PRESS entry to define the PVT properties above the initial saturation pressure.			
			psia	barsa	atma	None
2	RWS	RWU	A columnar vector of real positive numbers for both the saturated (RWS) and under saturated (RWU) Rw sub table entries. The RWS entry on the main table is the saturated WGR at the pressure indicated by PRESS and may be increasing or decreasing in value as PRESS varies. Subsequent under-saturated Rw for a sub table at the given PRESS, as defined by RWU, are monotonically decreasing for entries in a given sub table.			None

[225] *Natural gas that contains significant heavy hydrocarbons such as propane, butane and other liquid hydrocarbons is known as wet gas or rich gas. The general rule of thumb is if the gas contains less methane (typically less than 85% methane) and more ethane, and other more complex hydrocarbons, it is labeled as wet gas. Wet gas normally has GOR's less than 100,000 scf/stb or 18,000 Sm3/m3, with the condensate having a gravity greater than 50 oAPI.*

No.	Name		Description			Default
			Field	Metric	Laboratory	
			stb/Mscf	sm^3/sm^3	rcc/scc	
3	RVS	RVU	A columnar vector of real positive numbers for both the saturated (RVS) and under saturated (RVU) Rv sub table entries. The RVS entry on the main table is the saturated CGR at the pressure indicated by PRESS and may be increasing or decreasing in value as PRESS varies. Subsequent under-saturated Rv for a sub table at the given PRESS, as defined by RVU, are monotonically decreasing for entries in a given sub table.			
			stb/Mscf	sm^3/sm^3	rcc/scc	None
4	FVFS	FVFU	A columnar vector of real decreasing down the column values that defines the corresponding gas phase formation volume factor for a given pressure (PRESS) and for a given Rw (either RWS or RWU) and Rv (either RVS or RVU).			
			rb/Mscf	rm^3/sm^3	rcc/scc	None
5	VISS	VISU	VISS a columnar vector of real increasing down the column values that defines the corresponding gas phase viscosity for a given pressure (PRESS) and for a given RWS and RVS. VISU a columnar vector of real decreasing from VISS down the column values that defines the corresponding gas phase viscosity for a given pressure (PRESS) and for a given RWU and RVU.			
			cP	cP	cP	None

Notes:

1) The keyword is followed by NTPVT tables as declared on the TABDIMS keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NPPVT rows and NRPVT Rw entries as declared on the TABDIMS keyword in the RUNSPEC section.
3) Apart from the PRES data there must be same number of entries for each column.
4) Each sub table defining the under saturated gas properties must be terminated by "/".
5) Each main table is terminated by a single "/" on a separate line and there is no "/" terminator for the keyword.

Table 8.119: PVTGWO Keyword Description

See also the PVTG keyword in the PROPS section that defines the wet gas PVT properties when vaporized oil is present in the gas phase, and the PVTGW keyword in the PROPS section that defines the dry gas PVT properties when vaporized water is present in the gas phase. As an alternative to using the PVTGWO keyword, the PVTGW and PVTG keywords may be combined to fully define the wet gas PVT properties if both vaporized water and vaporized oil are present in the gas phase.

Example

```
--
--       GAS PVT TABLE FOR WET GAS WITH VAPORIZED WATER & OIL (OPM FLOW KEYWORD)
--
PVTGWO
--       PRES      RW          RV          BG          VISC
--       PSIA    STB/MSCF    STB/MSCF    RB/MSCF     CPOISE
--      ------   --------    ---------   -------     ------
          300    0.000479    0.000132    0.042340    0.01344
                 0           0           0.042310    0.01389    /
          600    0.000469    0.000124    0.020460    0.01420
                 0           0           0.020430    0.01450    /
          900    0.000403    0.000126    0.013280    0.01526
                 0           0           0.013250    0.01532    /
         1200    0.000354    0.000135    0.009770    0.01660
                 0           0           0.009730    0.01634    /
         1500    0.000272    0.000149    0.007730    0.01818
                 0           0           0.007690    0.01752    /
         1800    0.000225    0.000163    0.006426    0.01994
                 0           0           0.006405    0.01883    /
         2100    0.000191    0.000191    0.005541    0.02181
                 0           0           0.005553    0.02021    /
         2400    0.000163    0.000225    0.004919    0.02370
                 0           0           0.004952    0.02163    /
                                                                 / TABLE NO. 1
--       PRES      RW          RV          BG          VISC
--       PSIA    STB/MSCF    STB/MSCF    RB/MSCF     CPOISE
--      ------   --------    ---------   -------     ------
          300    0.000479    0.000132    0.042340    0.01344
          600    0.000469    0.000124    0.020460    0.01420
          900    0.000403    0.000126    0.013280    0.01526
         1200    0.000354    0.000135    0.009770    0.01660
         1500    0.000272    0.000149    0.007730    0.01818
         1800    0.000225    0.000163    0.006426    0.01994
         2100    0.000191    0.000191    0.005541    0.02181
         2400    0.000163    0.000225    0.004919    0.02370    /
                                                                 / TABLE NO. 2
```

The above example defines two wet PVT tables assuming NTPVT equals two and NPPVT is greater than or equal to eight on the TABDIMS keyword in the RUNSPEC section.

Notice that there is no terminating "/" for this keyword only for a table and a sub table.

8.3.232 PVTO - OIL PVT PROPERTIES FOR LIVE OIL

Description

PVTO defines the oil PVT properties for live oil[226] and the keyword should only be used if the there is both oil and gas phases in the model. This keyword should be used when the DISGAS keyword has be declared in the RUNSPEC section indicating that dissolved gas (more commonly referred to as solution gas) is present in the oil. The keyword may be used for oil-water and oil-water-gas input decks.

No.	Name		Description			Default
			Field	Metric	Laboratory	
1	RS		A real monotonically increasing down the column values that defines the saturated gas-oil ratio ("GOR") or Rs, that defines the oil formation volume factor and the oil viscosity for the tabulated corresponding pressure for stated saturated RS. For a given RS the variability of the oil formation volume factor and the oil viscosity with respect to the saturated RS and pressure is optionally included as a sub table under PRSU, FVFU and VISU columns, that is it is not necessary to repeat RS for each sub table entry. However, each sub table must be terminated by a "/". The under-saturated PRSU entries are optional, except for perhaps the last RS entry to define the PVT properties above the initial saturation pressure. If there are no following under-saturated PRSU entries then the RS entry row should be terminated by a "/", if there are under-saturated PRSU entries then the last PRSU entry row should be terminated by a "/".			
			Mscf/stb	sm^3/sm^3	scc/scc	None
2	PRSS	PRSU	PRSS is a real columnar vector of real monotonically increasing down the column values that defines the oil phase saturation pressure (bubble-point pressure), that defines the oil formation volume factor and the oil viscosity for the corresponding PRSS pressure for a given saturated RS. PRSU is a real columnar vector of real monotonically increasing down the column values that defines the oil phase under-saturated pressure that defines the oil formation volume factor and the oil viscosity for the corresponding PRSU pressure for a given saturated RS. Note that PRSU should be greater than PRSS.			
			psia	barsa	atma	None
3	FVFS	FVFU	FVFS is a columnar vector of real increasing down the column values that defines the corresponding oil phase saturated formation volume factor for a given pressure (PRSS) and for a given RS. FVFU is a columnar vector of real decreasing down the column values that defines the corresponding oil phase under-saturated formation volume factor for a given pressure (PRSU) and for a given RS.			
			rb/stb	rm^3/sm^3	rcc/scc	None

[226] *"Live" oil is oil that contains gas in solution, which is normally the case for most conventional oil reservoirs. However, for oil reservoirs classified as heavy oil reservoirs, the in situ dissolved gas may be negligible and oil would then be classified as gas-free oil which is commonly referred to as "dead" oil.*

No.	Name		Description			Default
			Field	Metric	Laboratory	
4	VISS	VISU	VISS a columnar vector of real increasing down the column values that defines the corresponding oil phase saturated viscosity for a given pressure (PRSS) and for a given RS. If this is the only entry for a given RS and PRSS then the record should be terminate by a "/". VISU a columnar vector of real decreasing from VISS down the column values that defines the corresponding oil phase under-saturated viscosity for a given pressure (PRSU) and for a given RS. If this is the only entry for a given RS and PRSU then the record should be terminate by a "/".			
			cP	cP	cP	None

Notes:

1) The keyword is followed by NTPVT tables as declared on the TABDIMS keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NPPVT rows and NRPVT Rs entries as declared on the TABDIMS keyword in the RUNSPEC section.
3) Apart from the RS data there must be same number of entries for each column.
4) Each sub table defining the under saturated oil properties must be terminated by "/".
5) Each main table is terminated by a single "/" on a separate line and there is no "/" terminator for the keyword.

Table 8.120: PVTO Keyword Description

Note that provided the first table has been entered, subsequent tables may be defaulted, in this case the prior table is copied to the current table. See the second example for an illustration on how to use this feature.

Example

The first example defines live oil PVT tables assuming NTPVT equals two, NPPVT is greater than or equal to two, and NRPVT is greater than or equal to 18 on the TABDIMS keyword in the RUNSPEC section.

```
--
--       OIL PVT TABLE FOR LIVE OIL
--
PVTO
--       RS        PSAT       BO        VISC
--       MSCF/STB  PSIA       RB/STB    CPOISE
--       --------  --------   -------   ------
          0.0010      14.7    1.05340   1.7230  /
          0.0890     500.0    1.08890   1.1670  /
          0.2060    1000.0    1.13850   0.8570  /
          0.3360    1500.0    1.19640   0.6840  /
          0.4750    2000.0    1.26110   0.5750  /
          0.5480    2250.0    1.29570   0.5340  /
          0.6220    2500.0    1.33160   0.5000  /
          0.6980    2750.0    1.36890   0.4700  /
          0.7750    3000.0    1.40740   0.4450  /
          0.8530    3250.0    1.44710   0.4220  /
          0.9330    3500.0    1.48790   0.4020  /
          1.0140    3750.0    1.52980   0.3840  /
          1.0960    4000.0    1.57280   0.3680  /
          1.1800    4258.0    1.61760   0.3530  /
          1.2630    4500.0    1.66190   0.3400  /
          1.3480    4750.0    1.70780   0.3280  /
          1.4340    5000.0    1.75480   0.3170  /
          1.6060    5500.0    1.85020   0.2980
                    6242.0    1.83040   0.3186  /
                                                / TABLE NO. 1
--
--
--       RS        PSAT       BO        VISC
--       MSCF/STB  PSIA       RB/STB    CPOISE
--       --------  --------   -------   ------
          0.0010      14.7    1.05340   1.7230  /
          0.0390     250.0    1.06830   1.4220  /
          0.0890     500.0    1.08890   1.1670  /
          0.1460     750.0    1.11250   0.9850  /
          0.2060    1000.0    1.13850   0.8570  /
          0.2700    1250.0    1.16660   0.7590  /
          0.3360    1500.0    1.19640   0.6840  /
          0.4050    1750.0    1.22800   0.6240  /
          0.4750    2000.0    1.26110   0.5750  /
          0.5480    2250.0    1.29570   0.5340  /
          0.6220    2500.0    1.33160   0.5000  /
          0.6980    2750.0    1.36890   0.4700  /
          0.7750    3000.0    1.40740   0.4450  /
          0.8530    3250.0    1.44710   0.4220  /
          0.9330    3500.0    1.48790   0.4020  /
          1.0140    3750.0    1.52980   0.3840  /
          1.0960    4000.0    1.57280   0.3680  /
          1.1800    4258.0    1.61760   0.3530  /
          1.2630    4500.0    1.66190   0.3400  /
          1.3480    4750.0    1.70780   0.3280  /
          1.4340    5000.0    1.75480   0.3170  /
          1.6060    5500.0    1.85020   0.2980
                    6242.0    1.83040   0.3186  /
                                                / TABLE NO. 2
```

Notice that there must be at least two entries for the last Rs value to enable the simulator to interpolate over the undersaturated pressure region.

The second example defines live oil PVT tables assuming NTPVT equals four, NPPVT is greater than or equal to two, and NRPVT is greater than or equal to 13 on the TABDIMS keyword in the RUNSPEC section. Here, tables two to four all default to table number one.

```
--
--       OIL PVT TABLE FOR LIVE OIL
--
PVTO
--       RS        PSAT       BO         VISC
--       MSCF/STB  PSIA       RB/STB     CPOISE
--       --------  --------   -------    ------
          0.0010      14.7    1.05340    1.7230  /
          0.0890     500.0    1.08890    1.1670  /
          0.2060    1000.0    1.13850    0.8570  /
          0.3360    1500.0    1.19640    0.6840  /
          0.4750    2000.0    1.26110    0.5750  /
          0.5480    2250.0    1.29570    0.5340  /
          0.6220    2500.0    1.33160    0.5000  /
          0.7750    3000.0    1.40740    0.4450  /
          0.9330    3500.0    1.48790    0.4020  /
          1.0960    4000.0    1.57280    0.3680  /
          1.2630    4500.0    1.66190    0.3400  /
          1.4340    5000.0    1.75480    0.3170  /
          1.6060    5500.0    1.85020    0.2980
                    6242.0    1.83040    0.3186  /
                                                 / TABLE NO. 1
                                                 / TABLE NO. 2
                                                 / TABLE NO. 3
                                                 / TABLE NO. 4
```

Again, note that there is no terminating "/" for this keyword only for a table and a sub table.

8.3.233 PVTSOL - OIL PVT PROPERTIES FOR LIVE OIL VERSUS CO2 MASS FRACTION

Description

PVTSOL defines the live oil PVT properties as a function of CO_2 mass fraction. The keyword automatically invokes the simulator's CO2 Dynamic EOR Model[227], that uses a fourth component to model the injected CO_2., for use in evaluating CO_2 Enhanced Oil Recovery ("EOR") projects. Normally CO_2 EOR projects are evaluated via compositional simulators to account for the mass transfer of the various components and phases. Unfortunately, compositional models are computationally expensive compared to the black-oil approach, which for field studies is challenging, especially if an ensemble approach is being used to capture the uncertainties. Previous extended black-oil formulations often poorly represent the PVT properties of the oil-CO_2 mixtures, resulting in poor agreement with the compositional formulation.

To overcome the limitations of the standard four component black-oil formulation, OPM Flow uses an improved extended black-oil formulation, the CO2 Dynamic EOR Model, with the black-oil properties dependent on the fraction of CO2 in the cell, as described by the PVTSOL keyword. This approach models the oil-CO2 mixture more accurately and thus give results closer to the compositional simulator. Data for the keyword should normally be derived from laboratory or numerical slim-tube experiments based on one-dimensional compositional Equation Of State ("EOS") simulations.

Note, if this keyword is absent from the input deck then the CO2 Standard EOR Model is used instead.

No.	Name		Description			Default
			Field	Metric	Laboratory	
1	CO2		A real monotonically increasing down the column values that stipulates the CO2 mass fraction, that defines the oil and gas properties, formation volume factor, viscosity etc., for the tabulated corresponding pressure for the stated CO2 mass fraction. CO2 should be greater than or equal to zero and less than or equal to one. Note it is not necessary to repeat the value of CO2 for the pressure column. However, for a given CO2 mass fraction, the last pressure entry (PRESS) of the sub table should be terminated by a "/".			
			dimensionless	dimensionless	dimensionless	None
2		PRESS	PRESS is a real columnar vector of real monotonically increasing down the column values that defines the oil phase saturation pressure (bubble-point pressure), that defines the oil formation volume factor and the oil viscosity for the corresponding PRESS pressure for a given saturated value of CO2.			
			psia	barsa	atma	None
3		OFVF	OFVF is a columnar vector of real increasing down the column values that defines the corresponding oil phase saturated formation volume factor for a given pressure (PRESS) and for a given saturated value of CO2.			
			rb/stb	rm^3/sm^3	rcc/scc	None

[227] *T. H. Sandve, O. Sævareid and I. Aavatsmark: "Improved Extended Blackoil Formulation -- for CO2 EOR Simulations." in ECMOR XVII – The 17th European Conference on the -- Mathematics of Oil Recovery, September 2020.*

No.	Name		Description			Default
			Field	Metric	Laboratory	
4		GFVF	GFVF is a columnar vector of real decreasing down the column values that defines the corresponding gas phase saturated formation volume factor for a given pressure (PRESS) and for a given saturated value of CO2.			
			rb/Mscf	rm^3/sm^3	rcc/scc	None
5		RS	RS is a real monotonically increasing down the column values that defines the saturated gas-oil ratio ("GOR") or Rs, for the given value of PRESS and for a given saturated value of CO2.			
			Mscf/stb	sm^3/sm^3	scc/scc	None
6		RV	RV is a real monotonically increasing down the column values that defines the saturated condensate-gas ratio ("CGR") or Rv, for the given value of PRESS and for a given saturated value of CO2.			
			stb/Mscf	sm^3/sm^3	scc/scc	None
7		XVOL	XVOL is a real positive value greater than or equal to zero and less than or equal to one, that stipulates the volumetric fraction of CO2 in the oil phase, that is: $$XVOL = \frac{Volume_{oil}(CO_2)}{Volume_{oil}(CO_2) + Volume_{oil}(Gas) + Volume_{oil}(Oil)}$$ where: Volume $_{oil}$ (CO_2) is the surface volume of CO_2 in the oil phase, Volume oil (Gas) is the surface volume of gas in the oil phase, and Volume $_{oil}$ (Oil) is the surface volume of oil in the oil phase.			
			dimensionless	dimensionless	dimensionless	None
8		YVOL	YVOL is a real positive value greater than or equal to zero and less than or equal to one, that defines the volumetric fraction of CO2 in the gas phase, that is: $$YVOL = \frac{Volume_{gas}(CO_2)}{Volume_{gas}(CO_2) + Volume_{gas}(Gas) + Volume_{gas}(Oil)}$$ where: Volume $_{gas}$ (CO_2) is the surface volume of CO_2 in the gas phase, Volume $_{gas}$ (Gas) is the surface volume of gas in the gas phase, and Volume gas (Oil) is the surface volume of oil in the gas phase.			
			dimensionless	dimensionless	dimensionless	None
9		OVISC	OVISC is a columnar vector of real decreasing down the column values that defines the corresponding oil phase saturated viscosity for a given pressure (PRESS) and for a given saturated value of CO2.			
			cP	cP	cP	None

No.	Name		Description			Default
			Field	Metric	Laboratory	
10		GVISC	GVISC is a columnar vector of real increasing down the column values that defines the corresponding gas phase saturated viscosity for a given pressure (PRESS) and for a given saturated value of CO2.			
			cP	cP	cP	None

Notes:

1) The keyword is followed by NTPVT tables as declared on the TABDIMS keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NPPVT rows and NRPVT Rs entries as declared on the TABDIMS keyword in the RUNSPEC section.
3) Except for the CO2 column, there must be same number of entries for each column.
4) Each sub table defining the properties for a given value of CO2 must be terminated by "/" and the keyword itself is terminated by "/".

Table 8.121: PVTSOL Keyword Description

See also the PVDS keyword in the PROPS section that can also be used to model CO2 injection, using the SOLVENT model (CO2 Standard EOR), this is the conventional approach, and does not take into account black-oil properties being dependent on the fraction of CO2.

Note

In the CO2 Dynamic EOR Model the oil properties within a cell are dependent on the CO2 saturation in the cell. However, the model is not restricted to just CO2 utilization, as long as the appropriate oil-injection-gas dependent properties are entered in the model via the PVTSOL keyword.

Thus, N2 or other miscible gases may be used instead of CO2, provided due care is taken in generating the required data for the PVTSOL keyword.

Another example would be a gas-condensate re-cycling project where the produced gas is stripped of the liquids and then re-injected back into the reservoir in order to maintain the reservoir pressure. Here, the CO2 fraction in the PVTSOL keyword would be replaced by the lean injected gas fraction. Again the other properties would be derived from laboratory or numerical slim-tube experiments based on one-dimensional compositional EOS simulations.

As an aside, the scope and accuracy of gas condensate modeling using black-oil reservoir simulators is well established in the industry; both depletion and gas cycling above the dew point can be modeled and yield an adequate match with the results from multi-component compositional simulators. The main inadequacy is with the treatment of gas injection below the dew point where the primary compositional effect, the stripping of liquid components is inversely proportional to their molecular weights, is completely ignored. The standard black-oil model assumes that the saturated hydrocarbon fluid properties are functions of pressure only and disregards any compositional dependence in the saturated fluid PVT properties. Thus, when dry gas is injected into a condensate below its dew point the gas continues to re-vaporize liquid at a rate governed only by the ambient pressure. The vapor saturates over a zone whose thickness is of the order of one grid block; in particular all the liquid in the vicinity of the injectors evaporates rapidly. Results obtained with fully compositional simulation models suggest that liquid saturation profiles would vary more slowly with increasing distance from the gas injectors. This compositional effect can thus be modeled via the CO2 Dynamic EOR Model, although there are no test cases at this stage.

Example

```
--
--       OIL PVT PROPERTIES FOR LIVE OIL VERSUS CO2 MASS FRACTION
--
PVTSOL
-- CO2     PSAT      BO         BG         RS         RV           XVOL     YVOL     OIL VISC    GAS VISC
-- MFRAC   PSIA      RB/STB     RB/MSCF    MSCF/STB   STB/MSCF                       CPOISE      CPOISE
-- -----   -------   ------   ----------   --------   ----------   ------   ------   ----------  ----------
   0.000     14.50   1.0000   1.7810e+02   0.0000     0.0000e+00   0.0000   0.0000   2.0340e-01  1.3016e-02
           725.18    1.0655   6.0855e+00   0.1302     1.4432e-04   0.0000   0.0000   2.0340e-01  1.3016e-02
           870.22    1.0806   5.0340e+00   0.1591     1.9594e-04   0.0000   0.0000   1.9973e-01  1.3222e-02
          1015.26    1.0961   4.2814e+00   0.1886     2.8389e-04   0.0000   0.0000   1.9609e-01  1.3452e-02
          1160.30    1.1121   3.7174e+00   0.2189     4.1914e-04   0.0000   0.0000   1.9246e-01  1.3708e-02
          1305.33    1.1285   3.2799e+00   0.2499     6.0779e-04   0.0000   0.0000   1.8886e-01  1.3994e-02
          1450.37    1.1454   2.9314e+00   0.2819     8.5105e-04   0.0000   0.0000   1.8527e-01  1.4309e-02
          1595.41    1.1629   2.6479e+00   0.3148     1.1490e-03   0.0000   0.0000   1.8171e-01  1.4657e-02
          1740.45    1.1809   2.4133e+00   0.3487     1.5030e-03   0.0000   0.0000   1.7819e-01  1.5037e-02
          1885.49    1.1996   2.2163e+00   0.3838     1.9155e-03   0.0000   0.0000   1.7469e-01  1.5450e-02
          2030.52    1.2189   2.0490e+00   0.4200     2.3899e-03   0.0000   0.0000   1.7122e-01  1.5897e-02
          2175.56    1.2389   1.9055e+00   0.4574     2.9302e-03   0.0000   0.0000   1.6779e-01  1.6377e-02
          2320.60    1.2597   1.7812e+00   0.4962     3.5412e-03   0.0000   0.0000   1.6440e-01  1.6890e-02
          2465.64    1.2812   1.6728e+00   0.5364     4.2273e-03   0.0000   0.0000   1.6104e-01  1.7435e-02
          2610.67    1.3037   1.5775e+00   0.5782     4.9942e-03   0.0000   0.0000   1.5772e-01  1.8011e-02
          2755.71    1.3270   1.4933e+00   0.6216     5.8471e-03   0.0000   0.0000   1.5443e-01  1.8617e-02
          2900.75    1.3514   1.4185e+00   0.6668     6.7918e-03   0.0000   0.0000   1.5117e-01  1.9253e-02
          3045.79    1.3768   1.3517e+00   0.7139     7.8347e-03   0.0000   0.0000   1.4795e-01  1.9917e-02
          3137.32    1.3937   1.3132e+00   0.7450     8.5549e-03   0.0000   0.0000   1.4592e-01  2.0353e-02
          4061.05    1.3705   1.3132e+00   0.7450     8.5549e-03   0.0000   0.0000   1.5939e-01  2.0353e-02 /
   0.010    14.50    1.0000   1.7810e+02   0.0000     0.0000e+00   0.0464   0.0381   2.0301e-01  1.3203e-02
           725.18    1.0670   6.0041e+00   0.1341     1.4845e-04   0.0464   0.0381   2.0301e-01  1.3203e-02
           ..... ..... ... ..... ..... ..... ..... ..... ..... ..... ..... ..... ..... ..... ... ..... ..... ..... .....
          3135.04    1.4082   1.2950e+00   0.7755     8.9113e-03   0.0298   0.0277   1.4422e-01  2.0743e-02
          4061.05    1.3840   1.2950e+00   0.7755     8.9113e-03   0.0298   0.0277   1.5772e-01  2.0743e-02 /
   0.100    14.50    1.0000   1.7810e+02   0.0000     0.0000e+00   0.3426   0.3023   1.9984e-01  1.4740e-02
           725.18    1.0795   5.3329e+00   0.1666     1.8243e-04   0.3426   0.3023   1.9984e-01  1.4740e-02
           ..... ..... ... ..... ..... ..... ..... ..... ..... ..... ..... ..... ..... ..... ... ..... ..... ..... .....
          3116.25    1.5281   1.1444e+00   1.0268     1.1849e-02   0.2492   0.2431   1.3023e-01  2.3961e-02
          4061.05    1.4955   1.1444e+00   1.0268     1.1849e-02   0.2492   0.2431   1.4400e-01  2.3961e-02 /
   0.200    14.50    1.0000   1.7810e+02   0.0000     0.0000e+00   0.5303   0.4898   1.9699e-01  1.5838e-02
           725.18    1.0909   4.8402e+00   0.1964     2.1919e-04   0.5303   0.4898   1.9699e-01  1.5838e-02
           ..... ..... ... ..... ..... ..... ..... ..... ..... ..... ..... ..... ..... ..... ... ..... ..... ..... .....
          4061.05    1.6501   1.0017e+00   1.3720     1.6728e-02   0.4220   0.4240   1.2963e-01  2.7757e-02 /
   0.300    14.50    1.0000   1.7810e+02   0.0000     0.0000e+00   0.6486   0.6159   1.9471e-01  1.6588e-02
           725.18    1.1003   4.4992e+00   0.2210     2.5440e-04   0.6486   0.6159   1.9471e-01  1.6588e-02
           ..... ..... ... ..... ..... ..... ..... ..... ..... ..... ..... ..... ..... ..... ... ..... ..... ..... .....
          4061.05    1.8529   8.8189e-01   1.8200     2.4541e-02   0.5491   0.5597   1.1636e-01  3.2195e-02 /
   0.400    14.50    1.0000   1.7810e+02   0.0000     0.0000e+00   0.7303   0.7059   1.9292e-01  1.7131e-02
           725.18    1.1079   4.2506e+00   0.2413     2.8832e-04   0.7303   0.7059   1.9292e-01  1.7131e-02
           ..... ..... ... ..... ..... ..... ..... ..... ..... ..... ..... ..... ..... ..... ... ..... ..... ..... .....
          4061.05    2.1298   7.8515e-01   2.4251     3.7664e-02   0.6468   0.6616   1.0441e-01  3.7755e-02 /
   0.500    14.50    1.0000   1.7810e+02   0.0000     0.0000e+00   0.7905   0.7733   1.9156e-01  1.7543e-02
           725.18    1.1140   4.0628e+00   0.2581     3.2217e-04   0.7905   0.7733   1.9156e-01  1.7543e-02
           ..... ..... ... ..... ..... ..... ..... ..... ..... ..... ..... ..... ..... ..... ... ..... ..... ..... .....
          4061.05    2.5282   7.1557e-01   3.2879     6.0741e-02   0.7245   0.7379   9.3885e-02  4.5038e-02 /
   0.600    14.50    1.0000   1.7810e+02   0.0000     0.0000e+00   0.8372   0.8259   1.9061e-01  1.7868e-02
           725.18    1.1188   3.9168e+00   0.2720     3.5813e-04   0.8372   0.8259   1.9061e-01  1.7868e-02
           ..... ..... ... ..... ..... ..... ..... ..... ..... ..... ..... ..... ..... ..... ... ..... ..... ..... .....
          4061.05    3.1455   6.8307e-01   4.6154     1.0176e-01   0.7880   0.7951   8.4761e-02  5.4583e-02 /
   0.700    14.50    1.0000   1.7810e+02   0.0000     0.0000e+00   0.8755   0.8686   1.9018e-01  1.8132e-02
           725.18    1.1223   3.8017e+00   0.2833     4.0084e-04   0.8755   0.8686   1.9018e-01  1.8132e-02
           ..... ..... ... ..... ..... ..... ..... ..... ..... ..... ..... ..... ..... ..... ... ..... ..... ..... .....
          4061.05    3.3568   6.6803e-01   5.4784     1.3282e-01   0.9106   0.9124   8.9321e-02  6.2163e-02 /
   0.800    14.50    1.0000   1.7810e+02   0.0000     0.0000e+00   0.9520   0.9591   1.8807e-01  1.8720e-02
           725.18    1.1335   3.5254e+00   0.3153     5.0228e-04   0.9520   0.9591   1.8807e-01  1.8720e-02
           ..... ..... ... ..... ..... ..... ..... ..... ..... ..... ..... ..... ..... ..... ... ..... ..... ..... .....
          4061.05    2.7406   5.5115e-01   3.5079     8.5588e-02   0.8828   0.8905   8.1404e-02  7.0425e-02 /
   0.900    14.50    1.0000   1.7810e+02   0.0000     0.0000e+00   0.9846   0.9855   1.9003e-01  1.8852e-02
           725.18    1.1293   3.5076e+00   0.3007     4.6299e-04   0.9846   0.9855   1.9003e-01  1.8852e-02
             ..... ..... ... ..... ..... ..... ..... ..... ..... ..... ..... ..... ..... ..... ... ..... ..... ..... .....
..... ..... ... ..... ..... ...
          4061.05    1.9854   5.0045e-01   2.0535     3.9219e-02   0.9400   0.9448   1.0088e-01  6.5831e-02 /
   1.000    14.50    1.0000   1.7810e+02   0.0000     0.0000e+00   1.0000   1.0000   1.7716e-01  1.8887e-02
           725.18    1.1418   3.5421e+00   0.2487     1.7810e-10   1.0000   1.0000   1.7716e-01  1.8887e-02
             ..... ..... ... ..... ..... ..... ..... ..... ..... ..... ..... ..... ..... ..... ... ..... ..... ..... .....
..... ..... ... ..... ..... ...
          4061.05    1.2145   4.3773e-01   0.6633     4.2745e-09   1.0000   1.0000   1.4288e-01  6.2211e-02 /
/
```

The above example defines one oil PVT Properties for Live Oil versus CO2 Mass Fraction table, and assumes that NTPVT equals one the TABDIMS keyword in the RUNSPEC section.

8.3.234 PVTW - DEFINE WATER FLUID PROPERTIES FOR VARIOUS REGIONS

Description

PVTW defines the water properties for various regions in the model. The number of PVTW vector data sets is defined by the NTPVT parameter on the TABDIMS keyword in the RUNSPEC section and the allocation of the PVTW tables to different grid blocks in the model is done via the PVTNUM keyword in the REGIONS section. One data set consists of one record or line which is terminated by a "/". If the water phase is active in the model, which is normally the case, then this keyword must be defined in the OPM Flow input deck.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	PRES	PRES is a real number defining the water reference pressure (P) for the other parameters for this data set.			
		psia	barsa	atma	None
2	WFVF	WFVF is a real number defining the water formation volume factor (Bw) at the water reference pressure.			
		rb/stb 1.0	rm^3/sm^3 1.0	rcc/scc 1.0	Defined
3	WCOMP	WCOMP is a real number defining the water compressibility (Cw) at the water reference pressure and is defined as: $$C_w = -\frac{1}{B_w}\left(\frac{dB_w}{dP}\right)$$			
		1/psia 0.00004	1/barsa 0.00004	1/atma 0.00004	Defined
4	WVISC	WVISC is a real number defining the water viscosity (μw) at the water reference pressure.			
		cP 0.50	cP 0.50	cP 0.50	Defined
5	WVISCOMP	WVISCOMP is a real number defining the water viscosibility (μwc) at the water reference pressure, μwc(Pref) and is defined as: $$\mu_{wc} = -\frac{1}{\mu_w}\left(\frac{d\mu_w}{dP}\right)$$			
		1/psia 0.0	1/barsa 0.0	1/atma 0.0	Defined

Notes:

1) The keyword is followed by NTPVT tables as declared on the TABDIMS keyword in the RUNSPEC section.
2) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.122: PVTW Keyword Description

Note that provided the first table has been entered, subsequent tables may be defaulted, in this case the prior table is copied to the current table. See the third example for an illustration on how to use this feature.

Examples

The following shows the PVTW keyword for when NTPVT on the TABDIMS keyword in the RUNSPEC section is set to one.

```
--
--        WATER PVT TABLE
--
PVTW
--        REF PRES  BW         CW        VISC     VISC
--        PSIA      RB/STB     1/PSIA    CPOISE   GRAD
--        --------  --------   -------   ------   ------
          4840.0    1.019      2.7E-6    0.370    1*          / TABLE NO. 01
```

The next example shows the PVTW keyword for when NTPVT on the TABDIMS keyword in the RUNSPEC section is set to three.

```
--
--        WATER PVT TABLE
--
PVTW
--        REF PRES  BW         CW        VISC     VISC
--        PSIA      RB/STB     1/PSIA    CPOISE   GRAD
--        --------  --------   -------   ------   ------
          4640.0    1.008      2.5E-6    0.350    1*          / TABLE NO. 01
          4840.0    1.019      2.7E-6    0.370    1*          / TABLE NO. 02
          4940.0    1.030      2.8E-6    0.390    1*          / TABLE NO. 03
```

The above example defines three water PVT tables and assumes that NTPVT equals three on the TABDIMS keyword in the RUNSPEC section.

The third, and final example, shows the PVTW keyword for when NTPVT on the TABDIMS keyword in the RUNSPEC section is set to four. Here table two defaults to table one, and table four defaults to table three

```
--
--        WATER PVT TABLE
--
PVTW
--        REF PRES  BW         CW        VISC     VISC
--        PSIA      RB/STB     1/PSIA    CPOISE   GRAD
--        --------  --------   -------   ------   ------
          4640.0    1.008      2.5E-6    0.350    1*          / TABLE NO. 01
                                                              / TABLE NO. 02
          4940.0    1.030      2.8E-6    0.390    1*          / TABLE NO. 03
                                                              / TABLE NO. 04
```

Note that there is no terminating "/" for this keyword.

8.3.235 PVTWSALT - DEFINE BRINE WATER FLUID PROPERTIES FOR VARIOUS REGIONS

Description

PVTWSALT defines the brine water properties for various regions in the model, for when the brine phase has been activated by the BRINE keyword in the RUNSPEC section. In this case PVTWSALT is used instead of PVTW in the input file. However, if the ECLMC keyword has been entered in the RUNSPEC section to invoke the Multi-Component Brine model, the PVTW keyword should be used instead of PVTWSALT, as with this combination the salinity effect on the density is ignored.

The number of PVTWSALT table data sets is defined by the NTPVT parameter on the TABDIMS keyword in the RUNSPEC section and the allocation of the PVTWSALT tables to different grid blocks in the model is done via the PVTNUM keyword in the REGION section.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1-1	PRESS	Single real positive value that defines the reference pressure for the data in the following records (Pref). PRESS should be approximately equal to the average reservoir pressures in the model. The simulator uses the previous time step values to forecast the current time step water properties by linear interpolation. If PRESS is not representative of the average reservoir pressures in the model then the linear interpolation might result in nonphysical values of the water saturation and water viscosity.			
		psia	barsa	atma	None
1-2	SALTSURF	A real value that defines the reference salt concentration in the solution in the surface stock tank water (Cs, ref) If defaulted SALTSURF is taken as the minimum salt concentration entered in the SALTCON columnar vector in the second record for this keyword. This should be in most cases be zero.			
		lb/stb	kg/sm^3	gm/scc	Defined
1-3	/	Record terminated by a "/"			Not Applicable
2-1	SALTCON	A columnar vector of real monotonically increasing values that defines the salt concentration in the solution (Cs).			
		lb/stb	kg/sm^3	gm/scc	None
2-2	WFVF	WFVF is a real columnar vector defining the water formation volume factor (Bw) at the reference pressure PRESS, for the corresponding salt concentration SALTCON.			
		rb/stb	rm^3/sm^3	rcc/scc	None

No.	Name	Description			Default
		Field	Metric	Laboratory	
2-3	WCOMP	WCOMP is a real columnar vector defining the water compressibility (Cw) at the water reference pressure PRESS, for the corresponding salt concentration SALTCON. The water compressibility is defined as: $$C_w = -\frac{1}{B_w}\left(\frac{dB_w}{dP}\right)$$			
		1/psia	1/bars	1/atma	None
2.4	WVISC	WVISC is a real columnar vector defining the water viscosity (μw) at the water reference pressure PRESS, for the corresponding salt concentration SALTCON.			
		cP	cP	cP	None
2.5	WVISCOMP	WVISCOMP is a real columnar vector defining the water viscosibility (μwc) at the water reference pressure PRESS, for the corresponding salt concentration SALTCON. The water viscosibility is defined as: $$\mu_{wc} = -\frac{1}{\mu_w}\left(\frac{d\mu_w}{dP}\right)$$			
		1/psia	1/barsa	1/atma	None
2-6	/	Table and record terminated by a "/"			Not Applicable

Notes:

1) The keyword is followed by NTPVT data sets as declared on the TABDIMS keyword in the RUNSPEC section and each data set consists of two records, with items 1-1 to 1-3 representing record one items and 2-1 to 2-6 representing record number two items, etc., in the "No." column in this table.
2) Each table must contain a minimum of two rows and a maximum of NPPVT rows as declared on the TABDIMS keyword in the RUNSPEC section.
3) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.123: PVTWSALT Keyword Description

As mentioned above, the simulator first calculates the water properties as functions of the salt concentration at the previous time step by linear interpolation in salt concentration for water compressibility (Cw), water viscosibility (μwc), $\frac{1}{B_w}$ and $\frac{1}{B_w\ \mu_w}$. It then calculates the values of B_w and $B_w\ \mu_w$ at the current time step using the current pressure P, using the following equations:

$$B_w(P,C_s) = \frac{B_w(P_{ref},C_{s,ref})}{1 + C_w(P-P_{ref}) + \frac{(C_w(P-P_{ref}))^2}{2}} \quad (8.86)$$

and

$$B_w(P,C_s)\ \mu_w(P,C_s) = \frac{B_w(P_{ref},C_{s,ref})\ \mu_w(P_{ref},C_{s,ref})}{1 + (C_w - \mu_{wc})(P-P_{ref}) + \frac{((C_w - \mu_{wc})(P-P_{ref}))^2}{2}} \quad (8.87)$$

See also the BDENSITY keyword in the PROPS section that defines the brine surface densities for the salt concentrations declared on the PVTWSALT keyword. Note that if the BDENSITY keyword is absent from the input file then the brine surface densities will be set to the water density values declared via the DENSITY keyword in the PROPS section. In this case there is no variation in brine surface density with respect to salt concentration.

Example

The following shows the PVTWSALT keyword for when NTPVT on the TABDIMS keyword in the RUNSPEC section is set equal to two and NPPVT is set to greater than four on the TABDIMS keyword.

```
--
--        WATER SALT PVT TABLE
--
PVTWSALT
--        REF PRES  REF SALT
--        PSIA      LB/STB
--        --------  --------
          4500.0    0.000                                  / TABLE NO. REF. DATA
--
--        SALTCON BW          CW         VISC     VISC
--        LB/STB    RB/STB      1/PSIA     CPOISE   GRAD
--        --------  --------    -------    ------   ------
             0.0    1.020       2.7E-6     0.370    0.0
             2.0    1.010       2.7E-6     0.370    0.0
             4.0    1.000       2.7E-6     0.370    0.0
            10.0    0.950       2.7E-6     0.370    0.0      / TABLE NO. 01 SALT DATA
--
--        REF PRES  REF SALT
--        PSIA      LB/STB
--        --------  --------
          4000.0    0.000                                  / TABLE NO. 02 REF. DATA
--
--        SALTCON  BW          CW         VISC     VISC
--        LB/STB    RB/STB      1/PSIA     CPOISE   GRAD
--        --------  --------    -------    ------   ------
             0.0    1.005       2.5E-6     0.320    0.0
             3.0    1.000       2.5E-6     0.320    0.0
             6.0    0.985       2.5E-6     0.320    0.0
            12.0    0.930       2.5E-6     0.320    0.0      / TABLE NO. 02 SALT DATA
```

Note that each table is terminated by a "/" and there is no "/" terminator for the keyword.

8.3.236 PYEND – END THE DEFINITION OF A PYINPUT SECTION

The PYINPUT and PYEND keywords are a part of OPM Flow's Python scripting facility that processes standard Python commands that can be used to manipulate and define the simulators input parameters during processing of the input deck. The main purpose of the facility is to script the construction of the various keywords. A PYINPUT Definition Section is terminated by a PYEND keyword on a separate single line.

See *PYEND – End the Definition of a PYINPUT Section* in the GRID section for a full description.

8.3.237 PYINPUT – DEFINE THE START OF A PYINPUT SECTION

The PYINPUT and PYEND keywords are a part of OPM Flow's Python scripting facility that processes standard Python commands that can be used to manipulate and define the simulators input parameters during processing of the input deck. The main purpose of the facility is to script the construction of the various keywords used by the simulator. PYINPUT declares the start of a PYINPUT Definition Section on a single separate line, which is then followed by various standard Python commands, with one command per line.

See *PYINPUT – Define the Start of a PYINPUT Section* in the GRID section for a full description.

8.3.238 PVZG - GAS PVT PROPERTIES FOR DRY GAS (Z-FACTOR)

Description

PVZG defines the gas PVT properties for dry gas[228] via the gas compressibility factor (z-factor), instead of the gas formation volume factor. If the gas has a constant and uniform vaporized oil concentration, Condensate-Gas Ratio ("CGR"), and if the reservoir pressure never drops below the saturation pressure (dew point pressure), then the model can be run more efficiently by omitting the OIL and VAPOIL keywords from the RUNSPEC section, treating the gas as a dry gas, and defining a constant Rv (CGR) value with keyword RVCONST or RVCONSTT in the PROPS section. This results in the model being run with as a dry gas problem with no active oil (condensate) phase. However, OPM Flow takes into account the constant Rv in the calculations and reporting.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

[228] *Natural gas that occurs in the absence of condensate or liquid hydrocarbons, or gas that had condensable hydrocarbons removed, is called dry gas. It is primarily methane with some intermediates. The hydrocarbon mixture is solely gas in the reservoir and there is no liquid (condensate surface liquid) formed either in the reservoir or at surface. The term dry indicates that the gas does not contain heavier hydrocarbons to form liquids at the surface conditions. Dry gas typically has GOR's greater than 100,000 scf/stb or 18,000 Sm3/m3.*

No.	Name	Description			Default
		Field	Metric	Laboratory	
1-1	RTEMP	Single real positive value that defines the reservoir temperature for the data in the following records.			
		°F	°C	°C	
1-2	/	Record terminated by a "/"			Not Applicable
2-1	PRESS	A columnar vector of real monotonically increasing down the column values that defines the gas phase pressure.			
		psia	barsa	atma	None
2-2	GZFAC	A columnar vector of real values that defines the corresponding gas phase z-factor at the given pressure, PRESS.			
		dimensionless	dimensionless	dimensionless	None
2-3	GVISC	A columnar vector of real increasing down the column values that defines the corresponding gas phase viscosity.			
		cP	cP	cP	None

Notes:

1) The keyword is followed by NTPVT data sets as declared on the TABDIMS keyword in the RUNSPEC section and each data set consists of two records, with items 1-1 to 1-2 representing record one items and 2-1 to 2-3 representing record number two items, etc., in the "No." column in this table.
2) Each table must contain a minimum of two rows and a maximum of NPPVT rows as declared on the TABDIMS keyword in the RUNSPEC section.
3) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.124: PVZG Keyword Description

See also the RVCONST and RVCONSTT keywords to define the constant Rv for dry gas.

The ideal gas law provides a relationship between the pressure, the temperature and the specific volume of an ideal gas (pure component). This relationship is modified by use of a compressibility factor, Z[229], to account for deviations, from ideal, to the behavior of real gases. The PVT relation for a real gas can be defined by:

$$PV = ZnRT \tag{8.88}$$

As the gas formation volume factor is used to relate the volume of gas, as measured at reservoir conditions, to the volume of gas as measured at standard conditions (60 °F and 14.7 psia, or 15 °C and 101.325 kPa). This gas property is then defined as the actual volume occupied by a certain amount of gas at a specified pressure and temperature, divided by the same amount of gas at standard conditions. Thus, using the above equation one can obtain the gas volumes at reservoir and standard conditions, i.e.

$$V_{sc} = \frac{Z_{sc} nRT_{sc}}{P_{sc}} \tag{8.89}$$

[229] *Standing, M. B.:" Volumetric and Phase Behaviour of Oil Field Hydrocarbon Systems", Renihold Publishing Corp., New York City (1952).*

$$V_i = \frac{Z_i n R T_i}{P_i} \tag{8.90}$$

Thus the gas formation volume factor can be expressed as:

$$E = \frac{V_{sc}}{V_i} \tag{8.91}$$

And substituting equation (8.89) and (8.90) into (8.91) we obtain

$$E = \left(\frac{P_i}{P_{sc}}\right)\left(\frac{T_{sc}}{T_i}\right)\left(\frac{1}{Z_i}\right) \tag{8.92}$$

Incorporating standard pressure and temperature values gives in SI units:

$$E = \left(\frac{P_i}{101.325}\right)\left(\frac{273.15 + 15}{T_i}\right)\left(\frac{1}{Z_i}\right) = 2.84\left(\frac{P_i}{Z_i T_i}\right) \tag{8.93}$$

or in field units:

$$E = \left(\frac{P_i}{14.7}\right)\left(\frac{460 + 60}{T_i}\right)\left(\frac{1}{Z_i}\right) = 35.37\left(\frac{P_i}{Z_i T_i}\right) \tag{8.94}$$

Where,

E	= gas formation volume factor (scf /rcft or Sm^3/m^3)
P	= pressure (psia or kPa)
Psc	= pressure standard conditions (psia or kPa)
Pi	= initial reservoir pressure (psia or kPa)
V	= volume (ft^3 or m^3)
T	= absolute temperature (°R or K)
Tsc	= temperature at standard conditions (°R or K)
Ti	= initial reservoir temperature (°R or K)
R	= gas constant (10.73 or 8.314)

RTEMP on this keyword is the Ti in the above equations and P_i is PRESS columnar vector.

Example

```
---
--       GAS PVT TABLE USING GAS Z-FACTOR
--
PVZG
--       RESERVOIR TEMPERATURE FOR Z TO BG CONVERSION
--
         180.0                                                  /
--
--        PRES       ZG          VISC
--        PSIA     DIMLESS      CPOISE
--       ------    ---------    -------
           14.7    0.998970     0.0130
          250.0    0.976260     0.0131
          500.0    0.954790     0.0134
          750.0    0.932050     0.0137
         1000.0    0.912990     0.0142
         1250.0    0.896320     0.0147
         1500.0    0.881610     0.0152
         1750.0    0.870830     0.0159
         2000.0    0.863130     0.0166
         2250.0    0.858920     0.0173
         2500.0    0.857800     0.0181
         2750.0    0.860430     0.0189
         3000.0    0.866440     0.0197
         3250.0    0.874980     0.0206
         3500.0    0.885470     0.0214
         3750.0    0.898350     0.0223
         4000.0    1.025120     0.0277                    / TABLE NO 01
--
--       GAS PVT TABLE USING GAS Z-FACTOR
--
PVZG
--       RESERVOIR TEMPERATURE FOR Z TO BG CONVERSION
--
         180.0                                                  /
--
--        PRES       ZG          VISC
--        PSIA     DIMLESS      CPOISE
--       ------    ---------    -------
           14.7    0.998970     0.0130
          250.0    0.976260     0.0131
          500.0    0.954790     0.0134
          750.0    0.932050     0.0137
         1000.0    0.912990     0.0142
         1250.0    0.896320     0.0147
         1500.0    0.881610     0.0152
         1750.0    0.870830     0.0159
         2000.0    0.863130     0.0166
         2250.0    0.858920     0.0173
         2500.0    0.857800     0.0181
         2750.0    0.860430     0.0189
         3000.0    0.866440     0.0197
         3250.0    0.874980     0.0206
         3500.0    0.885470     0.0214
         3750.0    0.898350     0.0223
         4000.0    1.025120     0.0277                    / TABLE NO 01
```

The above example defines two dry PVZG tables assuming NTPVT equals two and NPPVT is greater than or equal to 17 on the TABDIMS keyword in the RUNSPEC section. There is no terminating "/" for this keyword.

8.3.239 QHRATING – DEFINE RIVER MASS FLOW VERSUS DEPTH TABLES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The QHRATING keyword defines a river's mass flow rate versus depth tables, for when the River option has been activated via the RIVRDIMS keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.240 REFINE – START THE DEFINITION OF A LOCAL GRID REFINEMENT

The REFINE keyword defines the start of a Cartesian or radial local grid refinement ("LGR") definition that sets the properties of the selected LGR. The keyword is then followed by the property keywords associated with the section where the keyword is being invoked. For example, if the REFINE keyword is used in the GRID section then most of the keywords in that section can be used to set the grid properties for the LGR.

See *REFINE – Start the Definition of a Local Grid Refinement* in the GRID section for a full description.

8.3.241 RIVRXSEC – DEFINE RIVER CROSS-SECTION VERSUS DEPTH PARAMETERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The RIVRXSEC keyword defines a river's cross-sectional area and perimeter versus depth parameters. The River option must be activated via the RIVRDIMS keyword in the RUNSPEC section in order to use this keyword.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.242 RKTRMDIR - ACTIVATE ROCKTAB KEYWORD DIRECTIONAL TRANSMISSIBILITY MULTIPLIERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword activates the directional transmissibility multipliers for the ROCKTAB keyword. This results in two additional columns being inputted on the ROCKTAB keyword. This feature is currently not supported in OPM Flow.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.243 ROCK - DEFINE THE ROCK COMPRESSIBILITY FOR VARIOUS REGIONS

Description

ROCK defines the rock compressibility for various regions in the model. The number of ROCK vector data sets is defined by the NTPVT parameter on the TABDIMS keyword in the RUNSPEC section and the allocation of the ROCK tables to different grid blocks in the model is done via the PVTNUM keyword in the REGIONS section. One data set consists of one record or line which is terminated by a "/".

This keyword must be defined in the OPM Flow input deck.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	PRESS	PRESS is a real number defining the rock reference pressure for the other parameters for this data set.			
		psia 14.7	barsa 1.0132	atma 1.0	Default
2	RCOMP	RCOMP is a real number defining the rock compressibility (cf) at the rock reference pressure and is defined as: $c_f = -\frac{1}{V}\left(\frac{dV}{dP}\right)$			
		1/psia 0.0	1/barsa 0.0	1/atma 0.0	Defined

Notes:

1) The keyword is followed by NTPVT vectors as declared by the NTPVT parameter on the TABDIMS keyword in the RUNSPEC section.

   Note, however, that if ROCKOPTS(ROCKOPT3) parameter has been used to set the allocation of the ROCK data via the ROCKNUM array, then the number of ROCK vectors should correspond to the value entered on TABDIMS(NTROCC) in the RUNSPEC section. Similarly, if the ROCKOPTS(ROCKOPT3) has been used to set the assignment of the ROCK data via the SATNUM array, then the number of vectors should correspond to the value entered via the TABDIMS(NTSFUN) parameter, since the tables will be allocated via the SATNUM array.

2) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.125: ROCK Keyword Description

The simulator adjusts the pore volume based on the reference pressure (PRESS), that is:

$$V(P_i) = V(P_r)\left(1 + c_f(P_i - P_r) + \frac{\left(c_f(P_i - P_r)\right)^2}{2}\right) \quad (8.95)$$

where:

c_f = rock compressibility (RCOMP) at the reference pressure (PRESS),
P_i = initial grid cell pressure,
P_r = reference pressure (PRESS),
$V(P_i)$ = pore volume at initial conditions, and
$V(P_r)$ = pore volume at at the reference pressure.

Note

If the Rock Compaction option has been activated via the ROCKCOMP keyword in the RUNSPEC section, then the ROCKTAB keyword in the PROPS section should be used instead of ROCK keyword.

See also the ROCKOPTS and ROCKTAB keywords in the PROPS section.

Examples

The following shows the ROCK keyword for when NTPVT on the TABDIMS keyword in the RUNSPEC section is set to one.

```
--
--       ROCK COMPRESSIBILITY
--
--       REFERENCE PRESSURE IS TAKEN FROM THE HCPV WEIGHTED RESERVOIR PRESSURE
--       AS THE PORV IS ALREADY AT RESERVOIR CONDITIONS (OPM FLOW USES THE
--       REFERENCE PRESSURE) TO CONVERT THE GIVEN PORV TO RESERVOIR CONDITIONS
--       USING THE DATA ON THE ROCK KEYWORD)
--
--       REF PRES  CF
--       PSIA      1/PSIA
--       --------  --------
ROCK
         3966.9    5.0E-06                                  / ROCK COMPRESSIBILITY
```

The next example shows the ROCK keyword for when NTPVT on the TABDIMS keyword in the RUNSPEC section is set to three.

```
--
--       ROCK COMPRESSIBILITY
--
--       REFERENCE PRESSURE IS TAKEN FROM THE HCPV WEIGHTED RESERVOIR PRESSURE
--       AS THE PORV IS ALREADY AT RESERVOIR CONDITIONS (OPM FLOW USES THE
--       REFERENCE PRESSURE) TO CONVERT THE GIVEN PORV TO RESERVOIR CONDITIONS
--       USING THE DATA ON THE ROCK KEYWORD)
--
--       REF PRES  CF
--       PSIA      1/PSIA
--       --------  --------
ROCK
         3566.9    5.0E-06                       / ROCK COMPRESSIBILITY REGION 1
         3966.9    5.5E-06                       / ROCK COMPRESSIBILITY REGION 2
         4566.9    6.0E-06                       / ROCK COMPRESSIBILITY REGION 3
```

The above example defines three ROCK tables and assumes that NTPVT equals three on the TABDIMS keyword in the RUNSPEC section.

There is no terminating "/" for this keyword, and thus the number entries must match the value entered via the TABDIMS(NTPVT), TABDIMS(NTROCC), or TABDIMS(NTSFUN) parameters, depending on the option selected via the ROCKOPTS keyword.

8.3.244 ROCK2D – PORE VOLUME COMPACTION VERSUS PRESSURE AND SW TABLES

Description

The ROCK2D keyword defines rock compressibility pore volume multipliers as a function of pressure and water saturation ("Sw") for when the rock compaction option has been invoked by the ROCKCOMP keyword in the RUNSPEC section. The pressure values are defined on this keyword and the water saturations are declared on the associated ROCKWNOD keyword in the PROPS section

The rock compaction pore volume and transmissibility multipliers, entered via the ROCKTAB, ROCK2D and ROCK2DTR keywords, are applied to the pore pressure, unless the OVERBURD keyword in PROPS section is included in the input deck. When the OVERBURD keyword is present the multipliers are applied to the effective pore volume pressure, that is $P_{(effective)} = P_{(Pressure)} - P_{(overburden)}$. If the keyword is not present in the input deck then the overburden pressure is set to zero.

This keyword should only be used if compaction option has been enabled.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	PRESS	A columnar vector of real monotonically increasing down the column values that defines the corresponding overburden pressure for the subsequent MULT columnar vector.			
		psia	bars	atm	None
2	MULT	A columnar vector of real equal or decreasing down the column values that are less than or equal to one, that defines the rock compressibility pore volume multipliers corresponding to PRESS and for each water saturation entry in the ROCKWNOD keyword.			
		dimensionless	dimensionless	dimensionless	None

Notes:

1) The keyword is followed by NTROCC tables as declared on the ROCKCOMP keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NSSFUN rows as declared on the TABDIMS keyword in the RUNSPEC section.
3) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.126: ROCK2D Keyword Description

See also the OVERBURD, ROCKTAB, ROCK2DTR, and ROCKWNOD keywords in the PROPS section.

Example

The following example defines two pore volume compaction tables, assuming NTROCC is equal to two on the ROCKCOMP keyword and NSSFUN is greater than or equal to four on the TABDIMS keyword.

```
--
--       ROCK COMPACTION VERSUS PRESSURE AND SW TABLES
--
ROCK2D
--       PRESS    PORV          FIRST ROCK2D TABLE DATA
--       PSIA     MULTIPLER
--       ------   ----------
            0.0      0.850
                     0.850
                     0.850
                     0.085                              / P-SW SET TABLE NO. 01
--       PRESS    PORV
--       PSIA     MULTIPLER
--       ------   ----------
         1000.0      0.900
                     0.900
                     0.900
                     0.900                              / P-SW SET TABLE NO. 01
--       PRESS    PORV
--       PSIA     MULTIPLER
--       ------   ----------
         2500.0      0.950
                     0.950
                     0.950
                     0.950                              / P-SW SET TABLE NO. 01
--       PRESS    PORV
--       PSIA     MULTIPLER
--       ------   ----------
         5000.0      1.000
                     1.000
                     1.000
                     1.000                              / P-SW SET TABLE NO. 01
--
--       PRESS    PORV          SECOND ROCK2D TABLE DATA
--       PSIA     MULTIPLER
--       ------   ----------
            0.0      0.800
                     0.800
                     0.800
                     0.800                              / P-SW SET TABLE NO. 02
--       PRESS    PORV
--       PSIA     MULTIPLER
--       ------   ----------
         1000.0      0.880
                     0.880
                     0.880
                     0.880                              / P-SW SET TABLE NO. 02
--       PRESS    PORV
--       PSIA     MULTIPLER
--       ------   ----------
         2500.0      0.950
                     0.950
                     0.950
                     0.950                              / P-SW SET TABLE NO. 02
--       PRESS    PORV
--       PSIA     MULTIPLER
--       ------   ----------
         5000.0      1.000
                     1.000
                     1.000
                     1.000                              / P-SW SET TABLE NO. 02
```

Note that there must be exactly NTROCC tables entered for this keyword, otherwise an error will occur.

8.3.245 ROCK2DTR – TRANSMISSIBILITY COMPACTION VERSUS PRESSURE AND SW TABLES

Description

The ROCK2DTR keyword defines rock compressibility transmissibility multipliers as a function of pressure and water saturation ("Sw") for when the rock compaction option has been invoked by the ROCKCOMP keyword in the RUNSPEC section. The pressure values are defined on this keyword and the water saturations are declared on the associated ROCKWNOD keyword in the PROPS section

The rock compaction pore volume and transmissibility multipliers, entered via the ROCKTAB, ROCK2D and ROCK2DTR keywords, are applied to the pore pressure, unless the OVERBURD keyword in PROPS section is included in the input deck. When the OVERBURD keyword is present the multipliers are applied to the effective pore volume pressure, that is $P_{(effective)} = P_{(Pressure)} - P_{(overburden)}$. If the keyword is not present in the input deck then the overburden pressure is set to zero.

This keyword should only be used if compaction option has been enabled.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	PRESS	A columnar vector of real monotonically increasing down the column values that defines the corresponding overburden pressure for the subsequent MULT columnar vector.			
		psia	bars	atm	None
2	MULT	A columnar vector of real equal or decreasing down the column values that are less than or equal to one, that defines the rock compressibility transmissibility multipliers corresponding to PRESS and for each water saturation entry in the ROCKWNOD keyword.			
		dimensionless	dimensionless	dimensionless	None

Notes:

1) The keyword is followed by NTROCC tables as declared on the ROCKCOMP keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NSSFUN rows as declared on the TABDIMS keyword in the RUNSPEC section.
3) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.127: ROCK2DTR Keyword Description

See also the OVERBURD, ROCKTAB, ROCK2D, and ROCKWNOD keywords in the PROPS section.

Example

The following example defines two rock compressibility transmissibility compaction tables, assuming NTROCC is equal to two on the ROCKCOMP keyword and NSSFUN is greater than or equal to four on the TABDIMS keyword.

```
--
--       TRANSMISSIBILITY COMPACTION VERSUS PRESSURE AND SW TABLES
--
ROCK2DTR
--       PRESS    TRAN          FIRST ROCK2DTR TABLE DATA
--       PSIA     MULTIPLER
--       ------   ----------
            0.0      0.850
                     0.850
                     0.850
                     0.085                                  / P-SW SET TABLE NO. 01
--       PRESS    TRAN
--       PSIA     MULTIPLER
--       ------   ----------
         1000.0      0.900
                     0.900
                     0.900
                     0.900                                  / P-SW SET TABLE NO. 01
--       PRESS    TRAN
--       PSIA     MULTIPLER
--       ------   ----------
         2500.0      0.950
                     0.950
                     0.950
                     0.950                                  / P-SW SET TABLE NO. 01
--       PRESS    TRAN
--       PSIA     MULTIPLER
--       ------   ----------
         5000.0      1.000
                     1.000
                     1.000
                     1.000                                  / P-SW SET TABLE NO. 01
--
--       PRESS    TRAN          SECOND ROCK2DTR TABLE DATA
--       PSIA     MULTIPLER
--       ------   ----------
            0.0      0.800
                     0.800
                     0.800
                     0.800                                  / P-SW SET TABLE NO. 02
--       PRESS    TRAN
--       PSIA     MULTIPLER
--       ------   ----------
         1000.0      0.880
                     0.880
                     0.880
                     0.880                                  / P-SW SET TABLE NO. 02
--       PRESS    TRAN
--       PSIA     MULTIPLER
--       ------   ----------
         2500.0      0.950
                     0.950
                     0.950
                     0.950                                  / P-SW SET TABLE NO. 02
--       PRESS    TRAN
--       PSIA     MULTIPLER
--       ------   ----------
         5000.0      1.000
                     1.000
                     1.000
                     1.000                                  / P-SW SET TABLE NO. 02
```

Note that there must be exactly NTROCC tables entered for this keyword, otherwise an error will occur.

8.3.246 ROCKOPTS – DEFINE ROCK COMPACTION AND COMPRESSIBILITY OPTIONS

Description

The ROCKOPTS keyword defines various option with respect to rock compaction and rock compressibility.

No.	Name	Description	Default
1	ROCKOPT1	ROCKOPT1 is a character string that defines the treatment of how the overburden pressures supplied by the OVERBURD keyword are applied to the tabulated pressures in the ROCKTAB keywords: 1) STRESS: Use this option if the overburden pressures on the OVERBURD keyword are greater than the fluid pressure which results in the effective fluid pressure being negative. To avoid the rock compaction tables being entered with negative pressure values use this option. In this case the pore volume and transmissibility multipliers will be tabulated against the effective overburden pressure. 2) PRESSURE: In this case the pore volume and transmissibility multipliers should be effective pressure. This the default value. ROCKOPT1 should be set to PRESSURE if the OVERBURD is not used in the input deck. Only the default value of PRESSURE is supported.	PRESSURE
2	ROCKOPT2	ROCKOPT2 is a character string that sets the reference pressure option: 1) STORE: Copies the initial calculated grid block pressures into the overburden pressure array, resulting in the pore volumes being referenced at the initial pressures instead of the reference pressures as per the ROCKTAB keyword. 2) NOSTORE: This option results in the pore volumes being referenced as per the ROCKTAB keyword. This is the default value. Note that STORE option should not be used with the OVERBURD keywords as the OVERBURD data will be overwritten. Only the default value of NOSTORE is supported.	NOSTORE
3	ROCKOPT3	ROCKOPT3 is a character string that defines which region array should be used to allocate the various ROCK and ROCKTAB property tables in the model: 1) ROCKOPT3, should be set to ROCKNUM, SATNUM or PVTNUM. 2) If the parameter is defaulted or the ROCKOPT keyword is not present in the deck, then the PVTNUM array is used. 3) Secondly, if ROCKOPT3 is set to ROCKNUM but the NTROCC parameter on the TABDIMS keyword in the RUNSPEC section is defaulted, then again the PVTNUM array will be utilized. Only the PVTNUM and ROCKNUM options are currently supported.	PVTNUM

No.	Name	Description	Default
4	ROCKOPT4	ROCKOPT4 is a character string that sets the initial conditions for the HYSTER and BOBERG options: 1) DEFLATION: This option defines the reservoir rock to be fully compacted and the deflation curve is used to calculated the initial pore volume and transmissibility multipliers. This is the default value. 2) ELASTIC: This option sets the pore volume and transmissibility multipliers to one, as the reservoir rock is set to lie on the elastic curve. This parameter is ignored by OPM Flow as the ROCKCOMP(ROCKOPT) options of HYSTER and BOBERG are not supported by the simulator.	DEFLATION
Notes: 1) The keyword is terminated by a "/".			

Table 8.128: ROCKOPTS Keyword Description

Example

```
--
--        ROCKOPT1  ROCKOPT2   ROCKOPT3  ROCKOPT3
--        PRS/STRE  NO/STORE   ARRAY
--        --------  --------   --------  --------
ROCKOPTS
          PRESSURE  NOSTORE    PVTNUM                        / ROCK COMP OPTIONS
```

The above example defines the default values for the ROCKOPTS keyword.

8.3.247 ROCKPAMA – DEFINE COAL PALMER-MANSORRI ROCK MODEL PARAMETERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

ROCKPAMA defines the Palmer-Mansoori[230] and [231] parameters used for this rock model, for when the Coal Bed Methane option has been activated via the COAL keyword, and PALM-MAN has been declared for the ROCKOPT variable on the ROCKCOMP keyword; both keywords are in the RUNSPEC section. The Palmer-Mansoori rock model is used to calculate the impact on pore volume and permeability due to rock compaction.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

[230] *Palmer, I. and Mansoori, J. "How Permeability Depends on Stress and Pore Pressure in Coalbeds: A New Model," paper SPE 52607, SPE Reservoir Evaluation & Engineering (1998) 1, No. 6, 539-544.*

[231] *Clarkson, C.R., Pan, Z., Palmer, I. and Harpalani, S. "Predicting Sorption-Induced Strain and Permeability Increase With Depletion for Coalbed-Methane Reservoirs", SPE 114778-PA, SPE Journal (2010) 15, No. 1, 152–159.*

8.3.248 ROCKTAB – ROCK COMPACTION TABLES

Description

The ROCKTAB keyword defines the rock compaction attributes to be applied for when the rock compaction option has been invoked by the ROCKCOMP keyword in the RUNSPEC section. ROCKTAB defines pore volume and transmissibility multipliers versus pressure that are used in the compaction calculations. If the RKTRMDIR has been activated in the PROPS section, then the transmissibility multiplier is directional dependent and two additional columns are used to define the y and z direction transmissibility multipliers.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	PRESS	If the ROCKOPT1 variable has been set to PRESSURE on the ROCKOPTS keyword in the PROPS section, then PRESS should be a columnar vector of real monotonically increasing down the column values, that define the reference pressure for which the other parameters correspond to. If ROCKOPT1 has been set to STRESS, then PRESS should be a columnar vector of real monotonically decreasing down the column values.			
		psia	bars	atm	None
2	PORV	A columnar vector of real positive values that are either equal or increasing down the column that define the rock pore volume multiplier for a given PRESS.			
		dimensionless	dimensionless	dimensionless	None
3	TRANS	If the RKTRMDIR is absent from the input deck, then TRANS is a columnar vector of real positive values that are either equal or increasing down the column that define the x, y, and z directional transmissibility multipliers for the corresponding PRESS. If the RKTRMDIR is present in the input deck, then TRANS is a columnar vector of real positive values that are either equal or increasing down the column that define only the x directional transmissibility multipliers for the corresponding PRESS.			
		dimensionless	dimensionless	dimensionless	None
4	TRANSY	If the RKTRMDIR is absent from the input deck, then TRANSY is ignored. If the RKTRMDIR is present in the input deck, then TRANSY is a columnar vector of real positive values that are either equal or increasing down the column that define only the y directional transmissibility multipliers for the corresponding PRESS.			
		dimensionless	dimensionless	dimensionless	None

No.	Name	Description			Default
		Field	Metric	Laboratory	
5	TRANSZ	If the RKTRMDIR is absent from the input deck, then TRANSZ is ignored. If the RKTRMDIR is present in the input deck, then TRANSZ is a columnar vector of real positive values that are either equal or increasing down the column that define only the z directional transmissibility multipliers for the corresponding PRESS.			
		dimensionless	dimensionless	dimensionless	None

Notes:

1) The keyword is followed by NTROCC tables as declared on the ROCKCOMP keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NPPVT rows as declared on the TABDIMS keyword in the RUNSPEC section.
3) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.129: ROCKTAB Keyword Description

Examples

The example below defines two rock compaction tables, assuming NTROCC is equal to two on the ROCKCOMP keyword and NPPVT is greater than or equal to five on the TABDIMS keyword and that the RKTRMDIR keyword is present in the input deck.

```
--
--       ROCK COMPACTION TABLES
--
ROCKTAB
--       PRESS    PORV     TX(YZ)   TY       TZ
--                MULT     MULT     MULT     MULT
--       ------   ------   ------   ------   ------
         1000.0   0.9600   0.9650   0.9650   0.9650
         1500.0   0.9800   0.9850   0.9850   0.9500
         3000.0   0.9900   0.9950   0.9950   0.9950
         4500.0   1.0000   1.0000   1.0000   1.0000
         4750.0   1.0100   1.0100   1.0100   1.0100        / TABLE NO. 01
--       PRESS    PORV     TX(YZ)   TY       TZ
--                MULT     MULT     MULT     MULT
--       ------   ------   ------   ------   ------
         1000.0   0.9600   0.9650   0.9650   0.9650
         1500.0   0.9800   0.9850   0.9850   0.9500
         3000.0   0.9900   0.9950   0.9950   0.9950
         4500.0   1.0000   1.0000   1.0000   1.0000
         4750.0   1.0100   1.0100   1.0100   1.0100        / TABLE NO. 02
```

As the x, y and z directional transmissibility multipliers are identical in the above example, we could eliminate the RKTRMDIR keyword from the input deck and enter the data in the three column format, as shown on the next page.

```
--
--       ROCK COMPACTION TABLES
--
ROCKTAB
--       PRESS    PORV     TX(YZ)
--                MULT     MULT
--       ------   ------   ------
         1000.0   0.9600   0.9650
         1500.0   0.9800   0.9850
         3000.0   0.9900   0.9950
         4500.0   1.0000   1.0000
         4750.0   1.0100   1.0100                          / TABLE NO. 01
--       PRESS    PORV     TX(YZ)
--                MULT     MULT
--       ------   ------   ------
         1000.0   0.9600   0.9650
         1500.0   0.9800   0.9850
         3000.0   0.9900   0.9950
         4500.0   1.0000   1.0000
         4750.0   1.0100   1.0100                          / TABLE NO. 02
```

The net result of the two examples in this case is identical.

8.3.249 ROCKTABH – ROCK COMPACTION HYSTERESIS TABLES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The ROCKTABH keyword defines the rock compaction hysteresis attributes to be applied for when the rock compaction option has been invoked by the ROCKCOMP keyword in the RUNSPEC section. ROCKTABH defines pore volume and transmissibility multipliers versus pressure that are used in the compaction calculations. If the RKTRMDIR has been activated in the PROPS section, then the transmissibility multiplier is directional dependent and two additional columns are used to define the y and z direction transmissibility multipliers. The keyword should only be used if the Rock Compaction Hysteresis option has been activated by either setting the ROCKOPT parameter on the ROCKCOMP keyword to HYSTER or BOBERG.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	PRESS	If the ROCKOPT1 variable has been set to PRESSURE on the ROCKOPTS keyword in the PROPS section, then PRESS should be a columnar vector of real monotonically increasing down the column values, that define the reference pressure for which the other parameters correspond to. If ROCKOPT1 has been set to STRESS, then PRESS should be a columnar vector of real monotonically decreasing down the column values.			
		psia	bars	atm	None
2	PORV	A columnar vector of real positive values that are either equal or increasing down the column that define the rock pore volume multiplier for a given PRESS.			
		dimensionless	dimensionless	dimensionless	None
3	TRANS	If the RKTRMDIR is absent from the input deck, then TRANS is a columnar vector of real positive values that are either equal or increasing down the column that define the x, y, and z directional transmissibility multipliers for the corresponding PRESS. If the RKTRMDIR is present in the input deck, then TRANS is a columnar vector of real positive values that are either equal or increasing down the column that define only the x directional transmissibility multipliers for the corresponding PRESS.			
		dimensionless	dimensionless	dimensionless	None
4	TRANSY	If the RKTRMDIR is absent from the input deck, then TRANSY is ignored. If the RKTRMDIR is present in the input deck, then TRANSY is a columnar vector of real positive values that are either equal or increasing down the column that define only the y directional transmissibility multipliers for the corresponding PRESS.			
		dimensionless	dimensionless	dimensionless	None

No.	Name	Description			Default
		Field	Metric	Laboratory	
5	TRANSZ	If the RKTRMDIR is absent from the input deck, then TRANSZ is ignored. If the RKTRMDIR is present in the input deck, then TRANSZ is a columnar vector of real positive values that are either equal or increasing down the column that define only the z directional transmissibility multipliers for the corresponding PRESS.			
		dimensionless	dimensionless	dimensionless	None

Notes:

1) The keyword is followed by NTROCC tables as declared on the ROCKCOMP keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NPPVT rows as declared on the TABDIMS keyword in the RUNSPEC section.
3) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.130: ROCKTABH Keyword Description

Each data set consists of columnar vectors of pore volume and transmissibility multipliers versus pressure that specify the elastic contraction and expansion and of the reservoir rock. The deflation curve is derived from the first data elements on each elastic curve. If the ROCKOPT parameter on the ROCKCOMP keyword has been set to HYSTER, then the dilation curves are extrapolated to infinite pressure, that is the curves are unbounded. However, if ROCKCOMP is set to BOBERG the last points of each elastic curve are used as the dilation curves.

Example

The example below defines two rock compaction tables, assuming NTROCC is equal to two on the ROCKCOMP keyword and NPPVT is greater than or equal to four on the TABDIMS keyword and that the RKTRMDIR keyword is not present in the input deck.

```
--
--       ROCK COMPACTION HYSTERESIS TABLES
--
ROCKTABH
--       PRESS    PORV     TX(YZ)   TY       TZ
--                MULT     MULT     MULT     MULT
--       ------   ------   ------   ------   ------
         1500.0   0.9600   0.9800
         2500.0   0.9700   0.9850
         3500.0   0.9800   0.9900
         4500.0   0.9900   0.9950                      / NPPVT = 1
         2500.0   0.9900   0.9900
         3500.0   0.9950   0.9950
         4750.0   0.9980   0.9980                      / NPPVT = 2
         3500.0   1.0000   1.0000
         5500.0   1.0100   1.0100                      / NPPVT = 3
         4500.0   1.0100   1.0100
         5750.0   1.0200   1.0200                      / NPPVT = 4

                                                      / TABLE NO. 01
--       PRESS    PORV     TX(YZ)   TY       TZ
--                MULT     MULT     MULT     MULT
--       ------   ------   ------   ------   ------
         1500.0   0.9400   0.9700
         2750.0   0.9400   0.9700                      / NPPVT = 1
         2250.0   0.9800   0.9900
         3250.0   0.9800   0.9900                      / NPPVT = 2
         3000.0   1.0000   1.0000
         4250.0   1.0000   1.0000                      / NPPVT = 3
         4550.0   1.0200   1.0100
         5750.0   1.0200   1.0100                      / NPPVT = 4
                                                      / TABLE NO. 02
```

Here the deflation curve is define for table number one is:

```
         1500.0   0.9600   0.9800
         2500.0   0.9900   0.9900
         3500.0   1.0000   1.0000
         4500.0   1.0100   1.0100
```

and for table number 2:

```
         1500.0   0.9400   0.9700
         2250.0   0.9800   0.9900
         3250.0   1.0000   1.0000
         4250.0   1.0200   1.0100
```

And the dilation curve is define for table number one is:

```
         4500.0   0.9900   0.9950
         4750.0   0.9980   0.9980
         5500.0   1.0100   1.0100
         5750.0   1.0200   1.0200
```

and for table number 2:

```
         2250.0   0.9400   0.9700
         3250.0   0.9800   0.9900
         4250.0   1.0000   1.0000
         5250.0   1.0200   1.0100
```

8.3.250 ROCKTABW – ROCK COMPACTION TABLES (WATER INDUCED)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The ROCKTABW keyword defines the rock compaction tables induced by increasing water saturation within a grid cell due to water invasion, for when the rock compaction option has been invoked by the ROCKCOMP keyword in the RUNSPEC section. ROCKTABW defines pore volume and transmissibility multipliers versus water saturation that are used in the compaction calculations. The keyword should be used together with the ROCK, ROCKTAB or ROCKTABH keywords that specify the pore volume and transmissibility multipliers as functions of pressure. Alternatively the ROCKWNOD, ROCK2D and ROCK2DTR keywords can be used to enter two dimensional tables of the data. All keywords are in the PROPS section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.251 ROCKTHSG – ROCK COMPACTION HYSTERESIS TABLES (DUAL POROSITY)

Description

The ROCKTHSG keyword defines the rock compaction hysteresis attributes to be applied for when the rock compaction option has been invoked by the ROCKCOMP keyword in the RUNSPEC section and the either the Dual Permeability or Dual Porosity models are activated by the DUALPERM and DUALPORO keywords in the RUNSPEC section. ROCKTHSG specifies sigma multipliers versus pressure that are used in the dual porosity rock compaction calculations. The keyword should only be used if the Rock Compaction Hysteresis option has been activated by either setting the ROCKOPT parameter on the ROCKCOMP keyword to one of the available options.

Each data set consists of columnar vectors of sigma multipliers versus pressure that specify the elastic contraction and expansion and of the reservoir rock. The deflation curve is derived from the first data elements on each elastic curve. If the ROCKOPT parameter on the ROCKCOMP keyword has been set to HYSTER, then the dilation curves are extrapolated to infinite pressure, that is the curves are unbounded. However, if ROCKCOMP is set to BOBERG the last points of each elastic curve are used as the dilation curves.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.252 ROCKTSIG – ROCK COMPACTION TABLES (DUAL POROSITY)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The ROCKTSIG keyword defines the rock compaction attributes to be applied for when the rock compaction option has been invoked by the ROCKCOMP keyword in the RUNSPEC section, and the either the Dual Permeability or Dual Porosity models are activated by the DUALPERM and DUALPORO keywords in the RUNSPEC section. ROCKTSIG specifies sigma multipliers versus pressure that are used in the dual porosity rock compaction calculations. The keyword should only be used if the Rock Compaction Hysteresis option has been activated by either setting the ROCKOPT parameter on the ROCKCOMP keyword to one of the available options.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.253 ROCKWNOD – WATER SATURATION VALUES FOR COMPACTION PRESSURE-SW TABLES

Description

The ROCK2D and the ROCK2DTR keywords in the PROPS section define rock compressibility pore volume and transmissibility multipliers as a function of pressure and water saturation ("Sw"), for when the rock compaction option has been invoked by the ROCKCOMP keyword in the RUNSPEC section. The pressure values are defined on ROCK2D and the ROCK2DTR keywords together with the multipliers. This keyword ROCKWNOD, defines the water saturations that are used in conjunction with the ROCK2D and the ROCK2DTR keywords.

This keyword should only be used if compaction option has been enabled.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SWAT	A columnar vector of real monotonically increasing down the column values that defines the water saturations to be associated with the data on the ROCK2D and the ROCKTR keywords.			
		dimensionless	dimensionless	dimensionless	None

Notes:

1) The keyword is followed by NTROCC tables as declared on the ROCKCOMP keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NSSFUN rows as declared on the TABDIMS keyword in the RUNSPEC section.
3) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.131: ROCKWNOD Keyword Description

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

See also the OVERBURD, ROCKTAB, ROCK2D and ROCK2DTR keywords in the PROPS section.

Example

The following example defines two ROCKWNOD tables for the pore volume and transmissibility compaction tables, assuming NTROCC is equal to two on the ROCKCOMP keyword and NSSFUN is greater than or equal to four on the TABDIMS keyword.

```
--
--       WATER SATURATION VALUES FOR COMPACTION PRESSURE-SW TABLES
--
ROCKWNOD
--       COMPACT
--       SWAT
--       ------
          0.000
          0.200
          0.400
          1.000                                          / P-SW SET TABLE NO. 01
--       COMPACT
--       SWAT
--       ------
          0.000
          0.250
          0.750
          1.000                                          / P-SW SET TABLE NO. 02
```

Note that there must be exactly NTROCC tables entered for this keyword, otherwise an error will occur.

8.3.254 RPTPROPS – DEFINE PROPS SECTION REPORTING

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword defines the data in the PROPS section that is to be printed to the output print file in human readable format. The keyword has two distinct forms, the first of which consists of the keyword followed by a series of integers on the next line indicating the data to be printed (see the first example). This is the original formal in the commercial simulator and was subsequently superseded by the second format. The second format consists of the keyword followed by a series of character strings that indicate the data to be printed. In most cases the character string is the keyword used to load the data in the OPM Flow input deck, for example PVDG for the dry gas PVT tables. Its is anticipated that OPM Flow will eventually support the functionality of the second format only, the first format although recognized will be completely ignored.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

No.	Name	Description	Default
1	PVTDG	Print dry gas PVT tables	N/A
2	PVTG	Print wet gas PVT tables	N/A
3	SGFN	Print gas relative permeability saturation function tables.	N/A
4	SGL	Print connate gas saturation array.	N/A
….		….	N/A

Notes:

1) The keyword is terminated by a "/".

Table 8.132: RPTPROPS Keyword Description

Note

Except for tabular like data, PVTDG etc., this keyword has the potential to produce very large print files that some text editors may have difficulty loading. A more efficient solution for array type data is to load the *.INIT file into OPM ResInsight to view the data graphically, this also has the benefit of being able to filter the grid based on I, J, K ranges and grid properties.

Examples

The first example shows the original format of this keyword; although the keyword and format are recognized by OPM Flow, the format is ignored and is unlikely to be implemented in in the simulator.

```
--
--       DEFINE PROPS SECTION REPORT OPTION (ORIGINAL FORMAT)
-
RPTPROPS
         1        2*0      1        3*1                                        /
```

The next example shows the second format of the keyword which may be supported in a future release of OPM Flow.

```
--
--       DEFINE PROPS SECTION REPORT OPTIONS
--
RPTPROPS
         PVDO     SOF2     SGFN     SWFN                                       /
```

8.3.255 RSCONST – DEFINE CONSTANT GOR (RS) FOR ALL DEAD OIL PVT FLUIDS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

RSCONST defines a constant Gas-Oil Ratio ("GOR"), for all dead oil[232] PVT fluids. If the oil has a constant and uniform dissolved gas concentration, GOR, and if the reservoir pressure never drops below the saturation pressure (bubble point pressure), then the model can be run more efficiently by omitting the GAS and DISGAS keywords from the RUNSPEC section, treating the oil as a dead oil, and defining a constant Rs (GOR) value with keywords RSCONST or RSCONSTT in the PROPS section. This results in the model being run as a dead oil problem with no active gas phase. However, OPM Flow takes into account the constant Rs in the calculations and reporting.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	RS	A real positive value that defines the dead oil GOR for all oil PVT tables in the model			
		Mscf/stb	sm^3/sm^3	scc/scc	None
2	PRESS	A real positive value that defines that saturation pressure (bubble point pressure) for all the oil PVT tables in the model.			
		psia	barsa	atma	None
Notes: 1) The keyword is terminated by a "/".					

Table 8.133: RSCONST Keyword Description

See also the RSCONSTT keyword to define a different constant Rs to the various dead oil PVT tables and the PVDO and PVCDO keywords to enter the dead oil properties. All of the aforementioned keywords are in the PROPS section.

Example

The example sets the dead oil GOR to 5 scf/stb and the bubble point pressure to 14.7 psia.

```
--
--       DEAD OIL PVT CONSTANT GOR AND SATURATION PRESSURE
--
RSCONST
--       RS        PSAT
--       MSCF/STB  PSIA
--       --------  ------
          0.0050    14.7                              /
```

[232] *"Dead" oil is oil that it contains no dissolved gas or a relatively thick oil or residue that has lost its volatile components.*

8.3.256 RSCONSTT – DEFINE CONSTANT GOR (RS) FOR EACH DEAD OIL PVT FLUID

Description

RSCONSTT defines a constant Gas-Oil Ratio ("GOR"), for each dead oil[233] PVT fluid in the model. If the oil has a constant and uniform dissolved gas concentration, GOR, and if the reservoir pressure never drops below the saturation pressure (bubble point pressure), then the model can be run more efficiently by omitting the GAS and DISGAS keywords from the RUNSPEC section, treating the oil as a dead oil, and defining a constant Rs (GOR) value with keywords RSCONST or RSCONSTT in the PROPS section. This results in the model being run as a dead oil problem with no active gas phase. However, OPM Flow takes into account the constant Rs in the calculations and reporting.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	RS	A real positive columnar vector that defines the dead oil GOR for each oil PVT table in the model			
		Mscf/stb	sm^3/sm^3	scc/scc	None
2	PRESS	A real positive columnar vector that defines the saturation pressure (bubble point pressure) for each the oil PVT table in the model.			
		psia	barsa	atma	None

Notes:

1) The keyword is followed by NTPVT rows as declared on the TABDIMS keyword in the RUNSPEC section.
2) Each row must contain two values representing the RS and PRESS variables.
3) Each row is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.134: RSCONSTT Keyword Description

See also the RSCONST keyword to define a constant Rs to all the various dead oil PVT tables and the PVDO and PVCDO keywords to enter the dead oil properties. All of the aforementioned keywords are in the PROPS section.

[233] *"Dead" oil is oil that it contains no dissolved gas or a relatively thick oil or residue that has lost its volatile components.*

Example

The example sets the dead oil GOR to 5, 6.5 and 8.0 scf/stb for PVT tables one, two and three, respectively and the bubble point pressure to 14.7 psia for all three tables.

```
--
--       DEAD OIL PVT CONSTANT GOR AND SATURATION PRESSURE
--
RSCONSTT
--       RS        PSAT
--       MSCF/STB  PSIA
--       --------  ------
          0.0050     14.7                                / TABLE NO. 01
          0.0065     14.7                                / TABLE NO. 02
          0.0080     14.7                                / TABLE NO. 03
```

8.3.257 RSGI – DEFINE GAS-OIL RATIO VERSUS PRESSURE AND GI TABLES

Description

The RSGI keyword specifies the saturated oil Gas-Oil Ratio ("GOR") factors used to specify the variation of the maximum possible GOR of oil with respect to pressure and Gi values, for when the GIMODEL keyword in the RUNSPEC section has been used to activate the GI Pseudo Compositional option for the run. See also the GINODE, RSGI, RVGI, BGGI and BOGI keywords in the PROPS section to describe the fluid properties for the GI Pseudo Compositional option.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.258 RTEMP - DEFINE THE INITIAL RESERVOIR TEMPERATURE FOR THE MODEL

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword defines the initial reservoir temperature for the model. Note that the RTEMP keyword is an alias for RTEMPA, and that both keywords are supported by OPM Flow, in both the PROPS and SOLUTION sections, but are treated as being mutually exclusive.

The initial reservoir temperature must be defined when OPM Flow's thermal option has been activated by the THERMAL keyword in the RUNSPEC section. Note this is different to the commercial simulator that uses the TEMP keyword in the RUNSPEC section to activate the black-oil temperature model, and the THERMAL keyword to activate the compositional thermal model.

The initial reservoir temperature should be defined when OPM Flow's CO_2 or H_2 storage option has been activated by the CO2STORE or H2STORE keyword in the RUNSPEC section.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	RTEMP	Single real positive value that defines the reservoir temperature for the model.			
		°F	°C	°C	None
Notes: 1) The keyword is terminated by a "/".					

Table 8.135: RTEMP Keyword Description

See also the RTEMPVD keyword in SOLUTION section to define the reservoir temperature as a function of depth.

Example

```
--
--        RESERVOIR
--        TEMPERATURE
--        -----------
RTEMP
          190.0                                        / RESERVOIR TEMPERATURE
```

The above example defines the reservoir temperature to be 190 oF.

8.3.259 RTEMPA - DEFINE THE INITIAL RESERVOIR TEMPERATURE FOR THE MODEL

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword defines the initial reservoir temperature for the model. Note that the RTEMPA keyword is an alias for RTEMP, and that both keywords are supported by OPM Flow, in both the PROPS and SOLUTION sections, but are treated as being mutually exclusive.

The initial reservoir temperature must be defined when OPM Flow's thermal option has been activated by the THERMAL keyword in the RUNSPEC section. Note this is different to the commercial simulator that uses the TEMP keyword in the RUNSPEC section to activate the black-oil temperature model, and the THERMAL keyword to activate the compositional thermal model.

The initial reservoir temperature should be defined when OPM Flow's CO_2 or H_2 storage option has been activated by the CO2STORE or H2STORE keyword in the RUNSPEC section.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	RTEMPA	Single real positive value that define the reservoir temperature for the model.			
		°F	°C	°C	None
Notes: 1) The keyword is terminated by a "/".					

Table 8.136: RTEMPA Keyword Description

See also the RTEMPVD keyword in SOLUTION section to define the reservoir temperature as a function of depth.

Example

```
--
--        RESERVOIR
--        TEMPERATURE
--        -----------
RTEMPA
          190.0                                        / RESERVOIR TEMPERATURE
```

The above example defines the reservoir temperature to be 190 oF.

8.3.260 RTEMPVD - DEFINE THE INITIAL RESERVOIR TEMPERATURE VERSUS DEPTH TABLES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword defines the initial reservoir temperature versus depth tables for each equilibration region. Note that the RTEMPVD keyword is an alias for TEMPVD, and that both keywords are supported by OPM Flow, in both the PROPS and SOLUTION sections, but are treated as being mutually exclusive.

See *RTEMPVD - Define the Initial Reservoir Temperature versus Depth Tables* in the SOLUTION section, as the keyword is documented in the SOLUTION section, the same as the commercial simulator, but it can also be used in the PROPS section by OPM Flow.

8.3.261 RVCONST – DEFINE CONSTANT CGR (RV) FOR ALL DRY GAS PVT FLUIDS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

RVCONST defines a constant Condensate-Gas Ratio ("CGR" or Rv), for all dry gas[234] PVT fluids. If the gas has a constant and uniform dissolved condensate concentration, and if the reservoir pressure never drops below the saturation pressure (dew point pressure), then the model can be run more efficiently by omitting the OIL and VAPOIL keywords from the RUNSPEC section, treating the gas as a dry gas, and defining a constant Rv (CGR) value with keywords RVCONST or RVCONSTT in the PROPS section. This results in the model being run as a dry gas problem with no active oil phase. However, OPM Flow takes into account the constant Rv in the calculations and reporting.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	RV	A real positive value that defines the dry gas CGR for all dry gas PVT tables in the model			
		stb/Mscf	sm^3/sm^3	scc/scc	None
2	PRESS	A real positive value that defines that saturation pressure (dew point pressure) for all the dry gas PVT tables in the model.			
		psia	barsa	atma	0.0
Notes: 1) The keyword is terminated by a "/".					

Table 8.137: RVCONST Keyword Description

See also the RVCONSTT keyword to define a different constant Rv to the various dry gas PVT tables and the PVDG keyword to enter the dry gas properties. All of the aforementioned keywords are in the PROPS section.

Example

The example sets the dry gas CGR to 5 stb/MMscf and the bubble point pressure to 14.7 psia.

```
--
--       DRY GAS PVT CONSTANT GCR AND SATURATION PRESSURE
--
RVCONST
--       RV         PSAT
--       STB/MSCF   PSIA
--       --------   ------
          0.0050     14.7                                /
```

[234] *Natural gas that occurs in the absence of condensate or liquid hydrocarbons, or gas that had condensable hydrocarbons removed, is called dry gas. It is primarily methane with some intermediates. The hydrocarbon mixture is solely gas in the reservoir and there is no liquid (condensate surface liquid) formed either in the reservoir or at surface. The term dry indicates that the gas does not contain heavier hydrocarbons to form liquids at the surface conditions. Dry gas typically has GOR's greater than 100,000 scf/stb or 18,000 Sm3/m3.*

8.3.262 RVCONSTT – DEFINE CONSTANT CGR (RV) FOR EACH DRY GAS PVT FLUID

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

RVCONSTT defines a constant Condensate-Gas Ratio ("CGR" or Rv), for each dry gas[235] PVT fluid. If the gas has a constant and uniform dissolved condensate concentration, and if the reservoir pressure never drops below the saturation pressure (dew point pressure), then the model can be run more efficiently by omitting the OIL and VAPGAS keywords from the RUNSPEC section, treating the gas as a dry gas, and defining a constant Rv (CGR) value with keywords RVCONST or RVCONSTT in the PROPS section. This results in the model being run as a dry gas problem with no active oil phase. However, OPM Flow takes into account the constant Rv in the calculations and reporting.

This keyword is ignored by OPM Flow but is documented here for completeness.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	RS	A real positive value that defines the dry gas CGR for each dry gas PVT table in the model			
		stb/Mscf	sm^3/sm^3	scc/scc	None
2	PRESS	A real positive value that defines that saturation pressure (dew point pressure) for each dry gas PVT table in the model.			
		psia	barsa	atma	0.0

Notes:

1) The keyword is followed by NTPVT rows as declared on the TABDIMS keyword in the RUNSPEC section.
2) Each row must contain two values representing the RV and PRESS variables.
3) Each row is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.138: RVCONSTT Keyword Description

See also the RVCONST keyword to define a constant Rv to all the various dry gas PVT tables and the PVDG keyword to enter the dry gas properties. All of the aforementioned keywords are in the PROPS section.

[235] *Natural gas that occurs in the absence of condensate or liquid hydrocarbons, or gas that had condensable hydrocarbons removed, is called dry gas. It is primarily methane with some intermediates. The hydrocarbon mixture is solely gas in the reservoir and there is no liquid (condensate surface liquid) formed either in the reservoir or at surface. The term dry indicates that the gas does not contain heavier hydrocarbons to form liquids at the surface conditions. Dry gas typically has GOR's greater than 100,000 scf/stb or 18,000 Sm3/m3.*

Example

The example sets the dry gas CGR to 5, 6.5 and 8.0 stb/MMscf for PVT tables one, two and three, respectively and the bubble point pressure to 14.7 psia for all three tables.

```
--
--       DRY GAS PVT CONSTANT GCR AND SATURATION PRESSURE
--
RVCONSTT
--       RV         PSAT
--       STB/MSCF   PSIA
--       --------   ------
          0.0050      14.7                                  / TABLE NO. 01
          0.0065      14.7                                  / TABLE NO. 02
          0.0080      14.7                                  / TABLE NO. 03
```

8.3.263 RVGI – DEFINE CONDENSATE-GAS RATIO VERSUS PRESSURE AND GI TABLES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The RVGI keyword specifies the saturated gas Condensate-Gas Ratio ("CGR") factors used to specify the variation of the maximum possible CGR of gas with respect to pressure and Gi values, for when the GIMODEL keyword in the RUNSPEC section has been used to activate the GI Pseudo Compositional option for the run. See also the GINODE, RSGI, RVGI, BGGI and BOGI keywords in the PROPS section to describe the fluid properties for the GI Pseudo Compositional option.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.264 RWGSALT – WATER VAPORIZATION VERSUS PRESSURE AND SALT CONCENTRATION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

RWGSALT defines the relationship of water vaporization versus pressure and salt concentration. This keyword should be used when the VAPWAT keyword has be declared in the RUNSPEC section indicating that vaporized water is present in the gas phase. In addition, if the Salt Precipitation model has been activated via the BRINE and PRECSALT keywords, also in the RUNSPEC section, then this keyword must be present. The keyword may be used for gas-water and oil-water-gas input decks that contain the either dry or wet gas and vaporized water phases.

Note

This is an OPM Flow specific keyword for the simulator's Water Vaporization and Salt Precipitation Models, note that these are extensions to the simulator's standard Brine model.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	PRESS	A real monotonically increasing down the column values that define the gas phase pressure, that together with salt concentration, defines the vaporized water in gas ratio ("VWGR") or Rw			
		psia	barsa	atma	None
2	SALTCON	A real monotonically increasing positive columnar vector defining the salt concentration in water.			
		lb/stb	kg/sm3	gm/scc	None
3	RW	A columnar vector of real positive number values defining the vaporized water in gas ratio (Rw) that for a given PRESS and SALTCON.			
		stb/Mscf	sm^3/sm^3	scc/scc	None

Notes:

1) Each table is terminated by a "/" including the last table; however, there is no "/" terminator for the keyword.

Table 8.139: RWGSALT Keyword Description

Since the water component is in both the water and the gas phases, RWGSALT controls the amount of water component evaporated from the water phase into the gas phase, which is a function of both the water phase salinity and a grid cells pressure. The keyword should be used in conjunction with the dry gas PVTGW keyword or the wet gas PVTGWO keyword, both of which are in the PROPS section.

Example

The example defines two RWGSALT tables assuming NTPVT equals two and NPPVT is greater than or equal to eight on the TABDIMS keyword in the RUNSPEC section.

```
--
--       WATER VAPORIZATION TABLE FOR BRINE  (OPM FLOW KEYWORD)
--
RWGSALT
--       PRES     SALTCONC    RVW
--       PSIA     LB/STB      STB/MSCF
--       ------  ----------   ---------
          300     0           0.000132
                  0.5         0.000132
                  1           0.000132                    /
           600    0           0.000132
                  0.5         0.000132
                  1           0.000132                    /
           900    0           0.000132
                  0.5         0.000132
                  1           0.000132                    /
          1200    0           0.000132
                  0.5         0.000132
                  1           0.000132                    /
          1500    0           0.000132
                  0.5         0.000132
                  1           0.000132                    /
          1800    0           0.000132
                  0.5         0.000132
                  1           0.000132                    /
          2100    0           0.000132
                  0.5         0.000132
                  1           0.000132                    /
          2400    0           0.000132
                  0.5         0.000132
                  1           0.000132                    /
                                                          / Table NO. 1
--       PRES     SALTCONC    RW
--       PSIA     LB/STB      STB/MSCF
--       ------  ----------   ---------
          300     0           0.000132
                  0.5         0.000132
                  1           0.000132                    /
          600     0           0.000132
                  0.5         0.000132
                  1           0.000132                    /
          900     0           0.000132
                  0.5         0.000132
                  1           0.000132                    /
          1200    0           0.000132
                  0.5         0.000132
                  1           0.000132                    /
          1500    0           0.000132
                  0.5         0.000132
                  1           0.000132                    /
          1800    0           0.000132
                  0.5         0.000132
                  1           0.000132                    /
          2100    0           0.000132
                  0.5         0.000132
                  1           0.000132                    /
          2400    0           0.000132
                  0.5         0.000132
                  1           0.000132                    /
                                                          / Table NO. 2
```

8.3.265 SALINITY – DEFINE THE RESERVOIR SALINITY FOR ALL CELLS

Description

SALINITY defines a grid blocks salinity for all cells. The keyword should only be used with OPM Flow's CO2-Brine model which is activated via the CO2STORE keyword in the RUNSPEC section. This keyword is a compositional keyword in the commercial simulator but has been implemented in OPM Flow's black-oil CO2-Brine model.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SALINITY	A real positive value that defines the salinity for all grid blocks in the model for when the CO2-Brine model has been activated. Note that the units for salinity are molality, that is gm-M/Kg, and therefore the units are defined as given below with the 10^{-3} prefix.			
		10-3 x lb-M/lb	10-3 x kg-M/kg	10-3 x gm-M/gm	None
Notes: 1) The keyword is terminated by a "/".					

Table 8.140: SALINITY Keyword Description

See also the CO2STORE keyword in the RUNSPEC section.

Example

The example sets the salt salinity for all cells in the model to 0.001 lb-M/lb.

```
--
--       SET SALINITY FOR ALL CELLS (OPM FLOW KEYWORD)
--
SALINITY
         1.0                                                  /
```

Note that units for salinity are to the 10-3, that is a value of 0.001 lb-M/lb should be entered as 1.0 lb-M/lb, as per the example.

8.3.266 SALTNODE – SALT CONCENTRATION BASED PVTNUM ARRAY

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

SALTNODE defines the salt concentration value based on a cells PVTNUM number. The SALTNODE property is used in the calculation of a polymer viscosity when the polymer and the salt options has been activated by the POLYMER and BRINE keywords in the RUNSPEC section. In the RUNSPEC section the number of PVTNUM functions is declared by NTPVT variable on the TABDIMS keyword and allocated to individual cells by the PVTNUM property array in the REGIONS section. NPPVT on the TABDIMS keyword in the RUNSPEC section defines the maximum number of rows (or pressure values) in the PVT tables and also sets the maximum number of entries for each SALNODE data set. The number of values for each data set must correspond to the number of polymer solution adsorption entries on the PLYADSS keyword. For example if there are three sets of PVT tables and four values on the PLYADSS keyword, then three SALTNODE data sets with four values of salt concentrations need to be entered.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SALTNODE	A real monotonically increasing positive columnar vector defining the salt concentration for a given PVTNUM table.			
		lb/stb	kg/sm3	gm/scc	None
Notes: 1) Each table is terminated by a "/" including the last table; however, there is no "/" terminator for the keyword.					

Table 8.141: SALTNODE Keyword Description

An alternative manner of entering the salt concentrations is by utilizing the PVTNUM region array by using the ADSALNOD keyword in the PROPS section.

Example

Given three sets of relative permeability tables and four values on the PLYADSS keyword and two SALNODE data sets with four values of salt concentrations then the data should be entered as follows:

```
--
--    SETS SALT CONCENTRATION FOR POLYMER SOLUTION ADSORPTION
--    VIA PVTNUM ARRAY ALLOCATION
--
--    SALT
--
SALTNODE
      1.0
      5.0
      10.5
      25.0      / PVTNUM TABLE NO. 01
      1.0
      3.0
      7.5
      15.0      / PVTNUM TABLE NO. 02
```

See also the ADSALNOD keyword.

8.3.267 SALTSOL – DEFINE THE SALT SOLUBILITY LIMIT BY REGION

Description

SALTSOL defines a grid block's maximum salt solubility for each PVTNUM region. The keyword should only be used with OPM Flow's Salt Precipitation model which is activated via the PRECSALT keyword in the RUNSPEC section.

Note

This is an OPM Flow specific keyword for the simulator's Salt Precipitation model that is activated by the PRECSALT keyword and declaring that vaporized water is present in the run via the VAPWAT in the RUNSPEC section.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SALTSOL	A real positive value that defines the maximum salt solubility for all grid blocks in a PVTNUM region.			None
		lb/stb	kg/sm3	gm/scc	
2	SALTDEN	SALTDEN is a real number defining the density of salt at surface conditions.			
		lb/ft^3 135.469	kg/m^3 2170	gm/cc 2.170	Defined

Notes:

1) The keyword is followed by NTPVT tables as declared on the TABDIMS keyword in the RUNSPEC section.
2) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.142: SALTSOL Keyword Description

See also the PRECSALT and VAPWAT keywords in the RUNSPEC section and the PVTGW and PVTGWO keywords in the PROPS section.

Example

The first example sets the maximum salt solubility for all cells in the model to 134.6 lb/stb, assuming that there is only one PVT region, that is NTPVT is equal to one on the TABDIMS keyword in the RUNSPEC section.

```
--
--       SET SALT SOLUBILITY LIMIT FOR EACH REGION (OPM FLOW KEYWORD)
--
SALTSOL
--       MAX       SALT
--       SALTSOL   DENSITY
         134.6     1*                                              /
```

The 134.6 lb/stb, (380 kg/sm3 or 0.384 gm/scc for metric and laboratory units, respectively) is based on the solubility of NACL at 212 °F (100 °C) and should be used with care.

The next example shows how to set the maximum salt solubility for when NTPVT is equal to three on the TABDIMS keyword in the RUNSPEC section.

```
--
--       SET SALT SOLUBILITY LIMIT FOR EACH REGION (OPM FLOW KEYWORD)
--
SALTSOL
         134.6                                          / PVT REGION NO. 1
         124.0                                          / PVT REGION NO. 2
                                                        / PVT REGION NO. 3
```

Here the last entry, which is for region number three, is defaulted, and results in region's three maximum salt solubility to take the previous value, in this case 124.0 lb/stb.

8.3.268 SCALECRS – DEFINE END-POINT SCALING OPTION

Description

The SCALECRS keyword sets the end-point scaling option to be either two-point or three-point scaling, for when the End-Point Scaling option has been invoked by the ENDSCALE keyword in the RUNSPEC section. This determines which end-points on the relative permeability curves are used for scaling based on the supplied end-point arrays (SGCR, SWCR, etc.).

No.	Name	Description	Default
1	SCALEOPT	SCALEOPT is a character string that sets the end-point scaling option and should be set to either NO or YES: 1) NO: Activates two-point end-point scaling. 2) YES: Activates three-point end-point	NO
Notes: 1) The keyword is terminated by a "/".			

Table 8.143: SCALECRS Keyword Description

The end-point scaled for each option and the arrays used in the end-point scaling are summarized in the following table:

Option	Phases	Relative Permeability End-Point	Minimum Saturation End-Point	Middle Saturation End-Point	Maximum Saturation End-Point
Two-Point	Water	KRW	SWCR		SWU
	Gas	KRG	SGCR		SGU
	Oil-Water	KRORW	SOWCR		(1.0 – SWL - SGL)
	Oil-Gas	KRORG	SOGCR		(1.0 – SWL - SGL)
Three-Point	Water	KRW	SWCR	(1.0 – SOWCR - SGL)	SWU
	Gas	KRG	SGCR	(1.0 - SOGCR-SWL)	SGU
	Oil-Water	KRORW	SOWCR	(1.0 – SWCR - SGL)	(1.0 – SWL - SGL)
	Oil-Gas	KRORG	SOGCR	(1.0 – SGCR - SGL)	(1.0 – SWL - SGL)
	Two Phase Gas-Water Simulations				
	Water	KRW	SWCR	(1.0 - SGCR)	SWU
	Gas	KRG	SGCR	(1.0 -SWCR)	SGU

Table 8.144: End-Point Arrays Used in the End-Point Scaling Options

See also the TZONE keyword in the PROPS section that sets the transition zone end-point scaling options for the oil, gas and water phases.

Example

```
--
--       TWO-POINT END-POINT SCALING IS NO THREE POINT IS YES
--
--       SCALEOPT
--       ---------
SCALECRS
         YES                                                  / SCALING OPTION
```

The above example activates three-point end-point scaling of the relative permeability curves.

8.3.269 SCALELIM – END-POINT SCALING VERSUS DEPTH MAXIMUM WATER SATURATION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword defines the maximum water saturation allowed in a cell for when the end-point versus depth tables are used in the End-Point Scaling option to calculate the water saturation for a grid block. The End-Point Scaling option must be invoked by the ENDSCALE keyword in the RUNSPEC section to use this keyword, and the keyword may only be used in two phase runs containing water, or if the Miscible Flood option has been activated by the MISCIBLE keyword in the RUNSPEC section. This keyword functionality is not supported in OPM Flow.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.270 SDENSITY – DEFINE THE MISCIBLE OR SOLVENT SURFACE GAS DENSITY

Description

The SDENSITY keyword defines density at surface conditions of either the miscible injection gas for when the MISCIBLE option has been invoked in the RUNSPEC section, or the solvent for when the SOLVENT option has been invoked in the RUNSPEC section. This keyword must be invoked if either the MISCIBLE or SOLVENT options have been activated in the RUNSPEC section.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SDENSITY	SDENSITY is a real positive number defining the density at surface conditions of either: 1) the miscible injection gas for when the MISCIBLE option has been invoked in the RUNSPEC section, or, 2) the solvent for when the SOLVENT option has been invoked in the RUNSPEC section.			
		lb/ft^3	kg/m^3	gm/cc	None
Notes: 1) The keyword is followed by NTPVT tables as declared on the TABDIMS keyword in the RUNSPEC section. 2) Each table is terminated by a "/" and there is no "/" terminator for the keyword.					

Table 8.145: SDENSITY Keyword Description

In addition to this keyword, the surface density or gravity of the in-place natural gas must be entered using either the DENSITY or GRAVITY keywords.

Examples

The following shows the SDENSITY keyword for when NTPVT on the TABDIMS keyword in the RUNSPEC section is set to one.

```
--
--       MIS-SOL
--       DENSITY
--       -------
SDENSITY
         0.04520                                        / MIS-SOL DENSITY
```

The next example shows the SDENSITY keyword for when NTPVT on the TABDIMS keyword in the RUNSPEC section is set to three.

```
--
--       MIS-SOL
--       DENSITY
--       -------
SDENSITY
         0.04520                                        / MIS-SOL DENSITY 1
         0.05520                                        / MIS-SOL DENSITY 2
         0.06420                                        / MIS-SOL DENSITY 3
```

There is no terminating "/" for this keyword.

8.3.271 SGCR – END-POINT SCALING GRID CELL CRITICAL GAS SATURATIONS

Description

SGCR defines the critical gas saturation for all the cells in the model via an array when the end-point scaling option has been invoked via the ENDSCALE keyword in the RUNSPEC section. The critical gas saturation is defined as the maximum gas saturation for which the gas relative permeability is zero in a two-phase relative permeability table.

The keyword can be used with all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SGCR	SGCR is an array of real numbers assigning the critical gas saturation values to each cell in the model. The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword. Repeat counts may be used, for example 30*0.03			Taken from cell allocated relative permeability table.
		dimensionless	dimensionless	dimensionless	

Notes:

1) Note this the directional independent version of the critical gas saturation used with the end-point scaling option. If directional end-point scaling has been activated then the SGCRX± , SGCRX± and SGCRX± series of keyword should be used.
2) If the value for a cell has been defaulted then OPM Flow uses the value from the cell's relative permeability table.
3) The keyword is terminated by a "/".

Table 8.146: SGCR Keyword Description

End-point scaling allows the entered relative permeability functions to be re-scaled based on the saturation end-points allocated to each cell by the SWL, SWCR, SWU, SGL, SGCR, SGU, SOWCR, and SOGCR saturation grid arrays for the saturation end-points, and the KRG, KRGR, KRO, KRORG, KRORW, KRW and KRWR relative permeability grid cell arrays for the relative permeability end-point data. In addition end-point scaling may be directional dependent in which case the directional dependent versions of the aforementioned arrays should be used, that is SGCRX, SGCRY and SGCRZ instead of SGCR. There is also the facility to make the directional end-point scaling reversible or non-reversible and if the non-reversible option is selected the non-reversible versions of the aforementioned arrays should be used, that is SGCRX, SGCRX-, SGCRY, SGCRY-, SGCRZ and SGCRZ-, instead of the SGCR keyword.

Example

```
--
--       DEFINE GRID BLOCK END-POINT SGCR DATA FOR ALL CELLS
--       (FOR NX x NY x NZ = 300)
--
SGCR
         300*0.050                                                    /
```

The above example defines a constant critical gas saturation of 0.05 to all 300 cells in the model as defined by the DIMENS keyword in the RUNSPEC section.

8.3.272 SGCWMIS – MISCIBLE CRITICAL GAS VERSUS WATER SATURATION FUNCTIONS

Description

SGCWMIS defines the dependency between the miscible critical gas saturation and the water saturation, for when the MISCIBLE keyword in the RUNSPEC section has been activated. The keyword can only be used with the MISCIBLE option and for when the oil, water and gas phases are active in the model.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SWAT	A columnar vector of real monotonically increasing down the column values starting from zero and terminating atone, that defines the water saturation.			
		dimensionless	dimensionless	dimensionless	None
2	SGCMIS	A columnar vector of real equal or increasing down the column values that are greater than or equal to zero and less than one, that define the corresponding miscible gas critical gas saturation for the corresponding water saturation SWAT.			
		dimensionless	dimensionless	dimensionless	None

Notes:

1) The keyword is followed by NTMISC tables as declared on the MISCIBLE keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NSMISC rows as declared on the MISCIBLE keyword in the RUNSPEC section.
3) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.147: SGCWMIS Keyword Description

Example

```
--
--       MISCIBLE CRITICAL GAS VERSUS WATER SATURATION TABLE
--
SGCWMIS
--       SWAT       SGCRMIS
--       FRAC       FRAC
--       -------    --------
          0.0000     0.0000
          0.2000     0.0300
          1.0000     0.0300                              / TABLE NO. 01
--       SWAT       SGCRMIS
--       FRAC       FRAC
--       -------    --------
          0.0000     0.0000
          0.3000     0.0500
          1.0000     0.0500                              / TABLE NO. 02
```

The above example defines two miscible critical gas saturation versus water saturation tables assuming NTMISC equals two and NSMISC is greater than or equal to three on the MISCIBLE keyword in the RUNSPEC section.

8.3.273 SGF32D – GAS SATURATION TABLES VERSUS OIL AND WATER SATURATIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The SGF32D keyword defines the gas relative permeability as a function of both oil and water saturations. This keyword should only be used if the gas is present in the run.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

See also the SWOF, SGOF, SLGOF series of keywords and the SWFN, SGFN, SGWFN, SOF2, SOF3, SOF32D series of keywords to enter relative permeability data.

8.3.274 SGFN – GAS SATURATION TABLES (FORMAT TYPE 2)

Description

The SGFN keyword defines the gas relative permeability and oil-gas capillary pressure data versus gas saturation tables for when gas is present in the input deck. This keyword should only be used if the gas is present in the run.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SGAS	A columnar vector of real monotonically increasing down the column values starting from zero and terminating at one, that defines the gas saturation.			
		dimensionless	dimensionless	dimensionless	None
2	KRG	A columnar vector of real values that are either equal or increasing down the column and that are greater than or equal to zero and less than or equal to one that defines the gas relative permeability.			
		dimensionless	dimensionless	dimensionless	None
3	PCOG	A columnar vector of real values that are either equal or increasing down the column that defines the oil-gas capillary pressure.			
		psia	bars	atm	None

Notes:

1) The keyword is followed by NTSFUN tables as declared on the TABDIMS keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NSSFUN rows as declared on the TABDIMS keyword in the RUNSPEC section.
3) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.148: SGFN Keyword Description

See also the *GSF - Gas Saturation Function Tables (Gas-Water Systems)* keyword in the PROPS section, that defines the gas relative permeability and gas-water capillary pressure data as a function of gas saturation, for when only the gas and water phases are present in the model. Note that the GSF keyword must be used in conjunction with the *WSF - Water Saturation Tables versus Water Saturation (Gas-Water and CO2STORE Systems)* keyword, also in the PROPS section. WSF defines the water saturation as a function of water saturation for when only the gas and water phases are present in the model.

Example

```
--
--       GAS RELATIVE PERMEABILITY TABLES (SGFN)
--
SGFN
--       SGAS       KRG        PCGO
--       FRAC                  PSIA
--       --------  --------   -------
             0.00    0.0000      0.0
             0.05    0.0000      0.0
             0.10    0.0000      0.0
             0.15    0.0000      0.0
             0.20    0.0002      0.0
             0.25    0.0010      0.0
             0.30    0.0062      0.0
             0.35    0.0140      0.0
             0.40    0.0273      0.0
             0.45    0.0450      0.0
             0.50    0.0707      0.0
             0.55    0.1020      0.0
             0.60    0.1412      0.0
             0.65    0.1870      0.0
             0.70    0.2412      0.0
             0.77    0.3288      0.0
             0.82    0.4000      0.0
             0.85    0.4450      0.0                    / TABLE NO. 01
--       SGAS       KRG        PCGO
--       FRAC                  PSIA
--       --------  --------   -------
             0.00    0.0000      0.0
             0.05    0.0000      0.0
             0.10    0.0000      0.0
             0.15    0.0000      0.0
             0.20    0.0002      0.0
             0.25    0.0010      0.0
             0.30    0.0062      0.0
             0.35    0.0140      0.0
             0.40    0.0273      0.0
             0.45    0.0450      0.0
             0.50    0.0707      0.0
             0.55    0.1020      0.0
             0.60    0.1412      0.0
             0.65    0.1870      0.0
             0.70    0.2412      0.0
             0.77    0.3288      0.0
             0.82    0.4000      0.0
             0.85    0.4450      0.0                    / TABLE NO. 02
```

The example defines two SGFN tables for when gas is present in the input deck.

8.3.275 SGL – END-POINT SCALING GRID CELL CONNATE GAS SATURATIONS

Description

SGL defines the connate gas saturation for all the cells in the model via an array when the end-point scaling option has been invoked via the ENDSCALE keyword in the RUNSPEC section. The connate gas saturation is defined as the minimum gas saturation in a two-phase gas relative permeability table.

The keyword can be used with all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SGL	SGL is an array of real numbers assigning the connate gas saturation values to each cell in the model. The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword. Repeat counts may be used, for example 30*0.03			Taken from cell allocated relative permeability table.
		dimensionless	dimensionless	dimensionless	

Notes:

1) Note this the directional independent version of the connate gas saturation array used with the end-point scaling option. If directional end-point scaling has been activated then the SGLX± , SGLY± and SGZ± series of keyword should be used.
2) If the value for a cell has been defaulted then OPM Flow uses the value from the cell's relative permeability table.
3) The keyword is terminated by a "/".

Table 8.149: SGL Keyword Description

End-point scaling allows the entered relative permeability functions to be re-scaled based on the saturation end-points allocated to each cell by the SWL, SWCR, SWU, SGL, SGCR, SGU, SOWCR, and SOGCR saturation grid arrays for the saturation end-points, and the KRG, KROG, KROW and KRW relative permeability grid cell arrays for the relative permeability end-point data. In addition end-point scaling may be directional dependent in which case the directional dependent versions of the aforementioned arrays should be used, that is SGLX, SGLY and SGLZ instead of SGL. There is also the facility to make the directional end-point scaling reversible or non-reversible and if the non-reversible option is selected the non-reversible versions of the aforementioned arrays should be used, that is SGLX, SGLX-, SGLY, SGLY-, SGLZ and SGLZ-, instead of the SGL keyword.

Example

```
--
--       DEFINE GRID BLOCK END-POINT SGL DATA FOR ALL CELLS
--      (FOR NX x NY x NZ = 300)
--
SGL
         300*0.030                                                    /
```

The above example defines a constant connate gas saturation of 0.03 to all 300 cells in the model as defined by the DIMENS keyword in the RUNSPEC section.

8.3.276 SGLPC – END-POINT SCALING GRID CELL CAPILLARY PRESSURE CONNATE GAS SATURATIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

SGLPC defines the connate gas saturation for all the cells in the model via an array when the end-point scaling option has been invoked via the ENDSCALE keyword in the RUNSPEC section. The connate gas saturation is defined as the minimum gas saturation in a two-phase gas relative permeability table. The keyword only applies the scaling to the drainage capillary pressure tables, unlike the SGL keyword that applies the scaling to both the capillary pressure and relative permeability tables. The keyword can be used with all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SGLPC	SGLPC is an array of real numbers assigning the connate gas saturation values to each cell in the model. The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword. If SGLPC is omitted from the input deck the values will be defaulted to those on the SGL series of keywords. If the SGL series of keywords are missing from the input deck then the values are taken from the cell allocated capillary pressure table. Repeat counts may be used, for example 30*0.03			Taken from SGL or from the cell allocated capillary pressure table.
		dimensionless	dimensionless	dimensionless	

Notes:

1) Note this the directional independent version of the connate gas saturation array used with the end-point scaling option. If directional end-point scaling has been activated then the SGLX± , SGLY± and SGZ± series of keyword should be used.

2) The keyword is terminated by a "/".

Table 8.150: SGLPC Keyword Description

End-point scaling allows the entered relative permeability functions to be re-scaled based on the saturation end-points allocated to each cell by the SWL, SWCR, SWU, SGL, SGCR, SGU, SOWCR, and SOGCR saturation grid arrays for the saturation end-points, and the KRG, KRGR, KRO, KRORG, KRORW, KRW and KRWR relative permeability grid cell arrays for the relative permeability end-point data. In addition end-point scaling may be directional dependent in which case the directional dependent versions of the aforementioned arrays should be used, that is SGLX, SGLY and SGLZ instead of SGL or SGLPC. There is also the facility to make the directional end-point scaling reversible or non-reversible and if the non-reversible option is selected the non-reversible versions of the aforementioned arrays should be used, that is SGLX, SGLX-, SGLY, SGLY-, SGLZ and SGLZ-, instead of the SGL or SGLPC keywords.

Missing Some Functionality - Use with Caution.

Example

```
--
--       DEFINE GRID BLOCK END-POINT SGLPC DATA FOR ALL CELLS
--       (FOR NX x NY x NZ = 300)
--
SGLPC
         300*0.030                                                              /
```

The above example defines a constant connate gas saturation of 0.03 to all 300 cells in the model as defined by the DIMENS keyword in the RUNSPEC section.

8.3.277 SGOF – GAS-OIL SATURATION TABLES VERSUS GAS (FORMAT TYPE 1)

Description

The SGOF keyword defines the oil and gas relative permeability and oil-gas capillary versus gas saturation tables for when oil and gas are present in the input deck. This keyword should only be used if both oil and gas are present in the run.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SGAS	A columnar vector of real monotonically increasing down the column values starting from zero and terminating at one, that defines the gas saturation.			
		dimensionless	dimensionless	dimensionless	None
2	KRG	A columnar vector of real values that are either equal or increasing down the column and that are greater than or equal to zero and less than or equal to one that defines the gas relative permeability.			
		dimensionless	dimensionless	dimensionless	None
3	KRO	A columnar vector of real values that are either equal or decreasing down the column and that are greater than or equal to zero and less than or equal to one that defines the oil relative permeability with respect to gas and connate water saturation. When water is active in the run, the first entry the column, that is at krog(Sg = 0), must be the same as the first entry in the corresponding SWOF table, that is at krow(So = 1 - Swco). The last value in the column should be zero.			
		dimensionless	dimensionless	dimensionless	None
4	PCOG	A columnar vector of real values that are either equal or increasing down the column that defines the oil-gas relative capillary pressure.			
		psia	bars	atm	None

Notes:

1) The keyword is followed by NTSFUN tables as declared on the TABDIMS keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NSSFUN rows as declared on the TABDIMS keyword in the RUNSPEC section.
3) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.151: SGOF Keyword Description

Example

The following example is based on NTSFUN equals two on the TABDIMS keyword in the RUNSPEC section.

```
--
--       GAS-OIL RELATIVE PERMEABILITY TABLES (SGOF)
SGOF
--       SG          KRG        KROG       PCOG
--       FRAC                              PSIA
--       -------   --------   -------    -------
         0.00000   0.000000   0.90000    0.0000
         0.03000   0.000000   0.82500    0.0000
         0.80000   0.900000   0.00000    0.0000             / TABLE No. 01
--       -------   --------   -------    -------
         0.00000   0.000000   0.90000    0.0000
         0.03000   0.000000   0.82500    0.0000
         0.04420   0.024200   0.80000    0.0000
         0.05850   0.048500   0.77500    0.0000
         0.07270   0.072700   0.75000    0.0000
         0.08700   0.097000   0.72500    0.0000
         0.10120   0.121200   0.70000    0.0000
         0.11550   0.145500   0.67500    0.0000
         0.12970   0.169700   0.65000    0.0000
         0.14390   0.193900   0.62500    0.0000
         0.15820   0.218200   0.60000    0.0000
         0.17240   0.242400   0.57500    0.0000
         0.18670   0.266700   0.55000    0.0000
         0.20090   0.290900   0.52500    0.0000
         0.21520   0.315200   0.50000    0.0000
         0.22940   0.339400   0.47500    0.0000
         0.24360   0.363600   0.45000    0.0000
         0.25790   0.387900   0.42500    0.0000
         0.27210   0.412100   0.40000    0.0000
         0.28640   0.436400   0.37500    0.0000
         0.30060   0.460600   0.35000    0.0000
         0.31480   0.484800   0.32500    0.0000
         0.32910   0.509100   0.30000    0.0000
         0.34330   0.533300   0.27500    0.0000
         0.35760   0.557600   0.25000    0.0000
         0.37180   0.581800   0.22500    0.0000
         0.38610   0.606100   0.20000    0.0000
         0.40030   0.630300   0.17500    0.0000
         0.41450   0.654500   0.15000    0.0000
         0.42880   0.678800   0.12500    0.0000
         0.44300   0.703000   0.10000    0.0000
         0.45730   0.727300   0.07500    0.0000
         0.47150   0.751500   0.05000    0.0000
         0.48580   0.775800   0.02500    0.0000
         0.50000   0.800000   0.00000    0.0000
         0.80000   0.900000   0.00000    0.0000             / TABLE No. 02
```

The example defines two SGOF tables for use when oil, gas and water are present in the run.

8.3.278 SGOFLET– GAS-OIL LET RELATIVE PERMEABILITY FUNCTIONS

Description

SGOFLET defines the relative permeability and capillary pressure parameters for the gas-oil LET family of models. Both the gas and oil phases should be made active in the model via the GAS and OIL keywords in the RUNSPEC section. See section *8.2.6Saturation Table Generation - LET Functions* and Lomeland et al.[236],[237] [238] and [239] for further information on the model.

The keyword is used as a replacement for the SGOF keyword for three-phase oil-gas-water systems, and the LET series of keywords cannot be combined with the standard set of relative permeability keywords.

Note

This is an OPM Flow specific keyword and will therefore cause an error in the commercial simulator.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SGL	SGL is a real positive number less than one that defines the connate gas saturation, that is smallest gas saturation in the LET function.			0.0
		dimensionless	dimensionless	dimensionless	
2	SGCR	SGCR is a real positive number greater than or equal to SGL and less than one, that defines the critical gas saturation, that is the largest gas saturation for which the gas relative permeability is zero.			0.0
		dimensionless	dimensionless	dimensionless	
3	LGAS	LGAS is a real positive number that defines the LET Lower empirical parameter Lg for the gas phase with the associated oil phase in the LET relative permeability equation.			1.0
		dimensionless	dimensionless	dimensionless	
4	EGAS	EGAS is a real positive number that defines the LET Elevation empirical parameter Eg for the gas phase with the associated oil phase in the LET relative permeability equation.			1.0
		dimensionless	dimensionless	dimensionless	

[236] *Lomeland F., Ebeltoft E. and Thomas W.H., 2005. A New Versatile Relative Permeability Correlation. Paper SCA2005-32 presented at the International Symposium of the Society of Core Analysts held in Toronto, Canada, 21-25 August, 2005.*

[237] *Lomeland F. and Ebeltoft E., 2008. A New Versatile Capillary Pressure Correlation. Paper SCA2008-08 presented at the International Symposium of the Society of Core Analysts held in Abu Dhabi, UAE, 29 Oct. – 2 Nov., 2008.*

[238] *Lomeland F., Hasanov B., Ebeltoft E. and Berge M., 2012. A Versatile Representation of Up-scaled Relative Permeability for Field Applications. Paper SPE 154487-MS presented at the EAGE Annual Conference & Exhibition incorporating SPE Europec held in Copenhagen, Denmark, 4-7 June 2012.*

[239] *Lomeland F., 2018.. Overview Of The Let Family Of Versatile Correlations For Flow Functions. Paper SCA2018-056 presented at the International Symposium of the Society of Core Analysts held in Trondheim, Norway, 27-30 August 2018.*

No.	Name	Description			Default
		Field	Metric	Laboratory	
5	TGAS	TGAS is a real positive number that defines the LET Top empirical parameter Tg for the gas phase with the associated oil phase in the LET relative permeability equation.			1.0
		dimensionless	dimensionless	dimensionless	
6	KRTGAS	KRTGAS is a real positive number less than one, that defines the relative permeability of gas at the maximum gas saturation Krgt in the LET relative permeability equation.			1.0
		dimensionless	dimensionless	dimensionless	
7	SORG	SORG is a real positive number less than one that defines the residual oil saturation in and gas-oil system in the LET equations.			
		dimensionless	dimensionless	dimensionless	0.0
8	SOGCR	SOGCR is a real positive number less than one that defines critical oil-in-gas saturation, that is the largest oil saturation for which the oil relative permeability is zero in a gas-oil system.			0.0
		dimensionless	dimensionless	dimensionless	
9	LOIL	LOIL is a real positive number that defines the LET Lower empirical parameter Lo for the oil phase with the associated gas phase in the LET relative permeability equation.			1.0
		dimensionless	dimensionless	dimensionless	
10	EOIL	EOIL is a real positive number that defines the LET Elevation empirical parameter Eo for the oil phase with the associated gas phase in the LET relative permeability equation.			1.0
		dimensionless	dimensionless	dimensionless	
11	TOIL	TOIL is a real positive number that defines the LET Top empirical parameter To for the oil phase with the associated gas phase in the LET relative permeability equation.			1.0
		dimensionless	dimensionless	dimensionless	
12	KRTOIL	KRTOIL is a real positive number less than or equal to one, that defines the relative permeability of oil at the residual oil saturation, Krot in the LET relative permeability equation.			1.0
		dimensionless	dimensionless	dimensionless	
13	LPC	LPC is a real positive number that defines the gas-oil LET Lower empirical parameter L in the LET capillary pressure equation.			1.0
		dimensionless	dimensionless	dimensionless	
14	EPC	EPC is a real positive number that defines the gas-oil LET Elevation empirical parameter E in the LET capillary pressure equation.			1.0
		dimensionless	dimensionless	dimensionless	

No.	Name	Description			Default
		Field	Metric	Laboratory	
15	TPC	TPC is a real positive number that defines the gas-oil LET Top empirical parameter T in the LET capillary pressure equation.			1.0
		dimensionless	dimensionless	dimensionless	
16	PCIR	PCIR is a real positive number that defines the gas-oil capillary pressure at connate water saturation (SWL) in the LET capillary pressure equations.			
		psia	bars	atm	0.0
17	PCIT	PCIT is a real positive number that defines the gas-oil threshold capillary pressure at the maximum water saturation in the LET capillary pressure equations.			
		psia	bars	atm	0.0

Notes:

1) The keyword is followed by NTSFUN definitions as declared on the TABDIMS keyword in the RUNSPEC section, with each definition consisting of the above 17 LET parameters.
2) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.152: SGOFLET Keyword Description

Note there a two versions of the LET functions, LET[236] for two-phase flowing conditions and LETx[240] for three-phase flowing conditions. This keyword implements the LET version for a gas-oil system.

The functions are dependent on the drainage and imbibition cycle of the wetting phase as well as drainage and inhibition cycle number, since a reservoir may undergo several flooding events. To account for this the system defines the flooding event using the three saturations: Sw, So, and Sg together with the state of the three saturations during the flooding event. The saturation state can be Increasing, Decreasing, or Constant, for a given flooding event cycle number (n). Thus, Sw(D), So(I), Sg(C)1or DIC1, means the water phase is decreasing, the oil phase is increasing and the gas phase is constant for the primary or first cycle (n equals one). This is case for when oil is migrating into the reservoir rock and displacing the initial water contained with the reservoir.

Note

All the LET parameters are dependent on the flooding event and flooding cycle, and thus are expected vary as such. To be clear, the values of SGCR, Lg, Lo etc. should be different for each flooding cycle.

See also the *SGWFLET – Gas-Water LET Relative Permeability Functions*, and *SWOFLET – Water-Oil LET Relative Permeability Functions* keywords in this section.

[240] *Lomeland F. and Ebeltoft E., 2013. Versatile Three-phase Correlations for Relative Permeability and Capillary Pressure. Paper SCA2013-034 presented at the International Symposium of the Society of Core Analysts held in Napa Valley, California, USA, 16-19 September, 2013.*

Example

The following example uses the SGOFLET keyword to define two relative gas-oil relative permeability tables, based on NTSFUN equals two on the TABDIMS keyword in the RUNSPEC section.

```
--
--       SGOFLET – GAS-OIL LET REL. PERMEABILITY FUNCTIONS (OPM FLOW KEYWORD)
--
SGOFLET
--       SGL       SGCR     L-GAS     E-GAS     T-GAS     KRT-GAS
--       SOR       SOGCR    L-OIL     E-OIL     T-OIL     KRT-OIL
--       L-PC      E-PC     T-PC      PCIR      PCIT
--       -------   ------   -------   -------   -------   -------
         0.00000   0.0000   1.00000   1.00000   1.00000   1.00000
         0.00000   0.0000   1.00000   1.00000   1.00000   1.00000
         1*        1*       1*        1*        1*                / TABLE NO. 01

         0.00000   0.0000   1.00000   1.00000   1.00000   1.00000
         0.00000   0.0000   1.00000   1.00000   1.00000   1.00000
         1*        1*       1*        1*        1*                / TABLE NO. 02
```

Here the SGOFLET keyword parameters are all set to their default values.

8.3.279 SGU – END-POINT SCALING GRID CELL GAS SATURATION

Description

SGU defines the maximum gas saturation for all the cells in the model via an array when the end-point scaling option has been invoked via the ENDSCALE keyword in the RUNSPEC section. The maximum gas saturation is defined as the maximum gas saturation in a two-phase gas relative permeability table.

The keyword can be used with all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SGU	SGU is an array of real numbers assigning the maximum gas saturation values to each cell in the model. The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword. Repeat counts may be used, for example 30*0.70			Taken from cell allocated relative permeability table.
		dimensionless	dimensionless	dimensionless	

Notes:

1) Note this the directional independent version of the maximum gas saturation array used with the end-point scaling option. If directional end-point scaling has been activated then the SGUX± , SGUX± and SGU± series of keyword should be used.
2) If the value for a cell has been defaulted then OPM Flow uses the value from the cell's relative permeability table.
3) The keyword is terminated by a "/".

Table 8.153: SGU Keyword Description

End-point scaling allows the entered relative permeability functions to be re-scaled based on the saturation end-points allocated to each cell by the SWL, SWCR, SWU, SGL, SGCR, SGU, SOWCR, and SOGCR saturation grid arrays for the saturation end-points, and the KRG, KROG, KROW and KRW relative permeability grid cell arrays for the relative permeability end-point data. In addition end-point scaling may be directional dependent in which case the directional dependent versions of the aforementioned arrays should be used, that is SGUX, SGUY and SGUZ instead of SGU. There is also the facility to make the directional end-point scaling reversible or non-reversible and if the non-reversible option is selected the non-reversible versions of the aforementioned arrays should be used, that is SGUX, SGUX-, SGUY, SGUY-, SGUZ and SGUZ-, instead of the SGU keyword.

Example

```
--
--       DEFINE GRID BLOCK END-POINT SGU DATA FOR ALL CELLS
--       (FOR NX x NY x NZ = 300)
--
SGU
         300*0.700                                                    /
```

The above example defines a constant connate gas saturation of 0.70 to all 300 cells in the model as defined by the DIMENS keyword in the RUNSPEC section.

8.3.280 SGWFLET – GAS-WATER LET RELATIVE PERMEABILITY FUNCTIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

SWGFLET defines the relative permeability and capillary pressure parameters for the water-gas LET family of models. Both the gas and water phases should be made active in the model via the GAS and WATER keywords in the RUNSPEC section. This keyword should only be used in two-phase models containing the gas and water phases only, that is the oil phase must be absent. See section *8.2.6Saturation Table Generation - LET Functions* and Lomeland et al.[241],[242] [243] and [244] for further information on the model.

The keyword is used as a replacement for the SGWFN keyword for two-phase gas-water systems, and the LET series of keywords cannot be combined with the standard set of relative permeability keywords.

Note

This is an OPM Flow specific keyword and will therefore cause an error in the commercial simulator.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SGL	SGL is a real positive number less than one that defines the connate gas saturation, that is smallest gas saturation in the LET function.			0.0
		dimensionless	dimensionless	dimensionless	
2	SGCR	SGCR is a real positive number greater than or equal to SGL and less than one, that defines the critical gas saturation, that is the largest gas saturation for which the gas relative permeability is zero.			0.0
		dimensionless	dimensionless	dimensionless	
3	LGAS	LGAS is a real positive number that defines the LET Lower empirical parameter Lg for the gas phase with the associated water phase in the LET relative permeability equations.			1.0
		dimensionless	dimensionless	dimensionless	
4	EGAS	EGAS is a real positive number that defines the LET Elevation empirical parameter Eg for the gas phase with the associated water phase in the LET relative permeability equations.			1.0
		dimensionless	dimensionless	dimensionless	

[241] *Lomeland F., Ebeltoft E. and Thomas W.H., 2005. A New Versatile Relative Permeability Correlation. Paper SCA2005-32 presented at the International Symposium of the Society of Core Analysts held in Toronto, Canada, 21-25 August, 2005.*

[242] *Lomeland F. and Ebeltoft E., 2008. A New Versatile Capillary Pressure Correlation. Paper SCA2008-08 presented at the International Symposium of the Society of Core Analysts held in Abu Dhabi, UAE, 29 Oct. – 2 Nov., 2008.*

[243] *Lomeland F., Hasanov B., Ebeltoft E. and Berge M., 2012. A Versatile Representation of Up-scaled Relative Permeability for Field Applications. Paper SPE 154487-MS presented at the EAGE Annual Conference & Exhibition incorporating SPE Europec held in Copenhagen, Denmark, 4-7 June 2012.*

[244] *Lomeland F., 2018.. Overview Of The Let Family Of Versatile Correlations For Flow Functions. Paper SCA2018-056 presented at the International Symposium of the Society of Core Analysts held in Trondheim, Norway, 27-30 August 2018.*

No.	Name	Description			Default
		Field	Metric	Laboratory	
5	TGAS	TGAS is a real positive number that defines the LET Top empirical parameter Tg for the gas phase with the associated water phase in the LET relative permeability equations.			1.0
		dimensionless	dimensionless	dimensionless	
6	KRTGAS	KRTGAS is a real positive number less than one, that defines the relative permeability of gas at the maximum gas saturation Krgt in the LET relative permeability equations.			1.0
		dimensionless	dimensionless	dimensionless	
7	SWL	SWL is a real positive number less than one, that defines the connate water saturation, that is the smallest water saturation in the LET function.			0.0
		dimensionless	dimensionless	dimensionless	
8	SWCR	SWCR is a real positive number greater than or equal to SWL and less than one, that defines the critical water saturation, that is the largest water saturation for which the water relative permeability is zero, Swirr in equations.			0.0
		dimensionless	dimensionless	dimensionless	
9	LWAT	LWAT is a real positive number that defines the LET Lower empirical parameter Lw for the water phase with the associated gas phase in the LET relative permeability equations.			1.0
		dimensionless	dimensionless	dimensionless	
10	EWAT	EWAT is a real positive number that defines the LET Elevation empirical parameter Ew for the water phase with the associated gas phase in the LET relative permeability equations.			1.0
		dimensionless	dimensionless	dimensionless	
11	TWAT	TWAT is a real positive number that defines the LET Top empirical parameter Tw for the water phase with the associated gas phase in the LET relative permeability equations.			1.0
		dimensionless	dimensionless	dimensionless	
12	KRTWAT	KRTWAT is a real positive number less than one, that defines the relative permeability of water at the maximum water saturation (normally the maximum water saturation is one) Krwt in the LET relative permeability equations.			1.0
		dimensionless	dimensionless	dimensionless	
13	LPC	LPC is a real positive number that defines the gas-water LET Lower empirical parameter L in the LET capillary pressure equation.			1.0
		dimensionless	dimensionless	dimensionless	

No.	Name	Description			Default
		Field	Metric	Laboratory	
14	EPC	EPC is a real positive number that defines the gas-water LET Elevation empirical parameter E in the LET capillary pressure equation.			1.0
		dimensionless	dimensionless	dimensionless	
15	TPC	TPC is a real positive number that defines the gas-water LET empirical parameter T in the LET capillary pressure equation.			1.0
		dimensionless	dimensionless	dimensionless	
16	PCIR	PCIR is a real positive number that defines the gas-water capillary pressure at connate water saturation (SWL) in the LET capillary pressure equations.			0.0
		psia	bars	atm	
17	PCIT	PCIT is a real positive number that defines the gas-water threshold capillary pressure at the maximum water saturation in the LET capillary pressure equations.			0.0
		psia	bars	atm	

Notes:

1) The keyword is followed by NTSFUN definitions as declared on the TABDIMS keyword in the RUNSPEC section, with each definition consisting of the above 17 LET parameters.
1) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.154: SGWFLET Keyword Description

Note there a two versions of the LET functions, LET[241] for two-phase flowing conditions and LETx[245] for three-phase flowing conditions. This keyword implements the LET version for a gas-water system.

The functions are dependent on the drainage and imbibition cycle of the wetting phase as well as drainage and inhibition cycle number, since a reservoir may undergo several flooding events. To account for this the system defines the flooding event using the three saturations: Sw, So, and Sg together with the state of the three saturations during the flooding event. The saturation state can be Increasing, Decreasing, or Constant, for a given flooding event cycle number (n). Thus, Sw(D), So(C), Sg(I), or DCI1, means the water phase is decreasing, the oil phase is constant and the gas phase is increasing for the primary or first cycle (n equals one). This is case for when gas is migrating into the reservoir rock and displacing the initial water contained with the reservoir.

Note

All the LET parameters are dependent on the flooding event and flooding cycle, and thus are expected vary as such. To be clear, the values of SWCR, Lg, Lw etc. should be different for each flooding cycle.

See also the *SGOFLET– Gas-Oil LET Relative Permeability Functions*, and *SWOFLET – Water-Oil LET Relative Permeability Functions* keywords in this section, that may be used for three-phase systems.

[245] *Lomeland F. and Ebeltoft E., 2013. Versatile Three-phase Correlations for Relative Permeability and Capillary Pressure. Paper SCA2013-034 presented at the International Symposium of the Society of Core Analysts held in Napa Valley, California, USA, 16-19 September, 2013.*

Example

The following example uses the SWGFLET keyword to define two relative gas-water relative permeability tables, based on NTSFUN equals two on the TABDIMS keyword in the RUNSPEC section.

```
--
--       SGWFLET – GAS-WATER LET REL. PERMEABILITY FUNCTIONS (OPM FLOW KEYWORD)
--
SGWFLET
--       SGL       SGCR     L-GAS     E-GAS     T-GAS     KRT-GAS
--       SWL       SWCR     L-WAT     E-WAT     T-WAT     KRT-WAT
--       L-PC      E-PC     T-PC      PCIR      PCIT
--       -------   ------   -------   -------   -------   -------
         0.00000   0.0000   1.00000   1.00000   1.00000   1.00000
         0.00000   0.0000   1.00000   1.00000   1.00000   1.00000
         1*        1*       1*        1*        1*                  / TABLE NO. 01

         0.00000   0.0000   1.00000   1.00000   1.00000   1.00000
         0.00000   0.0000   1.00000   1.00000   1.00000   1.00000
         1*        1*       1*        1*        1*                  / TABLE NO. 02
```

Here the SGWFLET keyword parameters are all set to their default values.

8.3.281 SGWFN – GAS-WATER SATURATION TABLES (FORMAT TYPE 2)

Description

The SGWFN keyword defines the gas and water relative permeability and gas-water capillary pressure data versus gas saturation tables for when gas and water are present in the input deck. This keyword should only be used if gas and water are present in the run.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SGAS	A columnar vector of real monotonically increasing down the column values starting from zero and terminating at one, that defines the gas saturation.			
		dimensionless	dimensionless	dimensionless	None
2	KRG	A columnar vector of real values that are either equal or increasing down the column and that are greater than or equal to zero and less than or equal to one that defines the gas relative permeability. Note that the first entry in the column must be zero.			
		dimensionless	dimensionless	dimensionless	None
3	KRW	A columnar vector of real values that are either equal or decreasing down the column and that are greater than or equal to zero and less than or equal to one that defines the water relative permeability with respect to gas saturation. The last value in the column should be zero.			
		dimensionless	dimensionless	dimensionless	None
4	PCGW	A columnar vector of real values that are either equal or increasing down the column that defines the gas-water relative capillary pressure.			
		psia	bars	atm	None

Notes:

1) The keyword is followed by NTSFUN tables as declared on the TABDIMS keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NSSFUN rows as declared on the TABDIMS keyword in the RUNSPEC section.
3) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.155: SGWFN Keyword Description

Example

```
--
--       GAS-WATER RELATIVE PERMEABILITY TABLES (SGWFN)
SGWFN
--       SG          KRG       KRW       PCOW
--       FRAC                            PSIA
--       --------    --------  -------   -------
         0.000000    0.0000    0.9000    0.000000
         0.200000    0.0002    0.7664    0.000000
         0.699099    0.4973    0.0000    0.000000
         0.700000    1.0000    0.0000    0.000000          / TABLE NO. 01
--       --------    --------  -------   -------
         0.000000    0.0000    0.9000    0.000000
         0.200000    0.0002    0.7664    0.000000
         0.245309    0.0004    0.7443    0.000000
         0.261989    0.0010    0.6907    0.000000
         0.303091    0.0044    0.5671    0.000000
         0.368269    0.0191    0.3962    0.000000
         0.435026    0.0519    0.2528    0.000000
         0.486387    0.0940    0.1643    0.000000
         0.522283    0.1339    0.1137    0.000000
         0.550683    0.1725    0.0803    0.000000
         0.575342    0.2115    0.0559    0.000000
         0.599076    0.2542    0.0367    0.000000
         0.621294    0.2991    0.0223    0.000000
         0.642171    0.3458    0.0120    0.000000
         0.658984    0.3868    0.0061    0.000000
         0.671123    0.4183    0.0030    0.000000
         0.679268    0.4403    0.0015    0.000000
         0.684963    0.4562    0.0008    0.000000
         0.688893    0.4674    0.0004    0.000000
         0.692025    0.4765    0.0002    0.000000
         0.694641    0.4841    0.0001    0.000000
         0.696976    0.4910    0.0000    0.000000
         0.699099    0.4973    0.0000    0.000000
         0.700000    1.0000    0.0000    0.000000          / TABLE NO. 02
```

The example defines two SGWFN tables for use when oil, gas and water are present in the run.

8.3.282 SHRATE - ACTIVATE AND DEFINE THE POLYMER SHEARING LOGARITHMIC PARAMETERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword activates the logarithm-based polymer shear thinning/thickening option and defines the shear rate constant. This keyword can only be used in conjunction with the PLYSHLOG in the PROPS section

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

See the PLYSHEAR keyword for the alternative polymer shear thinning/thickening option that is implemented in OPM Flow.

8.3.283 SKIP – ACTIVATE SKIPPING OF ALL KEYWORDS AND INPUT DATA

The SKIP keyword turns on skipping of keywords until the ENDSKIP activation keyword is encountered. All keywords and between the SKIP and ENDSKIP keywords are ignored by OPM Flow.

See *SKIP – Activate Skipping of All Keywords and Input Data* in the GLOBAL section for a full description.

8.3.284 SKIP100 – ACTIVATE SKIPPING OF BLACK-OIL KEYWORDS AND INPUT DATA

This keyword turns on skipping of black-oil keywords until the ENDSKIP activation keyword is encountered. All black-oil keywords between the SKIP100 and ENDSKIP keywords are ignored by OPM Flow.

See *SKIP100 – Activate Skipping of Black-Oil Keywords and Input Data* in the GLOBAL section for a full description.

8.3.285 SKIP300 – ACTIVATE SKIPPING OF "COMPOSITIONAL" KEYWORDS AND INPUT DATA

Turns on skipping of "compositional" keywords until the ENDSKIP activation keyword is encountered. All "compositional" keywords between the SKIP300 and ENDSKIP keywords are ignored by OPM Flow.

See *Error: Reference source not found* in the GLOBAL section for a full description.

8.3.286 SKPRPOLY – POLYMER MOLECULAR WEIGHT MODEL POLYMER INJECTION SKIN TABLE

Description

This keyword, SKPRPOLY, describes the relationship of a water injection well's injected polymer skin pressure as a function of polymer throughput and water velocity, for the simulator's Polymer Molecular Weight Transport option. The table is a two dimensional table that relates the polymer throughput values and water velocity values to derive the resulting wellbore skin pressure of the injected polymer, which is then used to calculate the total wellbore skin pressure based on the polymer concentration.

This keyword should only be used if the POLYMER and POLYMW keywords in the RUNSPEC section are also activated.

Note

This is an OPM Flow specific keyword that employs an alternative polymer flood model based on a Polymer Molecular Weight Transport equation, that is not available in the commercial simulator.

The model has been tested using metric units; however, using either field or laboratory units with the option should be considered experimental.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1-1	SKPRPNUM	A positive integer value greater than zero and less than or equal to the NTSKPOLY variable, as defined on the PINTDIMS keyword in the RUNSPEC section, that defines the SKPRPOLY Polymer Molecular Weight Model polymer injection skin pressure table number.			None
1-2	POLCON	A real positive value that the defines the reference polymer concentration for the table.			None
		lb/stb	kg/sm^3	gm/scc	
2-1	THRUPUT	A real positive monotonically increasing vector, that defines the polymer throughput values. The first entry should be zero to define a no throughput data set, and each vector record should be on a separate line terminated by a "/".			None
		$feet^3/feet^2$	m^3/m^2	cm^3/cm^2	
3-1	VELOCITY	A real positive monotonically increasing vector, that defines the polymer velocity values. The first entry should be zero to define a no velocity data set, and each vector record should be on a separate line terminated by a "/".			None
		feet/day	m/day	cm/hour	

No.	Name	Description			Default
		Field	Metric	Laboratory	
4-1	PRESS	A series of real positive vectors representing the wellbore skin pressure, for the given reference polymer concentration (POLCON), for all combinations of the polymer throughput values (THRUPUT) and velocity values (VELOCITY), organized as a series of vectors PRESS(THRUPUT, VELOCITY). Thus, the first vector represents the wellbore skin pressure of the first THRUPUT value and each entry in the vector is the corresponding wellbore skin pressure of the associated VELOCITY vector. Each vector should be on a separate line and should be terminated by a "/". Thus, if THRUPUT has three entries and VELOCITY has four, then there should be three vectors, with each vector containing four elements representing wellbore skin pressure values, as a function of THRUPUT and VELOCITY.			None
		psia	barsa	atma	

Notes:

1) The keyword should be terminated by a "/", but may be repeated up to NTSKPOLY times, as per the PINTDIMS keyword in the RUNSPEC section, to allow for the input of multiple tables.

Table 8.156: SKPRPOLY Keyword Description

Unlike other PROPS section table keywords, that enable multiple tables following the keyword to be entered, the SKPRPOLY keyword requires that the keyword itself must be repeated for each table, as is shown in the example on the following page.

The WSKPTAB keyword in the SCHEDULE section may be used to assign the SKPRPOLY and SKPRWAT tables to water injections wells, that enable the calculation of the wellbore skin pressure based on the fluids being injected.

See also the SKPRWAT, PLYMWINJ, and PLYVMH keywords, in the PROPS section, that are the additional property keywords required for the Polymer Molecular Weight Transport option. In addition, see also the WPMITAB keyword in the SCHEDULE section, that assigns the PLYMWINJ tables to the water injection wells.

Note that the standard polymer property data keywords: PLYROCK, PLYADS, PLYMAX, etc., are still required to fully describe the polymer fluid.

Example

Given NTSKPOLY equals two on the PINTDIMS keyword in the RUNSPEC section, then two SKPRPOLY tables are required to be entered:

```
--
--       POLYMER MOLECULAR WEIGHT MODEL POLYMER INJECTION SKIN TABLE
--       (OPM FLOW PROPS KEYWORD)
--
SKPRPOLY
--       TABLE   POLYMER REF
--       NO.     CONCENTRATION
         1       2.00                  /
--
--       THROUGHPUT VALUES
--
         0.0     200.0   400.0         /
--
--       VELOCITY VALUES
--
         0.0     50.0    80.0    100.0 /
--
--       PRESS SKIN VALUES
--
         0.0     10.0    20.0    40.0  / PRESS(THRUPUT=1, VELOCITY=1 TO N)
         0.0     50.0    100.0   200.0 / PRESS(THRUPUT=2, VELOCITY=1 TO N)
         0.0     100.0   200.0   400.0 / PRESS(THRUPUT=3, VELOCITY=1 TO N)
/
SKPRPOLY
--       TABLE   POLYMER REF
--       NO.     CONCENTRATION
         2       2.0                   /
--
--       THROUGHPUT VALUES
--
         0.0     200.0   400.0         /
--
--       VELOCITY VALUES
--
         0.0     30.0    50.0    100.0 /
--
--       PRESS SKIN VALUES
--
         0.0     10.0    20.0    40.0  / PRESS(THRUPUT=1, VELOCITY=1 TO N)
         0.0     50.0    100.0   200.0 / PRESS(THRUPUT=2, VELOCITY=1 TO N)
         0.0     100.0   200.0   400.0 / PRESS(THRUPUT=3, VELOCITY=1 TO N)
/
```

As mentioned previously, the SKPRPOLY keyword requires that the keyword itself must be repeated for each table, as is shown in the above example.

8.3.287 SKPRWAT – POLYMER MOLECULAR WEIGHT MODEL WATER INJECTION SKIN TABLE

Description

This keyword, SKPRWAT, describes the relationship of a water injection well's injected water skin pressure as a function of water throughput and water velocity, for the simulator's Polymer Molecular Weight Transport option. The table is a two dimensional table that relates the water throughput values and water velocity values to derive the resulting wellbore skin pressure of the injected water, which is then used to calculate the total wellbore skin pressure based on the polymer concentration.

This keyword should only be used if the POLYMER and POLYMW keywords in the RUNSPEC section are also activated.

Note

This is an OPM Flow specific keyword that employs an alternative polymer flood model based on a Polymer Molecular Weight Transport equation, that is not available in the commercial simulator.

The model has been tested using metric units; however, using either field or laboratory units with the option should be considered experimental.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1-1	SKPRWNUM	A positive integer value greater than zero and less than or equal to the NTSKWAT variable, as defined on the PINTDIMS keyword in the RUNSPEC section, that defines the SKPRWAT Polymer Molecular Weight Model water injection skin pressure table number.			None
2-1	THRUPUT	A real positive monotonically increasing vector, that defines the water throughput values. The first entry should be zero to define a no throughput data set, and each vector record should be on a separate line terminated by a "/".			None
		$feet^3/feet^2$	m^3/m^2	cm^3/cm^2	
3-1	VELOCITY	A real positive monotonically increasing vector, that defines the water velocity values. The first entry should be zero to define a no velocity data set, and each vector record should be on a separate line terminated by a "/".			None
		feet/day	m/day	cm/hour	

No.	Name	Description			Default
		Field	Metric	Laboratory	
4-1	PRESS	A series of real positive vectors representing the wellbore skin pressure, for all combinations of the water throughput values (THRUPUT) and velocity values (VELOCITY), organized as a series of vectors PRESS(THRUPUT,VELOCITY). Thus, the first vector represents the wellbore skin pressure of the first THRUPUT value and each entry in the vector is the corresponding wellbore skin pressure of the associated VELOCITY vector. Each vector should be on a separate line and should be terminated by a "/". Thus, if THRUPUT has three entries and VELOCITY has four, then there should be three vectors, with each vector containing four elements representing wellbore skin pressure values, as a function of THRUPUT and VELOCITY.			None
		psia	barsa	atma	
Notes: 1) The keyword should be terminated by a "/", but may be repeated up to NTSKWAT times, as per the PINTDIMS keyword in the RUNSPEC section, to allow for the input of multiple tables.					

Table 8.157: SKPRWAT Keyword Description

Unlike other PROPS section table keywords, that enable multiple tables following the keyword to be entered, the SKPRWAT keyword requires that the keyword itself must be repeated for each table, as is shown in the example on the following page.

The WSKPTAB keyword in the SCHEDULE section may be used to assign the SKPRPOLY and SKPRWAT tables to water injections wells, that enable the calculation of the wellbore skin pressure based on the fluids being injected.

See also the SKPRPOLY, PLYMWINJ, and PLYVMH keywords, in the PROPS section, that are the additional property keywords required for the Polymer Molecular Weight Transport option. In addition, see also the WPMITAB keyword in the SCHEDULE section, that assigns the PLYMWINJ tables to the water injection wells.

Note that the standard polymer property data keywords: PLYROCK, PLYADS, PLYMAX, etc., are still required to fully describe the polymer fluid.

Example

Given NTSKWAT equals two on the PINTDIMS keyword in the RUNSPEC section, then two SKPRWAT tables are required to be entered:

```
--
--        POLYMER MOLECULAR WEIGHT MODEL WATER INJECTION SKIN TABLE
--        (OPM FLOW PROPS KEYWORD)
--
SKPRWAT
          1                              / TABLE NUMBER
--
--        THROUGHPUT VALUES
--
          0.0     200.0   400.0          /
--
--        VELOCITY VALUES
--
          0.0     50.0    70.0    100.0 /
--
--        PRESS SKIN VALUES
--
          0.0     2.0     4.0     8.0   / PRESS(THRUPUT=1, VELOCITY=1 TO N)
          0.0     20.0    40.0    80.0  / PRESS(THRUPUT=2, VELOCITY=1 TO N)
          0.0     50.0    100.0   200.0 / PRESS(THRUPUT=3, VELOCITY=1 TO N)
/
SKPRWAT
          2                              / TABLE NUMBER
--
--        THROUGHPUT VALUES
--
          0.0     200.0   400.0          /
--
--        VELOCITY VALUES
--
          0.0     30.0    50.0    100.0 /
--
--        PRESS SKIN VALUES
--
          0.0     2.0     4.0     8.0   / PRESS(THRUPUT=1, VELOCITY=1 TO N)
          0.0     20.0    40.0    80.0  / PRESS(THRUPUT=2, VELOCITY=1 TO N)
          0.0     50.0    100.0   200.0 / PRESS(THRUPUT=3, VELOCITY=1 TO N)
/
```

As mentioned previously, the SKPRWAT keyword requires that the keyword itself must be repeated for each table, as is shown in the above example.

8.3.288 SKRO – END-POINT SCALING OF GRID CELL KRO(SWL) (SURFACTANT)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

SKRO defines the scaling parameter for the surfactant oil relative permeability value at the connate water saturation (SWL), for all the cells in the model via an array. The ENDSCALE keyword in the RUNSPEC section should be activated to enable end-point scaling and the use of this keyword. In addition, the Surfactant option must be enabled by either the SURFST or SURFSTES keywords in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SKRO	SKRO is an array of positive real numbers which are greater than zero and less than or equal to 1.0, that are the assigned scaling SKRO values for each cell in the model. Repeat counts may be used, for example 50*0.500.			Taken from cell allocated relative permeability table.
		dimensionless	dimensionless	dimensionless	

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) The keyword is terminated by a "/".

Table 8.158: SKRO Keyword Description

End-point scaling allows the entered surfactant relative permeability functions to be scale on the relative permeability values using the SKRO, SKRORG, SKRORW, SKRW and SKRWR surfactant relative permeability grid cell arrays for the relative permeability end-point data.

Example

The example defines an input box for the whole grid and for layers one to three, for layer one SKRO is set equal to 0.850, for layer two SKRO equals 0.875, and for layer three SKRO equals 0.900.

```
--
--       DEFINE INPUT BOX FOR EDITING INPUT ARRAYS (NX=100, NY=100)
--
--       ---------- BOX ---------
--       I1  I2   J1  J2   K1  K2
BOX
         1*  1*   1*  1*   1   3                          / DEFINE BOX AREA
--
--       SET SKRO VALUES FOR THREE LAYERS IN THE MODEL
--
SKRO
         1000*0.855  1000*0.875  1000.0.900               /
--
--       DEFINE END OF INPUT BOX EDITING OF INPUT ARRAYS
--
ENDBOX
```

8.3.289 SKRORG – END-POINT SCALING OF GRID CELL KRO(SGCR) (SURFACTANT)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

SKRORG defines the scaling parameter for the surfactant relative permeability of oil at the critical gas saturation (SGCR), for all the cells in the model via an array. The ENDSCALE keyword in the RUNSPEC section should be activated to enable end-point scaling and the use of this keyword. In addition, the Surfactant option must be enabled by either the SURFST or SURFSTES keywords in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SKRORG	SKRORG is an array of positive real numbers which are greater than zero and less than or equal to 1.0, that are the assigned scaling SKRORG values for each cell in the model. Repeat counts may be used, for example 50*0.850.			Taken from cell allocated relative permeability table.
		dimensionless	dimensionless	dimensionless	

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.

2) The keyword is terminated by a "/".

Table 8.159: SKRORG Keyword Description

End-point scaling allows the entered surfactant relative permeability functions to be scale on the relative permeability values using the SKRO, SKRORG, SKRORW, SKRW and SKRWR surfactant relative permeability grid cell arrays for the relative permeability end-point data.

Example

The example uses the EQUALS keyword to set layer one SKRORG equal to 0.750, layer two SKRORG equals 0.775, and layer three SKRORG equals 0.800.

```
--
--        ARRAY        CONSTANT      ---------- BOX ---------
--                                    I1   I2    J1   J2    K1   K2
EQUALS
          SKRORG       0.7550        1*   1*    1*   1*     1    1  / SKRORG FOR LAYER 1
          SKRORG       0.7750        1*   1*    1*   1*     2    2  / SKRORG FOR LAYER 2
          SKRORG       0.8000        1*   1*    1*   1*     3    3  / SKRORG FOR LAYER 3
/
```

8.3.290 SKRORW – END-POINT SCALING OF GRID CELL KRO(SWCR) (SURFACTANT)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

SKRORW defines the scaling parameter for the surfactant relative permeability of oil at the critical water saturation (SWCR), for all the cells in the model via an array. The ENDSCALE keyword in the RUNSPEC section should be activated to enable end-point scaling and the use of this keyword. In addition, the Surfactant option must be enabled by either the SURFST or SURFSTES keywords in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SKRORW	SKRORW is an array of positive real numbers which are greater than zero and less than or equal to 1.0, that are the assigned scaling SKRORW values for each cell in the model. Repeat counts may be used, for example 50*0.850			Taken from cell allocated relative permeability table.
		dimensionless	dimensionless	dimensionless	

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.

2) The keyword is terminated by a "/".

Table 8.160: SKRORW Keyword Description

End-point scaling allows the entered surfactant relative permeability functions to be scale on the relative permeability values using the SKRO, SKRORG, SKRORW, SKRW and SKRWR surfactant relative permeability grid cell arrays for the relative permeability end-point data.

Example

The example defines an input box for the whole grid and for layers one to three, for layer one SKRORW is set equal to 0.750, for layer two SKRORW equals 0.775, and for layer three SKRORW equals 0.800.

```
--
--       DEFINE INPUT BOX FOR EDITING INPUT ARRAYS (NX=100, NY=100)
--
--       ---------- BOX ---------
--       I1  I2   J1  J2   K1  K2
BOX
         1*  1*   1*  1*   1   3                          / DEFINE BOX AREA
--
--       SET SKRORW VALUES FOR THREE LAYERS IN THE MODEL
--
SKRORW
         1000*0.755  1000*0.775  1000.0.800               /
--
--       DEFINE END OF INPUT BOX EDITING OF INPUT ARRAYS
--
ENDBOX
```

8.3.291 SKRW – END-POINT SCALING OF GRID CELL KRW(SW =1.0) (SURFACTANT)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

SKRW defines the scaling parameter at the maximum surfactant water relative permeability value (SWU), that is for Sw = 1.0, for all the cells in the model via an array. The ENDSCALE keyword in the RUNSPEC section should be activated to enable end-point scaling and the use of this keyword. In addition, the Surfactant option must be enabled by either the SURFST or SURFSTES keywords in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SKRW	SKRW is an array of positive real numbers which are greater than zero and less than or equal to 1.0, that are the assigned scaling SKRW values for each cell in the model. Repeat counts may be used, for example 50*1.000.			Taken from cell allocated relative permeability table.
		dimensionless	dimensionless	dimensionless	

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.

2) The keyword is terminated by a "/".

Table 8.161: SKRW Keyword Description

End-point scaling allows the entered surfactant relative permeability functions to be scale on the relative permeability values using the SKRO, SKRORG, SKRORW, SKRW and SKRWR surfactant relative permeability grid cell arrays for the relative permeability end-point data.

Example

The example uses the EQUALS keyword to set SKRW for layer one equal to 0.850, layer two SKRW to 0.875, and layer three KRW to 0.900.

```
--
--        ARRAY        CONSTANT      ---------- BOX ---------
--                                   I1  I2   J1  J2   K1  K2
EQUALS
          SKRW         0.8550        1*  1*   1*  1*    1   1  / SKRW FOR LAYER 1
          SKRW         0.8750        1*  1*   1*  1*    2   2  / SKRW FOR LAYER 2
          SKRW         0.9000        1*  1*   1*  1*    3   3  / SKRW FOR LAYER 3
/
```

8.3.292 SKRWR – END-POINT SCALING OF GRID CELL KRWR(SOWCR) (SURFACTANT)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

SKRWR defines the scaling parameter at the critical oil to water saturation value (SOWCR), for the surfactant water relative permeability curve, for all the cells in the model via an array. The ENDSCALE keyword in the RUNSPEC section should be activated to enable end-point scaling and the use of this keyword.In addition, the Surfactant option must be enabled by either the SURFST or SURFSTES keywords in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SKRWR	SKRWR is an array of positive real numbers which are greater than zero and less than or equal to 1.0, that are the assigned scaling SKRWR values for each cell in the model. Repeat counts may be used, for example 50*1.000.			Taken from cell allocated relative permeability table.
		dimensionless	dimensionless	dimensionless	

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.

2) The keyword is terminated by a "/".

Table 8.162: SKRWR Keyword Description

End-point scaling allows the entered surfactant relative permeability functions to be scale on the relative permeability values using the SKRO, SKRORG, SKRORW, SKRW and SKRWR surfactant relative permeability grid cell arrays for the relative permeability end-point data.

Example

The first example defines an input box for the whole grid and for layers one to three, for layer one SKRWR is set equal to 0.750, for layer two SKRWR equals 0.775, and for layer three SKRWR equals 0.800.

```
--
--       DEFINE INPUT BOX FOR EDITING INPUT ARRAYS (NX=100, NY=100)
--
--       ---------- BOX ---------
--       I1  I2   J1  J2   K1  K2
BOX
         1*  1*   1*  1*   1   3                          / DEFINE BOX AREA
--
--       SET SKRWR VALUES FOR THREE LAYERS IN THE MODEL
--
SKRWR
         1000*0.755  1000*0.775  1000.0.800               /
--
--       DEFINE END OF INPUT BOX EDITING OF INPUT ARRAYS
--
ENDBOX
```

8.3.293 SLGOF – GAS-OIL SATURATION TABLES VERSUS GAS (FORMAT TYPE 1)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The SLGOF keyword defines the oil and gas relative permeability and oil-gas capillary pressure versus liquid saturation tables for when oil and gas are present in the input deck. This keyword should only be used if both oil and gas are present in the run.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SLIQ	A columnar vector of real monotonically increasing down the column values starting from zero and terminating at one, that defines the liquid saturation, that is the connate water saturation (SWL) plus the oil saturation. The first entry should correspond to residual liquid, that is Swc + Sorg and the last entry should be 1.0 to correspond to a gas saturation of zero.			
		dimensionless	dimensionless	dimensionless	None
2	KRG	A columnar vector of real values that are either equal or decreasing down the column and that are greater than or equal to zero and less than or equal to one that defines the gas relative permeability..			
		dimensionless	dimensionless	dimensionless	None
3	KRO	A columnar vector of real values that are either equal or increasing down the column and that are greater than or equal to zero and less than or equal to one that defines the oil relative permeability with respect to gas and connate water saturation. When water is active in the run, the last entry the column, that is at krog(Sg = 0), must be the same as the first entry in the corresponding SWOF table, that is at krow(So = 1 - Swco). The first value in the column should be zero.			
		dimensionless	dimensionless	dimensionless	None
4	PCOG	A columnar vector of real values that are either equal or decreasing down the column that defines the oil-gas relative capillary pressure.			
		psia	bars	atm	None

Notes:

1) The keyword is followed by NTSFUN tables as declared on the TABDIMS keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NSSFUN rows as declared on the TABDIMS keyword in the RUNSPEC section.
3) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.163: SLGOF Keyword Description

Example

```
--
--       GAS-OIL RELATIVE PERMEABILITY TABLES (SLGOF)
SLGOF
--       SLIQ       KRG       KROG      PCOG
--       FRAC                           PSIA
--       -------   --------  -------   -------
         0.30060    0.55000  0.0000     0.0000
         0.31480    0.42500  0.2848     0.0000
         0.32910    0.35000  0.3091     0.0000
         0.34330    0.27500  0.4333     0.0000
         0.35760    0.25000  0.5576     0.0000
         0.37180    0.22500  0.5818     0.0000
         0.38610    0.20000  0.6061     0.0000
         0.40030    0.17500  0.6303     0.0000
         0.41450    0.15000  0.6545     0.0000
         0.42880    0.12500  0.6788     0.0000
         0.44300    0.10000  0.7030     0.0000
         0.45730    0.07500  0.7273     0.0000
         0.47150    0.05000  0.7515     0.0000
         0.48580    0.02500  0.7758     0.0000
         0.50000    0.00000  0.8000     0.0000
         0.80000    0.00000  0.9000     0.0000             / TABLE No. 01
--       -------   --------  -------   -------
         0.30060    0.55000  0.0000     0.0000
         0.31480    0.42500  0.2848     0.0000
         0.32910    0.35000  0.3091     0.0000
         0.34330    0.27500  0.4333     0.0000
         0.35760    0.25000  0.5576     0.0000
         0.37180    0.22500  0.5818     0.0000
         0.38610    0.20000  0.6061     0.0000
         0.40030    0.17500  0.6303     0.0000
         0.41450    0.15000  0.6545     0.0000
         0.42880    0.12500  0.6788     0.0000
         0.44300    0.10000  0.7030     0.0000
         0.45730    0.07500  0.7273     0.0000
         0.47150    0.05000  0.7515     0.0000
         0.48580    0.02500  0.7758     0.0000
         0.50000    0.00000  0.8000     0.0000
         0.80000    0.00000  0.9000     0.0000             / TABLE No. 02
```

The example defines two SLGOF tables for use when oil, gas and water are present in the run.

8.3.294 SOCRS – END-POINT SCALING GRID CELL MISCIBLE CRITICAL OIL SATURATION WITH RESPECT TO WATER

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

SOCRS defines the miscible critical oil saturation for all the cells in the model via an array when the end-point scaling option has been invoked via the ENDSCALE keyword in the RUNSPEC section. The critical oil saturation with respect to water is defined as the maximum oil saturation for which the oil relative permeability is zero in a two-phase oil-water relative permeability table. The keyword is used with the Surfactant model to re-scale the surfactant relative permeability saturation tables allocated to a grid block by the by the SURFNUM keyword in the REGIONS section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SOCRS	SOCRS is an array of real numbers assigning the critical oil saturation with respect to water values to each cell in the model. The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword. Repeat counts may be used, for example 30*0.30			Taken from cell allocated relative permeability table.
		dimensionless	dimensionless	dimensionless	

Notes:

1) Note this the directional independent version of the critical gas saturation used with the end-point scaling option. If directional end-point scaling has been activated then the SOCRSX± , SOCRSX± and SOCRSX± series of keyword should be used.
2) If the value for a cell has been defaulted then OPM Flow uses the value from the cell's relative permeability table.
3) The keyword is terminated by a "/".

Table 8.164: SOGCR Keyword Description

End-point scaling allows the entered relative permeability functions to be re-scaled based on the saturation end-points allocated to each cell by the SWL, SWCR, SWU, SGL, SGCR, SGU, SOCRS, and SOGCR saturation grid arrays for the saturation end-points, and the KRG, KROG, KROW and KRW relative permeability grid cell arrays for the relative permeability end-point data.

Example

```
--
--       DEFINE GRID BLOCK END-POINT SOCRS DATA FOR ALL CELLS
--       (FOR NX x NY x NZ = 300)
--
SOCRS
         300*0.200                                                    /
```

The above example defines a constant critical oil saturation of 0.20 to all 300 cells in the model as defined by the DIMENS keyword in the RUNSPEC section.

8.3.295 SOF2 – OIL SATURATION TABLES WITH RESPECT TO GAS OR WATER (FORMAT TYPE 2)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The SOF2 keyword defines the oil relative permeability versus oil saturation tables for when oil and gas or oil and water are present in the input deck. The keyword is also used to define the relative permeability of the miscible hydrocarbon phase in SOLVENT runs. This keyword should only be used if oil is present in the run.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SOIL	A columnar vector of real monotonically increasing down the column values starting from zero and terminating at one, that defines the oil or the hydrocarbon solvent saturation. For two phase runs the oil saturation should be entered and for when the SOLVENT option has been activated in the RUNSPEC section the total hydrocarbon phase (including the solvent) should be entered, that is SOIL = So + Sg + Ss.			
		dimensionless	dimensionless	dimensionless	None
3	KRO	A columnar vector of real values that are either equal or increasing down the column and that are greater than or equal to zero and less than or equal to one that defines the oil relative permeability with respect to gas and connate water saturation. For two phase runs the oil relative permeability should be entered and for when the SOLVENT option has been activated in the RUNSPEC section the relative permeability of the miscible hydrocarbon phase with respect to water. The last value in the column should be zero.			
		dimensionless	dimensionless	dimensionless	None

Notes:

1) The keyword is followed by NTSFUN tables as declared on the TABDIMS keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NSSFUN rows as declared on the TABDIMS keyword in the RUNSPEC section.
3) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.165: SOF2 Keyword Description

Note this keyword should only be used for when the SOLVENT keyword in the RUNSPEC section has been activated. It should not be use for two-phase oil-water runs.

Example

```
--
--       OIL RELATIVE PERMEABILITY TABLES (SOF2)
--
SOF2
--       SOIL         KRO
--       FRAC         FRAC
--       --------     --------
            0.00      0.000000
            0.05      1.197e-5
            0.10      0.000191
            0.15      0.000969
            0.20      0.003065
            0.25      0.007483
            0.30      0.015517
            0.35      0.028747
            0.40      0.049041
            0.45      0.078555
            0.50      0.119730
            0.55      0.175297
            0.60      0.248272
            0.65      0.341961
            0.70      0.459956
            0.75      0.606134
            0.80      0.784664
            0.85      1.000000                          / TABLE NO. 01
--       --------     --------
            0.00      0.000000
            0.05      1.197e-5
            0.10      0.000191
            0.15      0.000969
            0.20      0.003065
            0.25      0.007483
            0.30      0.015517
            0.35      0.028747
            0.40      0.049041
            0.45      0.078555
            0.50      0.119730
            0.55      0.175297
            0.60      0.248272
            0.65      0.341961
            0.70      0.459956
            0.75      0.606134
            0.80      0.784664
            0.85      1.000000                          / TABLE NO. 02
```

The example defines two SOF2 tables for when oil and gas or oil and water are present in the input deck.

8.3.296 SOF3 – OIL SATURATION TABLES WITH RESPECT TO GAS AND WATER (FORMAT TYPE 2)

Description

The SOF3 keyword defines the oil relative permeability versus oil saturation tables for when oil, gas and water are present in the input deck. The keyword should only be used if oil, gas and water are present in the input deck.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SOIL	A columnar vector of real monotonically increasing down the column values starting from zero and terminating at one, that defines the oil or the hydrocarbon solvent saturation. The final entry should be at the connate water saturation, that is 1- Swc.			
		dimensionless	dimensionless	dimensionless	None
3	KROW	A columnar vector of real values that are either equal or increasing down the column and that are greater than or equal to zero and less than or equal to one that defines the oil relative permeability with respect to oil and water saturation. The first value in the column should be zero.			
		dimensionless	dimensionless	dimensionless	None
4	KROG	A columnar vector of real values that are either equal or increasing down the column and that are greater than or equal to zero and less than or equal to one that defines the oil relative permeability with respect to oil, gas and connate water saturation. The first value in the column should be zero.			
		dimensionless	dimensionless	dimensionless	None

Notes:

1) The keyword is followed by NTSFUN tables as declared on the TABDIMS keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NSSFUN rows as declared on the TABDIMS keyword in the RUNSPEC section.
3) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.166: SOF3 Keyword Description

Example

```
--
--       OIL RELATIVE PERMEABILITY TABLES (SOF3)
--
SOF3
--       SOIL         KRO          KROG
--       FRAC         FRAC         FRAC
--       --------     --------     --------
            0.00      0.000000    0.00000
            0.05      1.197e-5    0.00000
            0.10      0.000191    0.00000
            0.15      0.000969    0.00000
            0.20      0.003065    0.00000
            0.25      0.007483    0.00000
            0.30      0.015517    0.05932
            0.35      0.028747    0.13158
            0.40      0.049041    0.21082
            0.45      0.078555    0.29960
            0.50      0.119730    0.40095
            0.55      0.175297    0.51818
            0.60      0.248272    0.65476
            0.65      0.341961    0.81420
            0.70      0.459956    1.00000
            0.75      0.606134    1.00000
            0.80      0.784664    1.00000
            0.85      1.000000    1.00000                  / TABLE NO. 1
--       --------     --------     --------
            0.00      0.000000    0.00000
            0.05      1.197e-5    0.00000
            0.10      0.000191    0.00000
            0.15      0.000969    0.00000
            0.20      0.003065    0.00000
            0.25      0.007483    0.00000
            0.30      0.015517    0.05932
            0.35      0.028747    0.13158
            0.40      0.049041    0.21082
            0.45      0.078555    0.29960
            0.50      0.119730    0.40095
            0.55      0.175297    0.51818
            0.60      0.248272    0.65476
            0.65      0.341961    0.81420
            0.70      0.459956    1.00000
            0.75      0.606134    1.00000
            0.80      0.784664    1.00000
            0.85      1.000000    1.00000                  / TABLE NO. 2
```

The example defines two SOF3 tables for when oil, gas and water are present in the input deck.

8.3.297 SOF32D – OIL SATURATION TABLES WITH RESPECT TO WATER AND GAS (THREE PHASE)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The SOF32D keyword defines the three phase oil relative permeability versus water and gas saturation tables for when oil, gas and water are present in the input deck. The keyword should only be used if oil, gas and water are present in the input deck. Normally the simulator calculates the three-phase oil relative permeabilities based on the entered two phase tables of water-oil and gas-oil, combined with the STONE1 and STONE2 keywords in the PROPS section that determine the method used to generate the thee phase oil relative permeability curves. SOF32D allows for the direct input of the three phase tables, as such the STONE1 and STONE2 keywords should not be entered if SOF32D is used in the input deck.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.298 SOGCR – END-POINT SCALING GRID CELL CRITICAL OIL SATURATION WITH RESPECT TO GAS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

SOGCR defines the critical oil saturation for all the cells in the model via an array when the end-point scaling option has been invoked via the ENDSCALE keyword in the RUNSPEC section. The critical oil saturation with respect to gas is defined as the maximum oil saturation for which the oil relative permeability is zero in a two-phase gas-oil relative permeability table. The keyword can be used with all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SOGCR	SOGCR is an array of real numbers assigning the critical oil saturation with respect to gas values to each cell in the model. The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword. Repeat counts may be used, for example 30*0.30 dimensionless			Taken from cell allocated relative permeability table.
		dimensionless	dimensionless	dimensionless	

Notes:

1) Note this the directional independent version of the critical gas saturation used with the end-point scaling option. If directional end-point scaling has been activated then the SOGCRX± , SOGCRY± and SOGCRZ± series of keyword should be used.
2) If the value for a cell has been defaulted then OPM Flow uses the value from the cell's relative permeability table.
3) The keyword is terminated by a "/".

Table 8.167: SOGCR Keyword Description

End-point scaling allows the entered relative permeability functions to be re-scaled based on the saturation end-points allocated to each cell by the SWL, SWCR, SWU, SGL, SGCR, SGU, SOWCR, and SOGCR saturation grid arrays for the saturation end-points, and the KRG, KRGR, KRO, KRORG, KRORW, KRW and KRWR relative permeability grid cell arrays for the relative permeability end-point data. In addition end-point scaling may be directional dependent in which case the directional dependent versions of the aforementioned arrays should be used, that is SOGCRX, SOGCRY and SOGCRZ instead of SOGCR. There is also the facility to make the directional end-point scaling reversible or non-reversible and if the non-reversible option is selected the non-reversible versions of the aforementioned arrays should be used, that is SOGCRX, SOGCRX-, SOGCRY, SOGCRY-, SOGCRZ and SOGCRZ-, instead of the SOGCR keyword.

Example

```
--
--       DEFINE GRID BLOCK END-POINT SOGCR DATA FOR ALL CELLS
--       (FOR NX x NY x NZ = 300)
--
SOGCR
         300*0.200                                                    /
```

The above example defines a constant critical gas saturation of 0.20 to all 300 cells in the model as defined by the DIMENS keyword in the RUNSPEC section.

8.3.299 SOMGAS – STONE1 MODEL MINIMUM OIL SATURATION VERSUS GAS SATURATION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword defines the minimum oil saturation as a function of gas saturation for Stone's[246] first three phase oil relative permeability model as modified by Aziz and Settari[247]. If the STONE1 and STONE2 keywords are not present in the input deck then the default three phase oil relative permeability model is employed. The SOMGAS and STONE1 keywords should only be used in three phase runs containing the oil, gas and water phases. The keyword is optional.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

[246] *Stone, H. L. "Probability Model for Estimating Three-Phase Relative Permeability," paper SPE 2116, Journal of Canadian Petroleum Technology (1973) 22, No. 2, 214-218.*

[247] *Aziz, K. and Settari, A. Petroleum Reservoir Simulation, London, UK, Applied Science Publishers (1979), page 398.*

8.3.300 SOMWAT – STONE1 MODEL MINIMUM OIL SATURATION VERSUS WATER SATURATION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword defines the minimum oil saturation as a function of water saturation for Stone's[248] first three phase oil relative permeability model as modified by Aziz and Settari[249]. If the STONE1 and STONE2 keywords are not present in the input deck then the default three phase oil relative permeability model is employed. The SOMWAT and STONE1 keywords should only be used in three phase runs containing the oil, gas and water phases. The keyword is optional.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

[248] *Stone, H. L. "Probability Model for Estimating Three-Phase Relative Permeability," paper SPE 2116, Journal of Canadian Petroleum Technology (1973) 22, No. 2, 214-218.*

[249] *Aziz, K. and Settari, A. Petroleum Reservoir Simulation, London, UK, Applied Science Publishers (1979), page 398.*

8.3.301 SORWMIS – MISCIBLE RESIDUAL OIL VERSUS WATER SATURATION FUNCTIONS

Description

SORWMIS defines the dependency between the miscible residual oil saturation and the water saturation, for when the MISCIBLE keyword in the RUNSPEC section has been activated. The keyword can only be used with the MISCIBLE option and for when the oil, water and gas phases are active in the model.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SWAT	A columnar vector of real monotonically increasing down the column values starting from zero and terminating at one, that defines the water saturation.			
		dimensionless	dimensionless	dimensionless	None
2	SORMIS	A columnar vector of real equal or increasing down the column values that are greater than or equal to zero and less than one, that define the corresponding miscible residual oil saturation for the corresponding water saturation SWAT.			
		dimensionless	dimensionless	dimensionless	None

Notes:

1) The keyword is followed by NTMISC tables as declared on the MISCIBLE keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NSMISC rows as declared on the MISCIBLE keyword in the RUNSPEC section.
3) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.168: SORWMIS Keyword Description

Example

```
--
--       MISCIBLE RESIDUAL OIL VERSUS WATER SATURATION TABLE
--
SORWMIS
--       SWAT       SORWMIS
--       FRAC       FRAC
--       -------    --------
          0.0000     0.0000
          0.2000     0.0000
          1.0000     0.0000                              / TABLE NO. 01
--       SWAT       SORWMIS
--       FRAC       FRAC
--       -------    --------
          0.0000     0.0000
          0.3000     0.1000
          0.7500     0.1500                              / TABLE NO. 02
```

The above example defines two miscible residual oil versus water saturation tables assuming NTMISC equals two and NSMISC is greater than or equal to three on the MISCIBLE keyword in the RUNSPEC section.

8.3.302 SOWCR – END-POINT SCALING GRID CELL CRITICAL OIL SATURATION WITH RESPECT TO WATER

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

SOWCR defines the critical oil saturation for all the cells in the model via an array when the end-point scaling option has been invoked via the ENDSCALE keyword in the RUNSPEC section. The critical oil saturation with respect to water is defined as the maximum oil saturation for which the oil relative permeability is zero in a two-phase oil-water relative permeability table.

The keyword can be used with all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SOWCR	SOWCR is an array of real numbers assigning the critical oil saturation with respect to water values to each cell in the model. The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword. Repeat counts may be used, for example 30*0.30			Taken from cell allocated relative permeability table.
		dimensionless	dimensionless	dimensionless	

Notes:

1) Note this the directional independent version of the critical gas saturation used with the end-point scaling option. If directional end-point scaling has been activated then the SOWCRX± , SOWCRY± and SOWCRZ± series of keyword should be used.

2) If the value for a cell has been defaulted then OPM Flow uses the value from the cell's relative permeability table.

3) The keyword is terminated by a "/".

Table 8.169: SOWCR Keyword Description

End-point scaling allows the entered relative permeability functions to be re-scaled based on the saturation end-points allocated to each cell by the SWL, SWCR, SWU, SGL, SGCR, SGU, SOWCR, and SOGCR saturation grid arrays for the saturation end-points, and the KRG, KRGR, KRO, KRORG, KRORW, KRW and KRWR relative permeability grid cell arrays for the relative permeability end-point data. In addition end-point scaling may be directional dependent in which case the directional dependent versions of the aforementioned arrays should be used, that is SOWCRX, SOWCRY and SOWCRZ instead of SOWCR. There is also the facility to make the directional end-point scaling reversible or non-reversible and if the non-reversible option is selected the non-reversible versions of the aforementioned arrays should be used, that is SOWCRX, SOWCRX-, SOWCRY, SOWCRY-, SOWCRZ and SOWCRZ-, instead of the SOWCR keyword.

Example

```
--
--       DEFINE GRID BLOCK END-POINT SOWCR DATA FOR ALL CELLS
–        (FOR NX x NY x NZ = 300)
--
SOWCR
         300*0.200                                                    /
```

The above example defines a constant critical oil saturation of 0.20 to all 300 cells in the model as defined by the DIMENS keyword in the RUNSPEC section.

8.3.303 SPECHEAT – DEFINE THE SPECIFIC HEAT OF OIL, WATER AND GAS

Description

SPECHEAT defines the specific heat of the oil, water and gas phases for various PVT regions in the model for when the THERMAL option has been activated in the RUNSPEC section. The number of SPECHEAT vector data sets is defined by the NTPVT parameter on the TABDIMS keyword in the RUNSPEC section and the allocation of the SPECHEAT data sets to different grid blocks in the model is done via the PVTNUM keyword in the REGIONS section.

This keyword can only be used if OPM Flow's thermal option has been activated by the THERMAL keyword in the RUNSPEC section. Note this is different to the commercial simulator that uses the TEMP keyword in the RUNSPEC section to activate the black-oil thermal model.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	TEMP	A columnar vector of real monotonically increasing down the column values that define the temperature for the corresponding oil, water and gas specific heat values.			
		°F	°C	°C	None
2	OILSHEAT	OILSHEAT is a columnar vector of positive real numbers defining the specific heat of oil at the corresponding temperature, TEMP.			
		Btu/lb/oR	kJ/kg/K	J/gm/K	None
3	WATSHEAT	WATSHEAT is a columnar vector of positive real numbers defining the specific heat of water at the corresponding temperature, TEMP.			
		Btu/lb/oR	kJ/kg/K	J/gm/K	None
4	GASSHEAT	GASHEAT is a columnar vector of positive real numbers defining the specific heat of gas at the corresponding temperature, TEMP.			
		Btu/lb/oR	kJ/kg/K	J/gm/K	None

Notes:

1) The keyword is followed by NTPVT tables as declared on the TABDIMS keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NPPVT rows as declared on the TABDIMS keyword in the RUNSPEC section.
3) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.170: SPECHEAT Keyword Description

See also the SPECROCK keyword to define the reservoir rock specific heat.

Example

The example below defines three fluid phases specific heat versus temperature tables assuming NTPVT equals three and NPPVT is greater than or equal to two on the TABDIMS keyword in the RUNSPEC section.

```
--
--       SPECIFIC HEAT OF OIL, WATER AND GAS TABLE
--
SPECHEAT
--       TEMP       SPECHEAT   SPECHEAT   SPECHEAT
--                   OIL        WATER      GAS
--       -------    --------   --------   --------
           0.000     0.5000     1.5000     0.5000
         250.000     0.5000     1.5000     0.5000          / TABLE NO. 01
--       TEMP       SPECHEAT   SPECHEAT   SPECHEAT
--                   OIL        WATER      GAS
--       -------    --------   --------   --------
           0.000     0.5500     1.5000     0.5000
         260.000     0.5500     1.5000     0.5000          / TABLE NO. 02
--       TEMP       SPECHEAT   SPECHEAT   SPECHEAT
--                   OIL        WATER      GAS
--       -------    --------   --------   --------
           0.000     0.5500     1.5500     0.5000
         270.000     0.6000     1.5500     0.5000          / TABLE NO. 03
```

There is no terminating "/" for this keyword.

8.3.304 SPECROCK – DEFINE THE SPECIFIC HEAT OF THE RESERVOIR ROCK

Description

SPECROCK defines the specific heat of the reservoir rock for various PVT regions in the model for when the THERMAL option has been activated in the RUNSPEC section. The number of SPECROCK vector data sets is defined by the NTSFUN parameter on the TABDIMS keyword in the RUNSPEC section and the allocation of the SPECROCK data sets to different grid blocks in the model is done via the SATNUM keyword in the REGIONS section.

This keyword can only be used if OPM's Flow's thermal option has been activated by the THERMAL keyword in the RUNSPEC section. Note this is different to the commercial simulator that uses the TEMP keyword in the RUNSPEC section to activate the black-oil thermal model.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	TEMP	A columnar vector of real monotonically increasing down the column values that define the temperature for the corresponding rock specific heat values.			
		°F	°C	°C	None
2	ROCKHEAT	ROCKHEAT is a columnar vector of positive real numbers defining the specific heat of the rock at the corresponding temperature, TEMP.			
		Btu/ft3/oR	kJ/m3/K	J/cc/K	None

Notes:

1) The keyword is followed by NTSFUN tables as declared on the TABDIMS keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NSSFUN rows as declared on the TABDIMS keyword in the RUNSPEC section.
3) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.171: SPECROCK Keyword Description

See also the SPECHEAT keyword to define the specific heat relationships for the oil, water and gas phases.

Example

The example below defines three rock specific heat versus temperature tables assuming NTSFUN equals three and NSSFUN is greater than or equal to two on the TABDIMS keyword in the RUNSPEC section.

```
--
--        SPECIFIC HEAT OF ROCK
--
SPECROCK
--        TEMP        SPECHEAT
--                    ROCK
--        -------     --------
            0.000      20.000
          250.000      20.000                                / TABLE NO. 01
--        -------     --------
            0.000      21.000
          260.000      21.000                                / TABLE NO. 02
--        -------     --------
            0.000      23.000
          270.000      23.000                                / TABLE NO. 03
```

There is no terminating "/" for this keyword.

8.3.305 SSFN – SOLVENT AND GAS RELATIVE PERMEABILITY TABLES

Description

The SSFN keyword defines the miscible normalized relative permeability tables for when the SOLVENT option has been activated in the RUNSPEC section using the respective keyword. The SOLVENT keyword results in a four component model (oil, water and gas, plus a solvent). This keyword should only be used if the SOLVENT option has been activated.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SGAS	A columnar vector of real monotonically increasing down the column values starting from zero and terminating at one, that defines the gas plus solvent saturation ration which is defined as either: $\frac{S_g}{(S_g + S_s)}$ or $\frac{S_s}{(S_g + S_s)}$ Where Sg is the gas saturation and Ss is the solvent saturation.			
		dimensionless	dimensionless	dimensionless	None
2	KRGt	A columnar vector of real values that are either equal or increasing down the column and that are greater than or equal to zero and less than or equal to one that defines the gas relative permeability. The resulting gas relative permeability is calculated from: $k_{rg} = k_{rgt}(S_g + S_s)k_{rg}^{t}$ where krg^t is the data in this column and krgt is the gas relative permeability from the SGFN keyword.			
		dimensionless	dimensionless	dimensionless	None
3	KRSt	A columnar vector of real values that are either equal or increasing down the column and that are greater than or equal to zero and less than or equal to one that defines the solvent relative permeability. The resulting solvent relative permeability is calculated from: $k_{rs} = k_{rgt}(S_g + S_s)k_{rs}^{t}$ where krS^t is the data in this column and krgt is the gas relative permeability from the SGFN keyword.			
		dimensionless	dimensionless	dimensionless	None

Notes:

1) The keyword is followed by NTSFUN tables as declared on the TABDIMS keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NSSFUN rows as declared on the TABDIMS keyword in the RUNSPEC section.
3) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.172: SSFN Keyword Description

Example

```
--
--        SOLVENT RELATIVE PERMEABILITY TABLES
--
SSFN
--        SGAS        KRGT          KRST
--        FRAC
--        --------  --------      --------
           0.0000    0.0000        0.0000
           1.0000    1.0000        1.0000                    / TABLE NO. 01
--        --------  --------      --------
           0.0000    0.0000        0.0000
           0.2000    0.2000        0.3000
           0.4000    0.3000        0.5000
           0.6000    0.4000        0.7000
           0.8000    0.5000        0.7500
           1.0000    1.0000        1.0000                    / TABLE NO. 02
```

The above example defines two SSFN tables for use with the SOLVENT option.

8.3.306 SSGCR – END-POINT SCALING GRID CELL SURFACTANT CRITICAL GAS SATURATIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

SSGCR defines the surfactant critical gas saturation for all the cells in the model via an array when the end-point scaling option has been invoked via the ENDSCALE keyword and the surfactant phase has been activated by the SURFACT keyword in the RUNSPEC section. The critical gas saturation is defined as the maximum gas saturation for which the gas relative permeability is zero in a two-phase relative permeability table. SSGCR is used to scale the surfactant oil relative permeability to gas data.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SSGCR	SSGCR is an array of real numbers assigning the surfactant critical gas saturation values to each cell in the model. The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword. Repeat counts may be used, for example 30*0.03			Taken from cell allocated relative permeability table.
		dimensionless	dimensionless	dimensionless	

Notes:

1) If the value for a cell has been defaulted then OPM Flow uses the value from the cell's relative permeability table.
2) The keyword is terminated by a "/".

Table 8.173: SSGCR Keyword Description

End-point scaling allows the entered surfactant relative permeability functions to be re-scaled based on the saturation end-points allocated to each cell by the SSWL, SSWCR, SSWU, SSGL, SSGCR, SSGU, SSOWCR, and SSOGCR saturation grid arrays for the saturation end-points, and the SKRG, SKROG, SKROW and SKRW relative permeability grid cell arrays for the relative permeability end-point data.

Example

```
--
--       DEFINE GRID BLOCK END-POINT SSGCR DATA FOR ALL CELLS
--       (FOR NX x NY x NZ = 300)
--
SSGCR
         300*0.050                                                    /
```

The above example defines a constant surfactant critical oil saturation with respect to gas of 0.05 to all 300 cells in the model as defined by the DIMENS keyword in the RUNSPEC section.

8.3.307 SSGL – END-POINT SCALING GRID CELL SURFACTANT CONNATE GAS SATURATIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

SSGL defines the surfactant connate gas saturation for all the cells in the model via an array when the end-point scaling option has been invoked via the ENDSCALE keyword and the surfactant phase has been activated by the SURFACT keyword in the RUNSPEC section. The connate gas saturation is defined as the minimum gas saturation in a two-phase gas relative permeability table. SSGL is used to scale the surfactant oil and water relative permeability data.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SSGL	SSGL is an array of real numbers assigning the surfactant connate gas saturation values to each cell in the model. The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword. Repeat counts may be used, for example 30*0.03			Taken from cell allocated relative permeability table.
		dimensionless	dimensionless	dimensionless	

Notes:

1) If the value for a cell has been defaulted then OPM Flow uses the value from the cell's relative permeability table.
2) The keyword is terminated by a "/".

Table 8.174: SSGL Keyword Description

End-point scaling allows the entered surfactant relative permeability functions to be re-scaled based on the saturation end-points allocated to each cell by the SSWL, SSWCR, SSWU, SSGL, SSGCR, SSGU, SSOWCR, and SSOGCR saturation grid arrays for the saturation end-points, and the SKRG, SKROG, SKROW and SKRW relative permeability grid cell arrays for the relative permeability end-point data.

Example

```
--
--       DEFINE GRID BLOCK END-POINT SSGL DATA FOR ALL CELLS
--       (FOR NX x NY x NZ = 300)
--
SSGL
         300*0.030                                                    /
```

The above example defines a constant surfactant connate gas saturation of 0.03 to all 300 cells in the model as defined by the DIMENS keyword in the RUNSPEC section.

8.3.308 SSOGCR – END-POINT SCALING GRID CELL SURFACTANT CRITICAL OIL SATURATION WITH RESPECT TO GAS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

SSOGCR defines the surfactant critical oil saturation for all the cells in the model via an array when the end-point scaling option has been invoked via the ENDSCALE keyword and the surfactant phase has been activated by the SURFACT keyword in the RUNSPEC section. The critical oil saturation with respect to gas is defined as the maximum oil saturation for which the oil relative permeability is zero in a two-phase gas-oil relative permeability table. SSOGCR scales the surfactant oil relative permeability to gas data.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SSOGCR	SSOGCR is an array of real numbers assigning the surfactant critical oil saturation with respect to gas values to each cell in the model. The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword. Repeat counts may be used, for example 30*0.30 dimensionless			Taken from cell allocated relative permeability table.
		dimensionless	dimensionless	dimensionless	

Notes:

1) If the value for a cell has been defaulted then OPM Flow uses the value from the cell's relative permeability table.
2) The keyword is terminated by a "/".

Table 8.175: SSOGCR Keyword Description

End-point scaling allows the entered surfactant relative permeability functions to be re-scaled based on the saturation end-points allocated to each cell by the SSWL, SSWCR, SSWU, SSGL, SSGCR, SSGU, SSOWCR, and SSOGCR saturation grid arrays for the saturation end-points, and the SKRG, SKROG, SKROW and SKRW relative permeability grid cell arrays for the relative permeability end-point data.

Example

```
--
--       DEFINE GRID BLOCK END-POINT SSOGCR DATA FOR ALL CELLS
--       (FOR NX x NY x NZ = 300)
--
SSOGCR
         300*0.200                                                    /
```

The above example defines a surfactant constant critical oil saturation with respect to gas of 0.20 to all 300 cells in the model as defined by the DIMENS keyword in the RUNSPEC section.

8.3.309 SSOWCR – END-POINT SCALING GRID CELL SURFACTANT CRITICAL OIL SATURATION WITH RESPECT TO WATER

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

SSOWCR defines the surfactant critical oil saturation for all the cells in the model via an array when the end-point scaling option has been invoked via the ENDSCALE keyword and the surfactant phase has been activated by the SURFACT keyword in the RUNSPEC section. The critical oil saturation with respect to water is defined as the maximum oil saturation for which the oil relative permeability is zero in a two-phase oil-water relative permeability table. SSOWCR scales the surfactant oil relative permeability to water data.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SSOWCR	SSOWCR is an array of real numbers assigning the surfactant critical oil saturation with respect to water values to each cell in the model. The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword. Repeat counts may be used, for example 30*0.30			Taken from cell allocated relative permeability table.
		dimensionless	dimensionless	dimensionless	

Notes:

1) If the value for a cell has been defaulted then OPM Flow uses the value from the cell's relative permeability table.
2) The keyword is terminated by a "/".

Table 8.176: SSOWCR Keyword Description

End-point scaling allows the entered surfactant relative permeability functions to be re-scaled based on the saturation end-points allocated to each cell by the SSWL, SSWCR, SSWU, SSGL, SSGCR, SSGU, SSOWCR, and SSOGCR saturation grid arrays for the saturation end-points, and the SKRG, SKROG, SKROW and SKRW relative permeability grid cell arrays for the relative permeability end-point data.

Example

```
--
--       DEFINE GRID BLOCK END-POINT SSOWCR DATA FOR ALL CELLS
--       (FOR NX x NY x NZ = 300)
--
SSOWCR
         300*0.200                                                    /
```

The above example defines a constant critical oil saturation of 0.20 to all 300 cells in the model as defined by the DIMENS keyword in the RUNSPEC section.

8.3.310 SSWCR – END-POINT SCALING GRID CELL CRITICAL WATER SATURATION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

SSWCR defines the surfactant critical water saturation for all the cells in the model via an array when the end-point scaling option has been invoked via the ENDSCALE keyword and the surfactant phase has been activated by the SURFACT keyword in the RUNSPEC section. The critical water saturation is defined as the maximum water saturation for which the water relative permeability is zero in a two-phase relative permeability table. SSWCR scales the surfactant water relative permeability data.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SSWCR	SSWCR is an array of real numbers assigning the surfactant critical water saturation values to each cell in the model. The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword. Repeat counts may be used, for example 30*0.20			Taken from cell allocated relative permeability table.
		dimensionless	dimensionless	dimensionless	

Notes:

1) If the value for a cell has been defaulted then OPM Flow uses the value from the cell's relative permeability table.
2) The keyword is terminated by a "/".

Table 8.177: SSWCR Keyword Description

End-point scaling allows the entered surfactant relative permeability functions to be re-scaled based on the saturation end-points allocated to each cell by the SSWL, SSWCR, SSWU, SSGL, SSGCR, SSGU, SSOWCR, and SSOGCR saturation grid arrays for the saturation end-points, and the SKRG, SKROG, SKROW and SKRW relative permeability grid cell arrays for the relative permeability end-point data.

Example

```
--
--       DEFINE GRID BLOCK END-POINT SSWCR DATA FOR ALL CELLS
--       (FOR NX x NY x NZ = 300)
--
SSWCR
         300*0.200                                                    /
```

The above example defines a constant surfactant critical water saturation of 0.20 to all 300 cells in the model as defined by the DIMENS keyword in the RUNSPEC section.

8.3.311 SSWL – END-POINT SCALING GRID CELL SURFACTANT CONNATE WATER SATURATION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

SSWL defines the surfactant connate water saturation for all the cells in the model via an array when the end-point scaling option has been invoked via the ENDSCALE keyword and the surfactant phase has been activated by the SURFACT keyword in the RUNSPEC section. The connate water saturation is defined as the minimum water saturation in a two-phase water relative permeability table. SSWL scales the surfactant oil relative permeability to water and gas data.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SSWL	SSWL is an array of real numbers assigning the connate water saturation values to each cell in the model. The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword. Repeat counts may be used, for example 30*0.15			Taken from cell allocated relative permeability table.
		dimensionless	dimensionless	dimensionless	

Notes:

1) If the value for a cell has been defaulted then OPM Flow uses the value from the cell's relative permeability table.
2) The keyword is terminated by a "/".

Table 8.178: SSWL Keyword Description

End-point scaling allows the entered surfactant relative permeability functions to be re-scaled based on the saturation end-points allocated to each cell by the SSWL, SSWCR, SSWU, SSGL, SSGCR, SSGU, SSOWCR, and SSOGCR saturation grid arrays for the saturation end-points, and the SKRG, SKROG, SKROW and SKRW relative permeability grid cell arrays for the relative permeability end-point data.

Example

```
--
--        DEFINE GRID BLOCK END-POINT SSWL DATA FOR ALL CELLS
--        (FOR NX x NY x NZ = 300)
--
SSWL
          300*0.150                                                    /
```

The above example defines a constant surfactant connate water saturation of 0.15 to all 300 cells in the model as defined by the DIMENS keyword in the RUNSPEC section.

8.3.312 SSWU – END-POINT SCALING GRID CELL SURFACTANT MAXIMUM WATER SATURATION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

SSWU defines the surfactant maximum water saturation for all the cells in the model via an array when the end-point scaling option has been invoked via the ENDSCALE keyword and the surfactant phase has been activated by the SURFACT keyword in the RUNSPEC section. The maximum water saturation is defined as the maximum water saturation in a two-phase water relative permeability table. SSWU scales the surfactant water relative permeability data.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SSWU	SSWU is an array of real numbers assigning the surfactant maximum water saturation values to each cell in the model. The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword. Repeat counts may be used, for example 30*0.70			Taken from cell allocated relative permeability table.
		dimensionless	dimensionless	dimensionless	

Notes:

1) If the value for a cell has been defaulted then OPM Flow uses the value from the cell's relative permeability table.
2) The keyword is terminated by a "/".

Table 8.179: SSWU Keyword Description

End-point scaling allows the entered surfactant relative permeability functions to be re-scaled based on the saturation end-points allocated to each cell by the SSWL, SSWCR, SSWU, SSGL, SSGCR, SSGU, SSOWCR, and SSOGCR saturation grid arrays for the saturation end-points, and the SKRG, SKROG, SKROW and SKRW relative permeability grid cell arrays for the relative permeability end-point data.

Example

```
--
--        DEFINE GRID BLOCK END-POINT SSWU DATA FOR ALL CELLS
--        (FOR NX x NY x NZ = 300)
--
SSWU
          300*0.700                                                    /
```

The above example defines a constant surfactant connate gas saturation of 0.70 to all 300 cells in the model as defined by the DIMENS keyword in the RUNSPEC section.

8.3.313 STOG DEFINE CAPILLARY PRESSURE OIL-GAS SURFACE TENSION VERSUS PRESSURE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The STOG keyword defines capillary pressure oil-gas surface tension versus pressure tables used in adjusting the pressure independent capillary pressure vectors in the SGFN, SGOF or SLGOF saturation tables, entered by their respective keywords in the PROPS section. The SATOPTS keyword in the RUNSPEC section should state the SURFTENS character string to activate the Capillary Pressure Surface Tension Pressure Dependency option. If the STOG keyword is not supplied then no capillary pressure surface tension pressure scaling will occur and the capillary pressure values on the SGFN, SGOF or SLGOF saturation tables will be used directly.

Capillary pressure surface tension pressure scaling can also be used with multi-segment wells, but this facility has not been incorporated in OPM Flow's multi-segment well implementation.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.314 STONE – ACTIVATE STONE'S SECOND THREE PHASE OIL RELATIVE PERMEABILITY MODEL (ALIAS FOR STONE2)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword is an alias for STONE2 keyword that activates Stone's[250] second three phase oil relative permeability model as modified by Aziz and Settari[251]. If the STONE, STONE1 and STONE2 keywords are not present in the input deck then the default three phase oil relative permeability model is employed. The STONE keyword should only be used in three phase runs containing the oil, gas and water phases.

There is no data required for this keyword.

Example

```
--
--       ACTIVATE STONE'S SECOND THREE PHASE RELATIVE PERMEABILITY MODEL
--
STONE
```

The above example switches on the Modified Stone three phase relative permeability model.

[250] *Stone, H. L. "Estimation of Three-Phase Relative Permeability and Residual Oil Data," Journal of Canadian Petroleum Technology (1973) 12, No. 4, 53-61.*

[251] *Aziz, K. and Settari, A. Petroleum Reservoir Simulation, London, UK, Applied Science Publishers (1979), page 398.*

8.3.315 STONE1 – ACTIVATE STONE'S FIRST THREE PHASE OIL RELATIVE PERMEABILITY MODEL

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword activates Stone's[252] first three phase oil relative permeability model as modified by Aziz and Settari[253]. If the STONE1 and STONE2 keywords are not present in the input deck then the default three phase oil relative permeability model is employed. The STONE1 keyword should only be used in three phase runs containing the oil, gas and water phases.

There is no data required for this keyword.

Example

```
--
--       ACTIVATE STONE'S FIRST THREE PHASE RELATIVE PERMEABILITY MODEL
--
STONE1
```

The above example switches on the Modified Stone three phase relative permeability model.

[252] *Stone, H. L. "Probability Model for Estimating Three-Phase Relative Permeability," paper SPE 2116, Journal of Canadian Petroleum Technology (1973) 22, No. 2, 214-218.*

[253] *Aziz, K. and Settari, A. Petroleum Reservoir Simulation, London, UK, Applied Science Publishers (1979), page 398.*

8.3.316 STONE1EX – DEFINE STONE'S FIRST THREE PHASE OIL RELATIVE PERMEABILITY PARAMETER

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword defines the exponent used in Stone's[254] first three phase oil relative permeability model as modified by Aziz and Settari[255]. The STONE1EX keyword should only be used in three phase runs containing the oil, gas and water phases and when the STONE1 keyword in the PROPS section has been used to activate Stone's first three phase oil relative permeability model.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	STONEPAR1	A real positive value that defines the exponent to be used in the Modified Stone first three phase oil relative permeability model.			
		dimensionless	dimensionless	dimensionless	None

Notes:

1) The keyword is followed by NTSFUN records as declared on the TABDIMS keyword in the RUNSPEC section.
2) Each record must contain only one value and is terminated by a "/"
3) There is no "/" terminator for the keyword.

Table 8.180: STONE1EX Keyword Description

If the STONE, STONE1 and STONE2 keywords are not present in the input deck then the default three phase oil relative permeability model is employed.

Example

Given NTSFUN equals five on the TABDIMS keyword in the RUNSPEC section, then:

```
--
--       DEFINE STONE'S FIRST THREE PHASE RELATIVE PERMEABILITY MODEL PARAMETER
--
STONE1EX
         1.000                                          / SATURATION TABLE NO. 01
         1.000                                          / SATURATION TABLE NO. 02
         2.000                                          / SATURATION TABLE NO. 03
         1.000                                          / SATURATION TABLE NO. 04
         3.000                                          / SATURATION TABLE NO. 05
```

Defines the exponents to be used in the Modified Stone first three phase oil relative permeability model, for each of the five saturation tables.

[254] *Stone, H. L. "Probability Model for Estimating Three-Phase Relative Permeability," paper SPE 2116, Journal of Canadian Petroleum Technology (1973) 22, No. 2, 214-218.*

[255] *Aziz, K. and Settari, A. Petroleum Reservoir Simulation, London, UK, Applied Science Publishers (1979), page 398.*

8.3.317 STONE2 – ACTIVATE STONE'S SECOND THREE PHASE OIL RELATIVE PERMEABILITY MODEL

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword activates Stone's[256] second three phase oil relative permeability model as modified by Aziz and Settari[257]. If the STONE, STONE1 and STONE2 keywords are not present in the input deck then the default three phase oil relative permeability model is employed. The STONE2 keyword should only be used in three phase runs containing the oil, gas and water phases.

There is no data required for this keyword.

Example

```
--
--       ACTIVATE STONE'S SECOND THREE PHASE RELATIVE PERMEABILITY MODEL
--
STONE2
```

The above example switches on the Modified Stone three phase relative permeability model.

[256] *Stone, H. L. "Estimation of Three-Phase Relative Permeability and Residual Oil Data," Journal of Canadian Petroleum Technology (1973) 12, No. 4, 53-61.*

[257] *Aziz, K. and Settari, A. Petroleum Reservoir Simulation, London, UK, Applied Science Publishers (1979), page 398.*

8.3.318 STOW DEFINE CAPILLARY PRESSURE OIL-WATER SURFACE TENSION VERSUS PRESSURE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The STOW keyword defines capillary pressure oil-water surface tension versus pressure tables used in adjusting the pressure independent capillary pressure vectors in the SWFN or SWOF saturation tables, entered by their respective keywords in the PROPS section. The SATOPTS keyword in the RUNSPEC section should state the SURFTENS character string to activate the Capillary Pressure Surface Tension Pressure Dependency option. If the STOW keyword is not supplied then no capillary pressure surface tension pressure scaling will occur and the capillary pressure values on the SWFN or SWOF saturation tables will be used directly.

Capillary pressure surface tension pressure scaling can also be used with multi-segment wells, but this facility has not been incorporated in OPM Flow's multi-segment well implementation.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.319 STWG DEFINE CAPILLARY PRESSURE WATER-GAS SURFACE TENSION VERSUS PRESSURE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The STWG keyword defines capillary pressure water-gas surface tension versus pressure tables for use with multi-segment wells. This facility has not been incorporated in OPM Flow's multi-segment well implementation. Note that STWG is not required for Capillary Pressure Surface Tension Pressure Dependency option.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.320 SURFADDW – DEFINED SURFACTANT ADSORBED CONCENTRATION VERSUS WETTABILITY FRACTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

SURFADDW defines tables of surfactant adsorbed concentration versus wettability fraction for when the SURFACTW keyword in the RUNSPEC section as been declared to activate the surfactant phase with changing wettability. The tables consists of columnar vectors of adsorbed surfactant concentration versus a wettability fraction that indicates the fraction of phase wettability. Here, a wettability fraction of zero indicates a 100% water wet rock resulting in the SURFWNUM allocated saturation tables being used, and a value of one meaning 100% oil wet rock, with the SATNUM allocated saturations tables being employed. Both the SURFWNUM and SATNUM keywords are in the REGIONS section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.321 SURFADS - DEFINE SURFACTANT ROCK ADSORPTION TABLES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The SURFADS keyword defines the rock surfactant adsorption tables for when the surfactant option has been activated by the SURFACT keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SURCON	A columnar vector of real monotonically increasing down the column values that defines the surfactant concentration in the solution surrounding the rock. The first entry should be zero to define a no surfactant concentration.			
		lb/stb	kg/sm^3	gm/scc	None
2	SURRATIO	A columnar vector of real increasing down the column values that defines the mass of adsorbed surfactant per unit mass of rock of the saturated concentration of surfactant adsorbed by the rock. The first entry should be zero to define a zero ratio of surfactant concentration.			
		lb/lb	kg/kg	gm/gm	None

Notes:

1) The keyword is followed by NTSFUN tables as declared on the TABDIMS keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NSSFUN rows as declared on the TABDIMS keyword in the RUNSPEC section.
3) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.181: SURFADS Keyword Description

See also the ADSORP keyword in the PROPS section that employs adsorption functions, as oppose to adsorption tables, to define rock surfactant adsorption behavior.

Example

```
--
--       SURFACTANT ROCK ADSORPTION TABLE
--
SURFADS
--       SURF       SURF
--       SURCON     SURRATIO
--       -------    --------
             0.0     0.00000
             2.0     0.00003
             4.0     0.00005
             6.0     0.00007
             8.0     0.00009
            10.0     0.00011
            12.0     0.00012
            14.0     0.00015                              / TABLE NO. 01
--       SURF       SURF
--       SURCON     SURRATIO
--       -------    --------
             0.0     0.00000
             3.0     0.00004
             5.0     0.00006
             7.0     0.00008
             8.0     0.00009
            10.0     0.00011                              / TABLE NO. 02
```

The above example defines two surfactant rock adsorption tables assuming NTSFUN equals two and NSSFUN is greater than or equal to eight on the TABDIMS keyword in the RUNSPEC section.

There is no terminating "/" for this keyword.

8.3.322 SURFCAPD – CAPILLARY NUMBER VERSUS MISCIBILITY TABLES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The SURFCAPD keyword defines the relationship between the log of the capillary number and the level of miscibility, for when the Surfactant option has been activated by the SURFACT keyword in the RUNSPEC section. A value of zero for the level of miscibility means fully immiscible conditions and consequently a value of one implies fully miscible conditions.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.323 SURFESAL – DEFINE SURFACTANT EFFECTIVE SALINITY COEFFICIENT

Description

This keyword, SURFESAL, defines the surfactant effective salinity coefficient as well as enabling the effective salinity calculation for surfactant adsorption. The keyword should only be used if the BRINE keyword has been declared to activate the brine phase, the ECLMC keyword to enable the Multi-Component Brine model, and the SURFACT keyword has been used to activate the surfact phase. All three keywords are in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.324 SURFROCK - DEFINE SURFACTANT-ROCK PROPERTIES

Description

The SURFROCK keyword defines rock properties for when the Surfactant option has been activated by the SURFACTANT keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	ADINDX	A positive integer of 1 or 2 that defines the surfactant desorption option. 1) then surfactant desorption may occur by retracing the surfactant adsorption isotherm when the local surfactant concentration in the solution decreases. 2) then no surfactant desorption may occur.			
		dimensionless 1	dimensionless 1	dimensionless 1	Defined
2	DENSITY	A real value that defines the rock in-situ density, that is at reservoir conditions.			
		lb/rtb	kg/rm^3	gm/rcc	None

Notes:

1) The keyword is followed by NTSFUN tables as declared on the TABDIMS keyword in the RUNSPEC section, for each surfactant flooding region. There should be only one row per table.

2) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.182: SURFROCK Keyword Description

Example

```
--
--       SURFACTANT-ROCK PROPERTIES
--
SURFROCK
--       DESORP   INSITU
--       OPTN     DENSITY
--       ------   -------
           1      1800.0                          / TABLE NO. 01
           2      1980.0                          / TABLE NO. 02
           1      2005.0                          / TABLE NO. 03
```

The above example defines three surfactant-rock tables, based on the NTSFUN variable on the TABDIMS keyword in the RUNSPEC section being equal to three.

There is no terminating "/" for this keyword.

8.3.325 SURFST - SURFACTANT WATER-OIL SURFACE TENSION VERSUS SURFACTANT CONCENTRATION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The SURFST keyword defines surfactant water-oil surface tension versus surfactant concentration in the water phase tables, used in adjusting the pressure independent capillary pressure vectors in the SWFN or SWOF saturation tables, entered by their respective keywords in the PROPS section. SURFST is also used to adjust the relative permeability curves on the aforementioned tables via the capillary number. The Surfactant option must have been activated by the SURFACTANT keyword in the RUNSPEC section to use this keyword and either this keyword or the SURFSTES keyword, also in the PROPS section, is obligatory in this case.

See also the SURFSTES that defines the surfactant water-oil surface tension as a function of surfactant concentration in the water phase and salt concentration or the effective salinity.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.326 SURFSTES - SURFACTANT WATER-OIL SURFACE TENSION VERSUS SURFACTANT AND SALT CONCENTRATIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The SURFSTES keyword defines surfactant water-oil surface tension versus surfactant concentration in the water phase tables, used in adjusting the pressure independent capillary pressure vectors in the SWFN or SWOF saturation tables, entered by their respective keywords in the PROPS section. SURFSTES is also used to adjust the relative permeability curves on the aforementioned tables via the capillary number. The Surfactant option must have been activated by the SURFACTANT keyword in the RUNSPEC section to use this keyword and either this keyword or the SURFST keyword, also in the PROPS section, is obligatory in this case. In addition, the BRINE keyword in the RUNSPEC section must be activated and the ESSNODE keyword in the PROPS section must be used to define the salt concentration or the effective salinity.

See also the SURFSTS that defines the surfactant water-oil surface tension as a function of surfactant concentration in the water phase only.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.327 SURFVISC – SURFACTANT SOLUTION VISCOSITY VERSUS CONCENTRATION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

SURFSVISC defines the surfactant viscosity relationship of solution water viscosity with respect to increasing surfactant concentration within a grid block. The surfactant option must be activated by the SURFACT keyword in the RUNSPEC section in order to use this keyword.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SURFCON	A columnar vector of real monotonically increasing down the column values that defines the surfactant concentration in the solution surrounding the rock. The first entry should be zero to define a no surfactant concentration.			
		lb/stb	kg/sm^3	gm/scc	None
2	SURFVISC	A columnar vector of real positive values that defines the solution water viscosity of the solution for the given SURFCON entry at the reference pressure value, PRES, entered on the PVTW keyword in the PROPS section.			
		cP	cP	cP	None

Notes:

1) The keyword is followed by NTPVT tables as declared on the TABDIMS keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NPPVT rows as declared on the TABDIMS keyword in the RUNSPEC section.
3) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.183: SURFVISC Keyword Description

Example

```
--
--       SURFACTANT SOLUTION WATER VISCOSITY TABLES
--
SURFVISC
--       SURF          VISCOSITY
--       SURFCON       SURFVISC
--       --------     -----------
          0.0000       0.3500
          0.0100       0.3900
          0.0200       0.4200
          0.0300       0.4300                          / TABLE NO. 01
```

```
--       SURF          VISCOSITY
--       SURFCON       VISFAC
--       --------     ---------
          0.0000        1.000
          0.0003       10.000
          0.0005       20.000
          0.0007       40.000
          0.0009       45.000
          0.0011       55.000                          / TABLE NO. 02
```

The example defines two surfactant viscosity scaling factor tables, based on the NTPVT variable on the TABDIMS keyword in the RUNSPEC section being equal to two and NPPVT variable on the same keyword being greater than or equal to six.

8.3.328 SWATINIT – DEFINE THE INITIAL WATER SATURATION ARRAY FOR CAPILLARY PRESSURE SCALING

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

SWATINIT defines the initial water saturation for all the cells in the model via an array. The keyword can be used with all grid types. SWATINIT is used to initialize the model by setting each grid block's initial water saturation ("Sw"). If the array is present in the input deck, then OPM Flow will re-scale the water-oil capillary pressure curves entered via the SWFN saturation functions in the PROPS section, so that the resulting initialized Sw matches the values in the SWATINIT array.

Normally the SWATINIT array is generated in the static earth model when calculating the hydrocarbons in-place. volumes using Saturation Height Functions ("SHF") derived from capillary pressure functions. Static earth models do not directly use capillary pressure in these type of calculations as individual cell pressures are not required. There is therefore some potential for inconsistencies to arise between the two sets of formulations. This is normally manifested by extreme scaling in the scaled capillary pressure values calculated by the simulator. If this is the case then the PPCWMAX keyword can be used to set a maximum scaled capillary pressure value. Note that as large values of scaled capillary pressures can result in numerical issues, a more technically sound approach would be to resolve these inconsistencies before continuing with the model build.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SWATINIT	SWATINIT is an array of real positive numbers that are greater than or equal to zero and less than or equal to one, that define the initial water saturation values to each cell in the model. Repeat counts may be used, for example 3000*0.15			
		dimensionless	dimensionless	dimensionless	None

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.

2) The keyword is terminated by a "/".

Table 8.184: SWATINIT Keyword Description

See also the PPCWMAX to control the re-scaling of the capillary pressure entries on the SWFN saturation function keyword in the PROPS section.

Example

```
--
--        DEFINE GRID BLOCK INITIAL SW DATA FOR ALL CELLS
--        (BASED ON NX x NY x NZ = 300)
--
SWATINIT
          300*0.300                                                    /
```

The above example defines a constant initial water saturation of 0.300 to all 300 cells in the model as defined by the DIMENS keyword in the RUNSPEC section.

8.3.329 SWCR – END-POINT SCALING GRID CELL CRITICAL WATER SATURATION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

SWCR defines the critical water saturation for all the cells in the model via an array when the end-point scaling option has been invoked via the ENDSCALE keyword in the RUNSPEC section. The critical water saturation is defined as the maximum water saturation for which the water relative permeability is zero in a two-phase relative permeability table.

The keyword can be used with all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SWCR	SWCR is an array of real numbers assigning the critical water saturation values to each cell in the model. The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword. Repeat counts may be used, for example 30*0.20			Taken from cell allocated relative permeability table.
		dimensionless	dimensionless	dimensionless	

Notes:

1) Note this the directional independent version of the critical water saturation used with the end-point scaling option. If directional end-point scaling has been activated then the SWCRX± , SWCRY± and SWCRZ± series of keyword should be used.

2) If the value for a cell has been defaulted then OPM Flow uses the value from the cell's relative permeability table.

3) The keyword is terminated by a "/".

Table 8.185: SWCR Keyword Description

End-point scaling allows the entered relative permeability functions to be re-scaled based on the saturation end-points allocated to each cell by the SWL, SWCR, SWU, SGL, SGCR, SGU, SOWCR, and SOGCR saturation grid arrays for the saturation end-points, and the KRG, KRGR, KRO, KRORG, KRORW, KRW and KRWR relative permeability grid cell arrays for the relative permeability end-point data. In addition end-point scaling may be directional dependent in which case the directional dependent versions of the aforementioned arrays should be used, that is SWCRX, SWCRY and SWCRZ instead of SWCR. There is also the facility to make the directional end-point scaling reversible or non-reversible and if the non-reversible option is selected the non-reversible versions of the aforementioned arrays should be used, that is SWCRX, SWCRX-, SWCRY, SWCRY-, SWCRZ and SWCRZ-, instead of the SWCR keyword.

Example

```
--
--        DEFINE GRID BLOCK END-POINT SWCR DATA FOR ALL CELLS
--        (FOR NX x NY x NZ = 300)
--
SWCR
          300*0.200                                                    /
```

The above example defines a constant critical water saturation of 0.20 to all 300 cells in the model as defined by the DIMENS keyword in the RUNSPEC section.

8.3.330 SWF32D – WATER SATURATION TABLES WITH RESPECT TO OIL AND GAS (THREE PHASE)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The SWF32D keyword defines the three phase water relative permeability versus oil and gas saturation tables for when oil, gas and water are present in the input deck. The keyword should only be used if oil, gas and water are present in the input deck. Normally the simulator calculates the three-phase oil relative permeabilities based on the entered two phase tables of water-oil and gas-oil, combined with the STONE1 and STONE2 keywords in the PROPS section that determine the method used to generate the thee phase water relative permeability curves. SWF32D allows for the direct input of the three phase tables, as such the STONE1 and STONE2 keywords should not be entered if SWF32D is used in the input deck.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.331 SWFN – WATER SATURATION TABLES (FORMAT TYPE 2)

Description

The SWFN keyword defines the water relative permeability and water-oil capillary pressure data versus water saturation tables for when water is present in the input deck. This keyword should only be used if water is present in the run.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SWAT	A columnar vector of real monotonically increasing down the column values starting from zero and terminating at one, that defines the water saturation.			
		dimensionless	dimensionless	dimensionless	None
2	KRW	A columnar vector of real values that are either equal or increasing down the column and that are greater than or equal to zero and less than or equal to one that defines the water relative permeability with respect to gas saturation. The first value in the column should be zero.			
		dimensionless	dimensionless	dimensionless	None
3	PCWO	A columnar vector of real values that are either equal or increasing down the column that defines the water-oil relative capillary pressure. If the SWATINIT keyword has been used to initialize the model then columnar vector has to be strictly monotonically increasing.			
		psia	bars	atm	None

Notes:

1) The keyword is followed by NTSFUN tables as declared on the TABDIMS keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NSSFUN rows as declared on the TABDIMS keyword in the RUNSPEC section.
3) Items (2) and (3) may be defaulted, in which case linear interpolation or constant extrapolation will be used to calculate the missing values. At least two values must be entered for item (2), and at least one value must be entered for item (3).
4) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.186: SWFN Keyword Description

Example

```
--
--       WATER RELATIVE PERMEABILITY TABLES (SWFN)
--
SWFN
--       SWAT       KRW        PCOW
--       FRAC       FRAC       PSIA
--       --------   --------   -------
            0.15     0.00000     0.0
            0.20     6.25e-6     0.0
            0.25     0.00010     0.0
            0.30     0.00050     0.0
            0.35     0.00160     0.0
            0.40     0.00390     0.0
            0.45     0.00810     0.0
            0.50     0.01500     0.0
            0.55     0.02560     0.0
            0.60     0.04100     0.0
            0.65     0.06250     0.0
            0.70     0.09150     0.0
            0.75     0.12960     0.0
            0.80     0.17850     0.0
            0.85     0.24010     0.0
            0.90     0.31640     0.0
            0.95     0.40960     0.0
            1.00     0.52200     0.0                    / TABLE NO. 1
--       --------   --------   -------
            0.15     0.00000     0.0
            0.20     6.25e-6     0.0
            0.25     0.00010     0.0
            0.30     0.00050     0.0
            0.35     0.00160     0.0
            0.40     0.00390     0.0
            0.45     0.00810     0.0
            0.50     0.01500     0.0
            0.55     0.02560     0.0
            0.60     0.04100     0.0
            0.65     0.06250     0.0
            0.70     0.09150     0.0
            0.75     0.12960     0.0
            0.80     0.17850     0.0
            0.85     0.24010     0.0
            0.90     0.31640     0.0
            0.95     0.40960     0.0
            1.00     0.52200     0.0                    / TABLE NO. 2
```

The example defines two SWFN tables for use when water is present in the run.

8.3.332 SWL – END-POINT SCALING GRID CELL CONNATE WATER SATURATION

Description

SWL defines the connate water saturation for all the cells in the model via an array when the end-point scaling option has been invoked via the ENDSCALE keyword in the RUNSPEC section. The connate water saturation is defined as the minimum water saturation in a two-phase water relative permeability table.

The keyword can be used with all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SWL	SWL is an array of real numbers assigning the connate water saturation values to each cell in the model. The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword. Repeat counts may be used, for example 30*0.15			Taken from cell allocated relative permeability table.
		dimensionless	dimensionless	dimensionless	

Notes:

1) Note this the directional independent version of the connate gas saturation array used with the end-point scaling option. If directional end-point scaling has been activated then the SWLX± , SWLY± and SWZ± series of keyword should be used.
2) If the value for a cell has been defaulted then OPM Flow uses the value from the cell's relative permeability table.
3) The keyword is terminated by a "/".

Table 8.187: SWL Keyword Description

End-point scaling allows the entered relative permeability functions to be re-scaled based on the saturation end-points allocated to each cell by the SWL, SWCR, SWU, SGL, SGCR, SGU, SOWCR, and SOGCR saturation grid arrays for the saturation end-points, and the KRG, KROG, KROW and KRW relative permeability grid cell arrays for the relative permeability end-point data. In addition end-point scaling may be directional dependent in which case the directional dependent versions of the aforementioned arrays should be used, that is SWLX, SWLY and SWLZ instead of SWL. There is also the facility to make the directional end-point scaling reversible or non-reversible and if the non-reversible option is selected the non-reversible versions of the aforementioned arrays should be used, that is SWLX, SWLX-, SWLY, SWLY-, SWLZ and SWLZ-, instead of the SWL keyword.

Example

```
--
--       DEFINE GRID BLOCK END-POINT SWL DATA FOR ALL CELLS
--       (FOR NX x NY x NZ = 300)
--
SWL
         300*0.150                                                    /
```

The above example defines a constant connate water saturation of 0.15 to all 300 cells in the model as defined by the DIMENS keyword in the RUNSPEC section.

8.3.333 SWLPC – END-POINT SCALING GRID CELL CAPILLARY PRESSURE CONNATE WATER SATURATIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

SWLPC defines the connate water saturation for all the cells in the model via an array when the end-point scaling option has been invoked via the ENDSCALE keyword in the RUNSPEC section. The connate water saturation is defined as the minimum water saturation in a two-phase water relative permeability table. The keyword only applies the scaling to the drainage capillary pressures tables, unlike the SWL keyword that applies the scaling to both the capillary pressure and relative permeability tables. The keyword can be used with all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SWLPC	SWLPC is an array of real numbers assigning the connate water saturation values to each cell in the model. The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword. If SWLPC is omitted from the input deck the values will be defaulted to those on the SWL series of keywords. If the SWL series of keywords are missing from the input deck then the values are taken from the cell allocated capillary pressure table. Repeat counts may be used, for example 30*0.03			Taken from SGL or from the cell allocated capillary pressure table.
		dimensionless	dimensionless	dimensionless	

Notes:

1) Note this the directional independent version of the connate gas saturation array used with the end-point scaling option. If directional end-point scaling has been activated then the SWLX± , SWLY± and SWZ± series of keyword should be used.

2) The keyword is terminated by a "/".

Table 8.188: SWLPC Keyword Description

End-point scaling allows the entered relative permeability functions to be re-scaled based on the saturation end-points allocated to each cell by the SWL, SWCR, SWU, SGL, SGCR, SGU, SOWCR, and SOGCR saturation grid arrays for the saturation end-points, and the KRG, KROG, KROW and KRW relative permeability grid cell arrays for the relative permeability end-point data. In addition end-point scaling may be directional dependent in which case the directional dependent versions of the aforementioned arrays should be used, that is SWLX, SWLY and SWLZ instead of SWL or SWLPC. There is also the facility to make the directional end-point scaling reversible or non-reversible and if the non-reversible option is selected the non-reversible versions of the aforementioned arrays should be used, that is SWLX, SWLX-, SWLY, SWLY-, SWLZ and SWLZ-, instead of the SWL or SWLPC keywords.

Missing Some Functionality - Use with Caution.

Example

```
--
--       DEFINE GRID BLOCK END-POINT SWLPC DATA FOR ALL CELLS
--       (FOR NX x NY x NZ = 300)
--
SWLPC
         300*0.150                                                              /
```

The above example defines a constant connate water saturation of 0.15 to all 300 cells in the model as defined by the DIMENS keyword in the RUNSPEC section.

8.3.334 SWOF – WATER-OIL SATURATION TABLES (FORMAT TYPE 1)

Description

The SWOF keyword defines the water and oil relative permeability and water-oil capillary pressure data versus water saturation tables for when water and oil are present in the input deck. This keyword should only be used if water and oil are present in the run.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SWAT	A columnar vector of real monotonically increasing down the column values starting from zero and terminating at one, that defines the water saturation. The first entry is the connate water saturation Swc and the last entry should be 1.0.			
		dimensionless	dimensionless	dimensionless	None
2	KRW	A columnar vector of real values that are either equal or increasing down the column and that are greater than or equal to zero and less than or equal to one that defines the water relative permeability with respect to gas saturation. The first value in the column should be zero.			
		dimensionless	dimensionless	dimensionless	None
3	KRO	A columnar vector of real values that are either equal or decreasing down the column and that are greater than or equal to zero and less than or equal to one that defines the oil relative permeability with respect to oil and water saturation. When gas is active in the run, the first entry the column, that is at krow(So = 1-Swc), must be the same as the first entry in the corresponding SGOF or SLGOF table, that is at krog(Sg = 0). The first value in the column should be one.			
		dimensionless	dimensionless	dimensionless	None
4	PCWO	A columnar vector of real values that are either equal or decreasing down the column that defines the water-oil relative capillary pressure. If the SWATINIT keyword has been used to initialize the model then columnar vector has to be strictly monotonically increasing.			
		psia	bars	atm	None

Notes:

1) The keyword is followed by NTSFUN tables as declared on the TABDIMS keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NSSFUN rows as declared on the TABDIMS keyword in the RUNSPEC section.
3) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.189: SWOF Keyword Description

Example

The following example is based on NTSFUN equals two on the TABDIMS keyword in the RUNSPEC section.

```
--
--       WATER-OIL RELATIVE PERMEABILITY TABLES (SWOF)
--
SWOF
--       SWAT       KRW       KROW      PCOW
--       FRAC                           PSIA
--       --------   --------  -------   -------
         0.200000   0.0000    0.9000    0.000000
         0.238616   0.0002    0.7664    0.000000
         0.245309   0.0004    0.7443    0.000000
         0.261989   0.0010    0.6907    0.000000
         0.303091   0.0044    0.5671    0.000000
         0.368269   0.0191    0.3962    0.000000
         0.435026   0.0519    0.2528    0.000000
         0.486387   0.0940    0.1643    0.000000
         0.550683   0.1725    0.0803    0.000000
         0.575342   0.2115    0.0559    0.000000
         0.599076   0.2542    0.0367    0.000000
         0.621294   0.2991    0.0223    0.000000
         0.642171   0.3458    0.0120    0.000000
         0.658984   0.3868    0.0061    0.000000
         0.671123   0.4183    0.0030    0.000000
         0.679268   0.4403    0.0015    0.000000
         0.684963   0.4562    0.0008    0.000000
         0.688893   0.4674    0.0004    0.000000
         0.692025   0.4765    0.0002    0.000000
         0.694641   0.4841    0.0001    0.000000
         0.696976   0.4910    0.0000    0.000000
         0.699099   0.4973    0.0000    0.000000
         0.700000   0.5000    0.0000    0.000000
         1.000000   0.9000    0.0000    0.000000          / TABLE NO. 01
--       --------   --------  -------   -------
         0.200000   0.0000    0.9000    0.000000
         0.238616   0.0002    0.7664    0.000000
         0.245309   0.0004    0.7443    0.000000
         0.261989   0.0010    0.6907    0.000000
         0.303091   0.0044    0.5671    0.000000
         0.368269   0.0191    0.3962    0.000000
         0.435026   0.0519    0.2528    0.000000
         0.486387   0.0940    0.1643    0.000000
         0.550683   0.1725    0.0803    0.000000
         0.575342   0.2115    0.0559    0.000000
         0.599076   0.2542    0.0367    0.000000
         0.621294   0.2991    0.0223    0.000000
         0.642171   0.3458    0.0120    0.000000
         0.658984   0.3868    0.0061    0.000000
         0.671123   0.4183    0.0030    0.000000
         0.679268   0.4403    0.0015    0.000000
         0.684963   0.4562    0.0008    0.000000
         0.688893   0.4674    0.0004    0.000000
         0.692025   0.4765    0.0002    0.000000
         0.694641   0.4841    0.0001    0.000000
         0.696976   0.4910    0.0000    0.000000
         0.699099   0.4973    0.0000    0.000000
         0.700000   0.5000    0.0000    0.000000
         1.000000   0.9000    0.0000    0.000000          / TABLE NO. 01
```

The example defines two SWOF tables for use when water and oil are present in the run. In the tables the water-oil capillary pressure data has been set to zero.

8.3.335 SWOFLET – WATER-OIL LET RELATIVE PERMEABILITY FUNCTIONS

Description

SWOFLET defines the relative permeability and capillary pressure parameters for the water-oil LET family of models. Both the oil and water phases should be made active in the model via the OIL and WATER keywords in the RUNSPEC section. See section *8.2.6Saturation Table Generation - LET Functions* and Lomeland et al.[258],[259] [260] and [261] for further information on the model.

The keyword is used as a replacement for the SWOF keyword for three-phase oil-gas-water systems, and the LET series of keywords cannot be combined with the standard set of relative permeability keywords.

Note

This is an OPM Flow specific keyword and will therefore cause an error in the commercial simulator.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SWL	SWL is a real positive number less than one, that defines the connate water saturation, that is the smallest water saturation in the LET function.			0.0
		dimensionless	dimensionless	dimensionless	
2	SWCR	SWCR is a real positive number greater than or equal to SWL and less than one, that defines the critical water saturation, that is the largest water saturation for which the water relative permeability is zero, Swirr in equations.			0.0
		dimensionless	dimensionless	dimensionless	
3	LWAT	LWAT is a real positive number that defines the LET Lower empirical parameter Lw for the water phase with the associated oil phase in the LET relative permeability equations.			1.0
		dimensionless	dimensionless	dimensionless	
4	EWAT	EWAT is a real positive number that defines the LET Elevation empirical parameter Ew for the water phase with the associated oil phase in the LET relative permeability equations.			1.0
		dimensionless	dimensionless	dimensionless	

[258] *Lomeland F., Ebeltoft E. and Thomas W.H., 2005. A New Versatile Relative Permeability Correlation. Paper SCA2005-32 presented at the International Symposium of the Society of Core Analysts held in Toronto, Canada, 21-25 August, 2005.*

[259] *Lomeland F. and Ebeltoft E., 2008. A New Versatile Capillary Pressure Correlation. Paper SCA2008-08 presented at the International Symposium of the Society of Core Analysts held in Abu Dhabi, UAE, 29 Oct. – 2 Nov., 2008.*

[260] *Lomeland F., Hasanov B., Ebeltoft E. and Berge M., 2012. A Versatile Representation of Up-scaled Relative Permeability for Field Applications. Paper SPE 154487-MS presented at the EAGE Annual Conference & Exhibition incorporating SPE Europec held in Copenhagen, Denmark, 4-7 June 2012.*

[261] *Lomeland F., 2018.. Overview Of The Let Family Of Versatile Correlations For Flow Functions. Paper SCA2018-056 presented at the International Symposium of the Society of Core Analysts held in Trondheim, Norway, 27-30 August 2018.*

No.	Name	Description			Default
		Field	Metric	Laboratory	
5	TWAT	TWAT is a real positive number that defines the LET Top empirical parameter Tw for the water phase with the associated oil phase in the LET relative permeability equations.			1.0
		dimensionless	dimensionless	dimensionless	
6	KRTWAT	KRTWAT is a real positive number less than one, that defines the relative permeability of water at the maximum water saturation (normally the maximum water saturation is one) Krwt in the LET relative permeability equations.			1.0
		dimensionless	dimensionless	dimensionless	
7	SORW	SORW is a real positive number less than one that defines the residual oil saturation in and oil-water system in the LET equation.s			0.0
		dimensionless	dimensionless	dimensionless	
8	SOWCR	SOWCR is a real positive number less than one that defines critical oil-in-water saturation, that is the largest oil saturation for which the oil relative permeability is zero in an oil-water system.			0.0
		dimensionless	dimensionless	dimensionless	
9	LOIL	LOIL is a real positive number that defines the LET Lower empirical parameter Lo for the oil phase with the associated water phase in the LET relative permeability equationss			1.0
		dimensionless	dimensionless	dimensionless	
10	EOIL	EOIL is a real positive number that defines the LET Elevation empirical parameter Eo for the oil phase with the associated water phase in the LET relative permeability equations.			1.0
		dimensionless	dimensionless	dimensionless	
11	TOIL	TOIL is a real positive number that defines the LET Top empirical parameter To for the oil phase with the associated water phase in the LET relative permeability equations.			1.0
		dimensionless	dimensionless	dimensionless	
12	KRTOIL	KRTOIL is a real positive number less than or equal to one, that defines the relative permeability of oil at the residual oil saturation, Krot in the LET relative permeability equations.			1.0
		dimensionless	dimensionless	dimensionless	
13	LPC	LPC is a real positive number that defines the water-oil LET Lower empirical parameter L in the LET capillary pressure equation.			1.0
		dimensionless	dimensionless	dimensionless	
14	EPC	EPC is a real positive number that defines the water-oil LET Elevation empirical parameter E in the LET capillary pressure equation.			1.0
		dimensionless	dimensionless	dimensionless	

No.	Name	Description			Default
		Field	Metric	Laboratory	
15	TPC	TPC is a real positive number that defines the water-oil LET empirical parameter T in the LET capillary pressure equations.			1.0
		dimensionless	dimensionless	dimensionless	
16	PCIR	PCIR is a real positive number that defines the water-oil capillary pressure at connate water saturation (SWL) in the LET capillary pressure equations.			0.0
		psia	bars	atm	
17	PCIT	PCIT is a real positive number that defines the water-oil threshold capillary pressure at the maximum water saturation in the LET capillary pressure equations.			0.0
		psia	bars	atm	

Notes:

1) The keyword is followed by NTSFUN definitions as declared on the TABDIMS keyword in the RUNSPEC section, with each definition consisting of the above 17 LET parameters.
1) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.190: SWOFLET Keyword Description

Note there a two versions of the LET functions, LET[259] for two-phase flowing conditions and LETx[262] for three-phase flowing conditions. This keyword implements the LET version for an oil-water system.

The functions are dependent on the drainage and imbibition cycle of the wetting phase as well as drainage and inhibition cycle number, since a reservoir may undergo several flooding events. To account for this the system defines the flooding event using the three saturations: Sw, So, and Sg together with the state of the three saturations during the flooding event. The saturation state can be Increasing, Decreasing, or Constant, for a given flooding event cycle number (n). Thus, Sw(D), So(I), Sg(C)1or DIC1, means the water phase is decreasing, the oil phase is increasing and the gas phase is constant for the primary or first cycle (n equals one). This is case for when oil is migrating into the reservoir rock and displacing the initial water contained with the reservoir.

Note

All the LET parameters are dependent on the flooding event and flooding cycle, and thus are expected vary as such. To be clear, the values of SWCR, Lo, Lw etc. should be different for each flooding cycle.

See also the *SGOFLET– Gas-Oil LET Relative Permeability Functions*, and the *SGWFLET – Gas-Water LET Relative Permeability Functions* keywords in this section.

[262] *Lomeland F. and Ebeltoft E., 2013. Versatile Three-phase Correlations for Relative Permeability and Capillary Pressure. Paper SCA2013-034 presented at the International Symposium of the Society of Core Analysts held in Napa Valley, California, USA, 16-19 September, 2013.*

Example

The following example uses the SWOFLET keyword to define two relative oi-water relative permeability tables, based on NTSFUN equals two on the TABDIMS keyword in the RUNSPEC section.

```
--
--       SWOFLET – WATER-OIL LET REL. PERMEABILITY FUNCTIONS (OPM FLOW KEYWORD)
--
SWOFLET
--       SWL       SWCR     L-WAT     E-WAT     T-WAT     KRT-WAT
--       SOR       SOWCR    L-OIL     E-OIL     T-OIL     KRT-OIL
--       L-PC      E-PC     T-PC      PCIR      PCIT
--       -------   ------   -------   -------   -------   -------
         0.00000   0.0000   1.00000   1.00000   1.00000   1.00000
         0.00000   0.0000   1.00000   1.00000   1.00000   1.00000
         1*        1*       1*        1*        1*                  / TABLE NO. 01

         0.00000   0.0000   1.00000   1.00000   1.00000   1.00000
         0.00000   0.0000   1.00000   1.00000   1.00000   1.00000
         1*        1*       1*        1*        1*                  / TABLE NO. 02
```

Here the SWOFLET keyword parameters are all set to their default values.

8.3.336 SWU – END-POINT SCALING GRID CELL MAXIMUM WATER SATURATION

Description

SWU defines the maximum water saturation for all the cells in the model via an array when the end-point scaling option has been invoked via the ENDSCALE keyword in the RUNSPEC section. The maximum water saturation is defined as the maximum water saturation in a two-phase water relative permeability table.

The keyword can be used with all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SWU	SWU is an array of real numbers assigning the maximum water saturation values to each cell in the model. The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword. Repeat counts may be used, for example 30*0.70			Taken from cell allocated relative permeability table.
		dimensionless	dimensionless	dimensionless	

Notes:

1) Note this the directional independent version of the maximum water saturation array used with the end-point scaling option. If directional end-point scaling has been activated then the SWUX± , SWUX± and SWU± series of keyword should be used.
2) If the value for a cell has been defaulted then OPM Flow uses the value from the cell's relative permeability table.
3) The keyword is terminated by a "/".

Table 8.191: SWU Keyword Description

End-point scaling allows the entered relative permeability functions to be re-scaled based on the saturation end-points allocated to each cell by the SWL, SWCR, SWU, SGL, SGCR, SGU, SOWCR, and SOGCR saturation grid arrays for the saturation end-points, and the KRG, KROG, KROW and KRW relative permeability grid cell arrays for the relative permeability end-point data. In addition end-point scaling may be directional dependent in which case the directional dependent versions of the aforementioned arrays should be used, that is SWUX, SWUY and SWUZ instead of SWU. There is also the facility to make the directional end-point scaling reversible or non-reversible and if the non-reversible option is selected the non-reversible versions of the aforementioned arrays should be used, that is SWUX, SWUX-, SWUY, SWUY-, SWUZ and SWUZ-, instead of the SWU keyword.

Example

```
--
--       DEFINE GRID BLOCK END-POINT SWU DATA FOR ALL CELLS
--       (FOR NX x NY x NZ = 300)
--
SWU
         300*0.700                                                    /
```

The above example defines a constant connate gas saturation of 0.70 to all 300 cells in the model as defined by the DIMENS keyword in the RUNSPEC section.

8.3.337 TEMPNODE - TEMPERATURE TABLE FOR POLYMER SOLUTION VISCOSITY

Description

This keyword defines the reservoir temperature table used to calculate the polymer solution viscosity when the temperature option has been activated by the TEMP keyword in the RUNSPEC section in the commercial simulator. Naturally, the polymer option must also be activated by the POLYMER keyword in the RUNSPEC section in order to use this keyword.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.338 TEMPTVD – ACTIVATE TEMPERATURE FLUX LIMITED TRANSPORT OPTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

TEMPTVD activates the Temperature Flux Limited Transport option in the commercial simulator, to reduce numerical dispersion for when either the TEMP or THERMAL keywords in the RUNSPEC section have been declared.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.339 TEMPVD - DEFINE THE INITIAL RESERVOIR TEMPERATURE VERSUS DEPTH TABLE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword defines the initial reservoir temperature versus depth tables for each equilibration region. Note that the TEMPVD keyword is an alias for RTEMPVD, and that both keywords are supported by OPM Flow, in both the PROPS and SOLUTION sections, but are treated as being mutually exclusive.

See *TEMPVD - Define the Initial Reservoir Temperature versus Depth Tables* in the SOLUTION section, as the keyword is documented in the SOLUTION section in this instance, but it can also be used in the PROPS section by OPM Flow.

8.3.340 TLMIXPAR – DEFINE THE MISCIBLE TODD-LONGSTAFF MIXING PARAMETERS

Description

The TLMIXPAR keyword defines the Todd-Longstaff[263] mixing parameters, for when either the miscible or solvent options have been activated by the MISCIBLE or SOLVENT keywords in the RUNSPEC section. This keyword must be present in the input deck if the MISCIBLE or SOLVENT keywords have been activated.

Note that If the POLYMER option has been activated by the POLYMER keyword in the RUNSPEC section, then this keyword is ignored and the mixing parameters are taken from the PLMIXPAR keyword instead.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	TLMVIS	A real positive value that is greater than or equal to zero and less than or equal to one, that defines the viscosity Todd-Longstaff mixing parameter for each miscibility region.			
		dimensionless	dimensionless	dimensionless	None
2	TLMDEN	A real positive value that is greater than or equal to zero and less than or equal to one, that defines the density Todd-Longstaff mixing parameter for each miscibility region.			The same value as entered for TLMVIS
		dimensionless	dimensionless	dimensionless	

Notes:

1) The keyword is followed by NTMISC tables as declared on the MISCIBLE keyword in the RUNSPEC section, for each rock region. There should be only one row per table.

2) Each entry is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.192: TLMIXPAR Keyword Description

Example

```
--
--       TODD-LONGSTAFF MIXING PARAMETERS
--
TLMIXPAR
--       TLM         TLM
--       VISCOS      DENSITY
--       -------     --------
         0.3500       0.3500                          / TABLE NO. 01
         0.2500       1*                              / TABLE NO. 02
         0.6500       0.7500                          / TABLE NO. 03
```

The above example defines three Todd-Longstaff mixing parameter data sets, based on the NTMISC variable on the MISCIBLE keyword in the RUNSPEC section being equal to three.

[263] *Todd, M. and Longstaff, W. "The Development, Testing and Application of a Numerical Simulator for Predicting Miscible Flood Performance," paper SPE 3484, Journal of Canadian Petroleum Technology (1972) 24, No. 7, 874-882.*

8.3.341 TOLCRIT – DEFINE THE CRITICAL SATURATION TOLERANCE

Description

Critical fluid saturations are determined from the relative permeability tables, that is the last saturation in a relative permeability table where the relative permeability of a phase is set equal to zero. Since floating-point numbers (as implemented in computers) are never exact, one cannot compare floating point numbers for exact equality. Thus, this keyword defines a value below which is considered equivalent to zero in determining the critical saturation for a phase.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	TOLCRIT	TOLCRIT is a real positive number greater than zero and less than one that defines the critical saturation tolerance used to determine the critical saturation of a fluid in the relative permeability tables. The default value of 1 x 10-6 means that saturation values less than this value will be treated as being equal to zero.			
		dimensionless	dimensionless	dimensionless	1 x 10^{-6}
Notes: 1) The keyword is terminated by a "/".					

Table 8.193: TOLCRIT Keyword Description

See also section *8.2.4 Saturation Tables (Relative Permeability and Capillary Pressure Tables)* for a description of the relative permeability tables and the various end-point definitions, including oil, water and gas critical saturations.

Example

```
---
--       SET THE CRITICAL SATURATION TOLERANCE
--
TOLCRIT
         1.0E-6                                                    /
```

The above example defines the critical saturation tolerance to be the default value of 1 x 10-6.

8.3.342 TPAMEPS – VOLUMETRIC STRAIN VERSUS COAL GAS CONCENTRATION TABLES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

TPAMEPS defines the volumetric strain versus coal gas concentration tables, for when the Coal Bed Methane option has been activated via the COAL keyword, and PALM-MAN has been declared for the ROCKOPT variable on the ROCKCOMP keyword; both keywords are in the RUNSPEC section. The Palmer-Mansoorii[264] and [265] rock model is used to calculate the impact on pore volume and permeability due to rock compaction.

See also the ROCKPARMA keyword in the PROPS section that defines the Palmer-Mansoori parameters.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

[264] *Palmer, I. and Mansoori, J. "How Permeability Depends on Stress and Pore Pressure in Coalbeds: A New Model," paper SPE 52607, SPE Reservoir Evaluation & Engineering (1998) 1, No. 6, 539-544.*

[265] *Clarkson, C.R., Pan, Z., Palmer, I. and Harpalani, S. "Predicting Sorption-Induced Strain and Permeability Increase With Depletion for Coalbed-Methane Reservoirs", SPE 114778-PA, SPE Journal (2010) 15, No. 1, 152–159.*

8.3.343 TPAMEPSS - VOLUMETRIC STRAIN VERSUS COAL SOLVENT CONCENTRATION TABLES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

TPAMEPSS defines the volumetric strain versus coal solvent concentration tables, for when the Coal Bed Methane option has been activated via the COAL keyword, and PALM-MAN has been declared for the ROCKOPT variable on the ROCKCOMP keyword; both keywords are in the RUNSPEC section. The Palmer-Mansoorii[266] and [267] rock model is used to calculate the impact on pore volume and permeability due to rock compaction.

See also the ROCKPARMA keyword in the PROPS section that defines the Palmer-Mansoori parameters.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

[266] *Palmer, I. and Mansoori, J. "How Permeability Depends on Stress and Pore Pressure in Coalbeds: A New Model," paper SPE 52607, SPE Reservoir Evaluation & Engineering (1998) 1, No. 6, 539-544.*

[267] *Clarkson, C.R., Pan, Z., Palmer, I. and Harpalani, S. "Predicting Sorption-Induced Strain and Permeability Increase With Depletion for Coalbed-Methane Reservoirs", SPE 114778-PA, SPE Journal (2010) 15, No. 1, 152–159.*

8.3.344 TRACER – DEFINE PASSIVE TRACER VARIABLES

Description

The TRACER keyword defines a series of passive tracers that are associated with a phase (oil, water, or gas) in the model. The maximum number of tracers for each phase are declared on the TRACERS keyword in the RUNSPEC section.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	NAME	A three letter character string defining the tracer's name. NAME is used by the TNUM keyword in the REGIONS section, and unlike other keywords, the TNUM keyword itself must be concatenated with the phase and the name of the tracer defined by NAME. Similarly for the TVDP keyword in the SOLUTION section, where the TVDP keyword itself must be concatenated with the either letter F (for free) or S (for solution), followed by the name of the tracer defined by NAME. Note it is best to void names beginning with the letters F, S, and T as these names may create naming issues in post-processing software.			None
2	PHASE	A three letter character string that defines the tracer given by NAME to a particular fluid phase. The character should be set to OIL, WAT or GAS.			None
3	UNITS	The units for the tracer. This can be any unit but should be consistent with values entered via the TBLK and TVDP keywords in the SOLUTION section. The default values are the same as the PHASE in the model.			Same as the phases in the model
		Liquid: stb Gas: Mscf	Liquid: sm^3 Gas: sm^3	Liquid: scc Gas: scc	
4	SOLPHASE	A three or four letter character string defining the partitioned tracer's solution phase. The character string should be set to OIL, WAT, GAS or MULT. Note that SOLPHASE only needs to be defined if the partitioned tracer option has been activate with the PARTTRAC keyword in the RUNSPEC section.			None
5	KPNUM	The table number to be used with the partitioned tracers defined by the PARTTRAC, TRACERKP and TRACERKM keywords. Note that KPNUM only needs to be defined if the partitioned tracer option has been activate with the PARTTRAC keyword in the RUNSPEC section.			None
6	ADSPHASE	A three letter character string defining the phase used for the adsorption calculation for when the MULT option has been for SOLPHASE. The character string should be set to OIL, WAT, GAS or ALL. Note that ADSPHASE only needs to be defined if the partitioned tracer option has been activate with the PARTTRAC keyword in the RUNSPEC section.			None

No.	Name	Description			Default
		Field	Metric	Laboratory	
Notes: 1) Each record (or row) should be terminated by a "/" and the keyword is terminated by a "/".					

Table 8.194: TRACER Keyword Description

See also the TNUM keyword in the REGION section that defines tracer regions, and the TVDP keyword in the SOLUTION section that sets the initial tracer saturation for all the cells as a function of depth.

Example

```
--
--       DEFINE TRACER NAMES
--
--       TRACER   TRACER
--       NAME     PHASE
--       ------   ------
TRACER
        'IGS'     'GAS'                                  / GAS INJECTOR
        'DGS'     'GAS'                                  / DISOLVED GAS
        'IW1'     'WAT'                                  / WAT INJECTOR 1
        'Iw2'     'WAT'                                  / WAT INJECTOR 2
/
```

The above example defines four passive tracers one for a gas injection well, one for tracking the dissolved gas, and two to track the injected water from two water injection wells.

8.3.345 TRACERKM – MULTI-PARTITIONED TRACER OPTION K(P) TABLES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, TRACERKM, defines the Multi-Partitioned Tracer option K(P) tables, for when the Partitioned Tracer option has been activate with the PARTTRAC keyword in the RUNSPEC section, and the SOLPHASE parameter on the TRACER keyword in the PROPS section has been set to MULT to activate the Multi-Partitioned Tracer option. Multi-partitioned tracers can partition into any number of phases (oil, water, gas etc.) and have adsorption, decay and diffusion parameters specific to each phase; whereas the standard partitioned tracers only have a "free" and "solution" phases. For the TRACERKM keyword the K(P) tables relate the ratio of the reference phase to the other phases versus pressure. So for example, given a multi-partitioned tracer in oil, water and gas, with the water phase acting as the reference phase, then TRACERKM would consist of columnar vectors of:

$$K_{ow}(P) = \frac{C_{oil}}{C_{water}} \quad \text{and} \quad K_{gw}(P) = \frac{C_{gas}}{C_{water}} \tag{8.96}$$

Where:

$Kow(P)$	= multi-partitioned oil-water K(P)
$Kgw(P)$	= multi-partitioned gas-water K(P)
$Coil$	= oil concentration
$Cgas$	= gas concentration
$Cwater$	= water concentration

See also the TRACERKP keyword in the PROPS section that provides similar data for standard partitioned tracers.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.346 TRACERKP – STANDARD PARTITIONED TRACER OPTION K(P) TABLES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, TRACERKP, defines the Standard Partitioned Tracer option K(P) tables, for when the Partitioned Tracer option has been activate with the PARTTRAC keyword in the RUNSPEC section. Standard partitioned tracers only have a "free" and "solution" phases; whereas, Multi-partitioned tracers can partition into any number of phases (oil, water, gas etc.) and have adsorption, decay and diffusion parameters specific to each phase. For the TRACERKP keyword the K(P) tables relate the ratio of the reference phase (the "free" phase) to the solution phase versus pressure. So for example, given a standard partitioned tracer in oil and gas, with the oil phase acting as the reference phase, then TRACERKP would consist of columnar vectors of:

$$K(P) = \frac{C_{gas}}{C_{oil}} \tag{8.97}$$

Where:

$K(P)$ = standard partitioned K(P)
$Coil$ = oil concentration
$Cgas$ = gas concentration

See also the TRACERKM keyword in the PROPS section that provides similar data for tmulti-partitioned tracers.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.347 TRACITVD – ACTIVATE AND DEFINE TRACER IMPLICIT FLUX LIMITED TRANSPORT OPTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

TRACITVD activates the Tracer Implicit Flux Limited Transport option and sets various parameters for this option. Basically the option is used to control numerical dispersion for tracers. Both the TRACERS keyword in the RUNSPEC section and the TRACER keyword in the PROPS section must be declared to activate tracers and to define the tracers.

See also the TRACTVD keyword in the PROPS section activates the Tracer Explicit Flux Limited Transport option.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.348 TRACTVD – ACTIVATE TRACER EXPLICIT FLUX LIMITED TRANSPORT OPTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

TRACTVD activates the Tracer Explicit Flux Limited Transport option. Basically the option is used to control numerical dispersion for tracers. Both the TRACERS keyword in the RUNSPEC section and the TRACER keyword in the PROPS section must be declared to activate tracers and to define the tracers.

See also the TRACITVD keyword in the PROPS section activates the Tracer Implicit Flux Limited Transport option.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.349 TRADS – ENVIRONMENTAL TRACER ADSORPTION TABLES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, TRADS, specifies the environmental tracer adsorption tables that describe how a tracer is absorbed by the surrounding rock, for when the MXENVTR parameter has been set to greater than zero on the TRACERS keyword in the RUNSPEC section to activate environmental tracers. The keyword can only be used with environmental tracers.

Unlike other keywords, the TRADS keyword must be concatenated with the three character name of the tracer declared by TRACER keyword in the PROPS section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.350 TRDCY – ENVIRONMENTAL TRACER DECAY TABLES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, TRDCY, specifies the environmental tracer decay tables that specifies the tracer decay half-life, for when the MXENVTR parameter has been set to greater than zero on the TRACERS keyword in the RUNSPEC section to activate environmental tracers. The keyword can only be used with environmental tracers.

Unlike other keywords, the TRSCY keyword must be concatenated with the three character name of the tracer declared by TRACER keyword in the PROPS section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.351 TRDIF – TRACER DIFFUSION TABLES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, TRDIF, specifies the tracer diffusion tables that specify the diffusion coefficient for a tracer. The keyword can be used with Environmental Tracers if the MXENVTR parameter has been set greater than zero on the TRACERS keyword in the RUNSPEC section. When used with a Standard Partitioned Tracer the diffusion coefficient applies to the solution phase, whereas as for a Multi-Partitioned Tracer the diffusion coefficient can be entered for each defined tracer phase. Unlike other keywords, the TRADS keyword must be concatenated with the three character name of the tracer declared by TRACER keyword in the PROPS section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.352 TRDIS – TRACER DISPERSION TABLE NUMBER ALLOCATION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, TRDIS, specifies the tracer diffusion tables that should be allocated to a tracer, the actual dispersion tables are specified by the DISPERSE keyword in the PROPS section. The keyword can be used with Environmental Tracers if the MXENVTR parameter has been set greater than zero on the TRACERS keyword in the RUNSPEC section. The option does not work with two-phase Standard Partitioned Tracers and Multi-Partitioned Tracers. Unlike other keywords, the TRADS keyword must be concatenated with the three character name of the tracer declared by TRACER keyword in the PROPS section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.353 TRNHD – ACTIVATE DISPERSION NON-HOMOGENEOUS DIFFUSION OPTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The TRNHD keyword activates the Dispersion Non-Homogeneous Diffusion option for when tracer dispersion is independent of velocity or tracer concentration. Unlike other keywords, the TRNHD keyword must be concatenated with the name of the tracer declared by TRACER keyword in the PROPS section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.354 TRROCK – ENVIRONMENTAL TRACER-ROCK PROPERTY DATA

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, TRROCK, specifies the environmental tracer rock data for the tracer adsorption model, for when the MXENVTR parameter has been set to greater than zero on the TRACERS keyword in the RUNSPEC section to activate environmental tracers. The keyword can only be used with environmental tracers.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.355 TZONE – END-POINT SCALING TRANSITION ZONE OPTIONS

Description

The TZONE keyword sets the transition end-point scaling options for the oil, water and gas phases, for when the End-Point Scaling option has been activated by the ENDSCALE keyword in the RUNSPEC section. The keyword determines if the phase critical saturation should or should not be set to the initial immobile saturation in areas where the initial saturation is below the entered critical saturation.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description	Default
1	OILZONE	OILZONE is a single character that sets the oil phase transition zone end-point scaling option and should be set to either T or F: 1) T: for true, results in the SOWCR values being adjusted to the initial immobile saturation for oil-water or oil-water-miscible gas simulations. For oil-gas simulations the SOGCR values are modified to be the initial immobile saturation. The modifications only occur in cells where the initial saturation is below the entered critical saturation. 2) F: for false, means the critical saturations are not modified.	F
2	WATZONE	WATZONE is a single character that sets the water phase transition zone end-point scaling option and should be set to either T or F: 1) T: for true, results in the SWCR values being adjusted to the initial immobile saturations. The modifications only occur in cells where the initial saturation is below the entered critical saturation values (SWCR). 2) F: for false, means the critical saturations are not modified.	F
3	GASZONE	GASZONE is a single character that sets the gas phase transition zone end-point scaling option and should be set to either T or F: 1) T: for true, results in the SGCR values being adjusted to the initial immobile saturation for oil-gas or gas-water simulations. The modifications only occur in cells where the initial saturation is below the entered critical saturation (SGCR). 2) F: for false, means the critical saturations are not modified.	F
Notes: 1) The keyword is terminated by a "/".			

Table 8.195: TZONE Keyword Description

See also the SCALECRS keyword in the PROPS section that sets the end-point scaling option to be either two-point or three-point scaling.

Example

```
--
--       END-POINT SCALING TRANSITION ZONE OPTIONS
--
--       OILZONE  WATZONE  GASZONE
--       -------  -------  -------
TZONE
         F        T        F                                / SCALING OPTION
```

The above example results in the SWCR values being adjusted to the initial immobile saturations.

8.3.356 VDFLOW – VELOCITY DEPENDENT FLOW COEFFICIENT FOR GRID BLOCK FLOW (GRID)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

VDFLOW activates non-Darcy flow between grid blocks and defines a constant non-Darcy flow coefficient for the whole grid, the coefficient only applies to the gas phase. The coefficient is normally derived from well tests or calculated analytically based on the coefficient of inertial resistance, usually known as β, in Forchheimer's flow equation,[268], [269] and [270]. Dake[271], in chapter eight, reports a typical value of β to be 10.07 cm-1.

See also the VDFLOWR keyword in the PROPS section that allows the non-Darcy coefficient to be entered for individual regions, and the WDFAC and WDFACCOR keywords in the SCHEDULE section that assigns the non-Darcy coefficient to well connections.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

[268] *Geertsma, J., 1974. Estimating the Coefficient of Inertial Resistance in Fluid Flow Through Porous Media. Soc.Pet.Eng.J., October: 445-450.*

[269] *Gewers, C.W.W. and Nichol, L.R., 1969. Gas Turbulence Factor in a Microvugular Carbonate. J.Can.Pet.Tech., April.*

[270] *Wong, S.W., 1970. Effects of Liquid Saturation on Turbulence Factors for Gas Liquid Systems. J.Can.Pet.Tech., October*

[271] *Dake, L.P. Fundamentals of Reservoir Engineering, Amsterdam, The Netherlands, Elsevier Science BV (1978) Chapter 8.6, pages 252-257.*

8.3.357 VDFLOWR – VELOCITY DEPENDENT FLOW COEFFICIENT FOR GRID BLOCK FLOW (REGION)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

VDFLOW activates non-Darcy flow between grid blocks and defines a constant non-Darcy flow coefficient for individual regions allocated by the SATNUM keyword in the REGIONS section. Note that the coefficient only applies to the gas phase. The coefficient is normally derived from well tests or calculated analytically based on the coefficient of inertial resistance, usually known as β, in Forchheimer's flow equation,[272], [273] and [274]. Dake[275], in chapter eight, reports a typical value of β to be 10.07 cm-1.

See also the VDFLOW keyword in the PROPS section that allows the non-Darcy coefficient to be entered for the whole grid, and the WDFAC and WDFACCOR keywords in the SCHEDULE section that assigns the non-Darcy coefficient to well connections.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

[272] *Geertsma, J., 1974. Estimating the Coefficient of Inertial Resistance in Fluid Flow Through Porous Media. Soc.Pet.Eng.J., October: 445-450.*

[273] *Gewers, C.W.W. and Nichol, L.R., 1969. Gas Turbulence Factor in a Microvugular Carbonate. J.Can.Pet.Tech., April.*

[274] *Wong, S.W., 1970. Effects of Liquid Saturation on Turbulence Factors for Gas Liquid Systems. J.Can.Pet.Tech., October*

[275] *Dake, L.P. Fundamentals of Reservoir Engineering, Amsterdam, The Netherlands, Elsevier Science BV (1978) Chapter 8.6, pages 252-257.*

8.3.358 VEFRAC – VERTICAL EQUILIBRIUM RELATIVE PERMEABILITY FRACTION (GRID)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword defines the Vertical Equilibrium ("VE") relative permeability weighting factor (α) used to calculate the VE relative permeability curves to be used in the simulation, for when the VE option has been activated by the VE keyword in the RUNSPEC section. If α = 1.0, then the VE model calculated relative permeability curves will be used, and if α = 0.0, then the curves entered via the SWOF, SGOF, SLGOF series of keywords or the SWFN, SGFN, SGWFN, SOF2, SOF3, SOF32D series of keywords, will be used. A value of α between zero and one will result in weighted average relative permeability curves being employed, that is:

$$VE_{(average)} = (1.0 - \alpha) \times (SATNUM_{curves}) + \alpha \times (VE\ Model_{curves}) \quad (8.98)$$

Note that VEFRAC sets α for the whole grid; whereas, the VEFRACV keyword in the PROPS section assigns α on a cell by cell basis, See also the VEFRACP and VEFRACPV keywords that apply the weighting factors to the capillary pressure data.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.359 VEFRACP – VERTICAL EQUILIBRIUM CAPILLARY PRESSURE FRACTION (GRID)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword defines the Vertical Equilibrium ("VE") capillary pressure weighting factor (α) used to calculate the VE capillary pressure curves to be used in the simulation, for when the VE option has been activated by the VE keyword in the RUNSPEC section. If α = 1.0, then the VE model calculated capillary pressure curves will be used, and if α = 0.0, then the curves entered via the SWOF, SGOF, SLGOF series of keywords or the SWFN, SGFN, SGWFN, SOF2, SOF3, SOF32D series of keywords, will be used. A value of α between zero and one will result in weighted average capillary pressure curves being employed, that is:

$$VE_{(average)} = (1.0 - \alpha) \times (SATNUM_{curves}) + \alpha \times (VE\ Model_{curves}) \quad (8.99)$$

Note that VEFRACP sets α for the whole grid; whereas, the VEFRACPV keyword in the PROPS section assigns α on a cell by cell basis, See also the VEFRAC and VEFRACV keywords that apply the weighting factors to the relative permeability data.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.360 VEFRACPV – VERTICAL EQUILIBRIUM CAPILLARY PRESSURE FRACTION (CELL)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword defines the Vertical Equilibrium ("VE") capillary pressure weighting factor (α) used to calculate the VE capillary pressure curves to be used in the simulation, for when the VE option has been activated by the VE keyword in the RUNSPEC section. If α = 1.0, then the VE model calculated capillary pressure curves will be used, and if α = 0.0, then the curves entered via the SWOF, SGOF, SLGOF series of keywords or the SWFN, SGFN, SGWFN, SOF2, SOF3, SOF32D series of keywords, will be used. A value of α between zero and one will result in weighted average capillary pressure curves being employed, that is:

$$VE_{(average)} = (1.0 - \alpha) \times (SATNUM_{curves}) + \alpha \times (VE\ Model_{curves}) \quad (8.100)$$

Note that VEFRACPV sets α on a cell by cell basis; whereas, the VEFRACP keyword in the PROPS section assigns α for the whole grid, See also the VEFRAC and VEFRACV keywords that apply the weighting factors to the relative permeability data.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.361 VEFRACV – VERTICAL EQUILIBRIUM RELATIVE PERMEABILITY FRACTION (CELL)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword defines the Vertical Equilibrium ("VE") relative permeability weighting factor (α) used to calculate the VE relative permeability curves to be used in the simulation, for when the VE option has been activated by the VE keyword in the RUNSPEC section. If α = 1.0, then the VE model calculated relative permeability curves will be used, and if α = 0.0, then the curves entered via the SWOF, SGOF, SLGOF series of keywords or the SWFN, SGFN, SGWFN, SOF2, SOF3, SOF32D series of keywords, will be used. A value of α between zero and one will result in weighted average relative permeability curves being employed, that is:

$$VE_{(average)} = (1.0 - \alpha) \times (SATNUM_{curves}) + \alpha \times (VE\ Model_{curves}) \quad (8.101)$$

Note that VEFRACVV sets α on a cell by cell basis; whereas, the VEFRAC keyword in the PROPS section assigns α for the whole grid, See also the VEFRACP and VEFRACPV keywords that apply the weighting factors to the capillary pressure data.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

8.3.362 VISCREF - DEFINE VISCOSITY-TEMPERATURE REFERENCE CONDITIONS

Description

VISCREF defines the reference conditions for the viscosity-temperature tables, GASVISCT, OILVISCT and WATVISCT, for when the thermal option has been activated by THERMAL keyword in the RUNSPEC section. This keyword can only be used if the thermal option has been activated by the THERMAL keyword in the RUNSPEC section. Note this is different to the commercial simulator that uses the TEMP keyword in the RUNSPEC section to activate the black-oil thermal model.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	PRES	PRES is a real positive number defining the reference pressure for the viscosity and temperature tables			
		psia	barsa	atma	None
2	RS	RS is a real positive number defining the reference gas-oil ratio for when the model contains gas dissolved as activated by the DISGAS keyword in the RUNSPEC section			
		Mscf/stb	sm^3/sm^3	scc/scc	None
3	API	API is a real number defining the oil API for when the API tracking option has been invoked by the API keyword in the RUNSPEC section. Note that OPM Flow does not support API tracking, and therefore this variable is ignored.			
		°API	°API	°API	None

Notes:

1) The keyword is followed by NTPVT tables as declared on the TABDIMS keyword in the RUNSPEC section.
2) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.196: VISCREF Keyword Description

OPM Flow currently does not support API tracking and therefore item (3) of this keyword is ignored. See also the OILVISCT, GASVISCT and WATVISCT keywords in the PROPS section.

Example

The following example shows the VISCREF keyword for when the thermal option has been activated by the THERMAL keyword in the RUNSPEC section and for when NTPVT on the TABDIMS keyword in the RUNSPEC section is set to five.

```
--
--       REF         REF        REF
--       PRESSURE    GOR        API
--       --------    -------    -------
VISCREF
          3000.0     0.500                                  / TABLE NO. 01
          3200.0     0.550                                  / TABLE NO. 02
          3300.0     0.580                                  / TABLE NO. 03
          3400.0     0.620                                  / TABLE NO. 04
          3500.0     0.625                                  / TABLE NO. 05
```

There is no terminating "/" for this keyword.

8.3.363 WAGHYSTR – DEFINE WATER-ALTERNATING-GAS HYSTERESIS PARAMETERS

Description

This keyword defines the parameters for the Water-Alternating-Gas ("WAG") hysteresis option, for when the hysteresis option has been activated by the WAGHYSTR variable on the SATOPTS keyword in the RUNSPEC section.

The WAG recovery mechanism is an Enhanced Oil Recovery ("EOR") process to optimize oil recovery by improving volumetric sweep efficiency. It was originally proposed as a method to improve the sweep efficiency of gas by using water to control the mobility ratio and to stabilize the front (Caudle and Dyes, 1958[276]; Christensen et al., 1998[277]; and Christensen et al., 2001[278]). WAG injection can lead to improved oil recovery by combining better mobility control and contacting upswept zones, and by leading to improved microscopic displacement. Although initially the injected gas was immiscible with respect to the oil (WAG Immiscible) the more common process is WAG Miscible, with alternating different types of hydrocarbon gases and non-hydrocarbon gases, such as N2 and CO2 Gases. WAG flooding has been successfully applied to various fields worldwide.

Only the gas phase relative permeability WAG hysteresis model has been implemented in the simulator.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	LANDS_PARAMETER	A real value greater than zero that defines Land's parameter, C. The Land's parameter controls how the trapped gas saturation depends on the maximum gas saturation attained and the critical gas saturation; and the shape of the imbibition curve. $$s_{gtrap} = s_{gcr} + \frac{(s_{gm} - s_{gcr})}{(1 + C(s_{gm} - s_{gcr}))}$$ where, s_{gtrap} is the trapped gas saturation, s_{gm} is the maximum gas saturation attained, and s_{gcr} is the critical gas saturation. Values of the Land's parameter that are too small give a trapped gas saturation close to the maximum gas saturation attained. This results in an unphysical steep relative permeability curve giving potential convergence problems.			None
		dimensionless	dimensionless	dimensionless	
2	SECONDARY_DRAINAGE_REDUCTION	A real value greater than or equal to zero that defines the secondary drainage reduction factor, alpha. As alpha increases the reduction in gas mobility on secondary drainage increases.			0.0

[276] *Caudle, B. H., & Dyes, A. B. (1958, January 1). Improving Miscible Displacement by Gas-Water Injection. Society of Petroleum Engineers.*

[277] *Christensen, J. R., Stenby, E. H., & Skauge, A. (1998, January 1). Review of WAG Field Experience. Society of Petroleum Engineers. doi:10.2118/39883-MS.*

[278] *Christensen, J. R., Stenby, E. H., & Skauge, A. (2001, April 1). Review of WAG Field Experience. Society of Petroleum Engineers. doi:10.2118/71203-PA.*

No.	Name	Description			Default
		Field	Metric	Laboratory	
		dimensionless	dimensionless	dimensionless	
3	GAS_MODEL	A defined character string that defines whether the gas hysteresis model should be used, and should be set to one of the following character strings: 1) YES: Use the WAG hysteresis model for the gas phase relative permeability. 2) NO: Turn off the WAG hysteresis model and use the drainage curve. Only the YES option is currently supported by the simulator.			YES
4	RES_OIL	A defined character string that defines whether the residual oil model should be used, and should be set to one of the following character strings: 1) YES: Use the trapped gas to modify the residual oil saturation (SOM) in the STONE I three-phase oil relative permeability model. No action is taken unless the STONE1 keyword has been entered. 2) NO: Do not modify the residual oil saturation. Only the NO option is currently supported by the simulator.			YES
5	WATER_MODEL	A defined character string that defines whether the water hysteresis model should be used, and should be set to one of the following character strings: 1) YES: Use the WAG hysteresis model for the water phase relative permeability 2) NO: Turn off the WAG hysteresis model. Note that the hysteresis model specified in EHYSTR keyword applies. Only the NO option is currently supported by the simulator.			YES
6	IMB_LINEAR_FRACTION	A real value greater than zero that defines the imbibtion curve linear fraction. This is the fraction of the curve between Sgm and Sgtrap that uses a linear transformation. A non-zero value for the linear fraction prevents the potential infinite gradient in the imbibition curve when using the Carlson analytic model.			0.1
		dimensionless	dimensionless	dimensionless	
7	THREEPHASE_SAT_LIMIT	A real value between zero and one that defines the three-phase model threshold saturation. When the water saturation exceeds this threshold above the connate water saturation the gas (non-wetting) phase hysteresis switches from the two-phase model to the three-phase model. In the two-phase model a secondary drainage process follows the imbibition curve. However, if the water saturation exceeds the connate saturation by the given threshold, at the beginning of the secondary drainage process a three-phase secondary drainage curve is followed. This value also defines the minimum percentage change in gas saturation to allow switching from drainage to imbibition curve and vice-versa. This threshold allows better control of the numerical sensitivity of the system, preventing it from being too unstable.			0.001
		dimensionless	dimensionless	dimensionless	

No.	Name	Description			Default
		Field	Metric	Laboratory	
8	RES_OIL_MOD_FRACTION	A real value between zero and one that defines the residual oil modification fraction. This is the fraction of the trapped gas saturation subtracted from the residual oil (SOM) in the STONE I three-phase oil relative permeability model. This is not supported and will be ignored by the simulator.			1.0
		dimensionless	dimensionless	dimensionless	

Notes:

1) The keyword is followed by one record for each saturation table region with each record terminated by a "/".
2) The number of saturation table regions NTSFUN is specified by the TABDIMS keyword in the RUNSPEC section.

Table 8.197: WAGHYSTR Keyword Description

Example

The following example defines the WAG hysteresis model parameters using the WAGHYSTR keyword for a case with three saturation table regions

```
--
--        WAG HYSTERESIS PARAMETERS
--
-- LAND   ALPHA     GAS     RES     WATER   LINEAR
-- C      FACTOR    MODEL   OIL     MODEL   FRAC
WAGHYSTR
2.0       1.0       YES     YES     YES     0.2     /
2.0       1.0       YES     NO      /
2.0       /
```

Here, saturation table region one uses the gas WAG hysteresis, residual oil and water WAG hysteresis models, with a Land's parameter of 2.0, an alpha factor of 1.0, and a imbibition curve linear factor of 0.2. The gas and water WAG hysteresis models are used in region two but the residual oil model is turned off, with a Land's parameter of 2.0, an alpha factor of 1.0, and a default imbibition curve linear factor of 0.1. A Land's parameter of 2.0 has been specified for region three with the remainder of the parameters defaulted.

Note that the above example uses some options that are not currently supported by OPM Flow.

8.3.364 WARN – ACTIVATE WARNING MESSAGES

Turns on warning messages to be printed to the print file (*.PRT); note that this keyword is activated by default and can subsequently be switched off by the NOWARN activation keyword. The warning messages may be turned on and off using keywords WARN and NOWARN. OPM Flow always prints error messages.

See *WARN – Activate Warning Messages* in the GLOBAL section for a full description.

8.3.365 WATDENT – DEFINE WATER DENSITY TEMPERATURE COEFFICIENTS

Description

WATDENT defines the water density as a function of temperature coefficients for when the thermal option has been activated by the THERMAL keyword in the RUNSPEC. Note this is an OPM Flow keyword used with OPM Flow's black-oil thermal model that is not available in the commercial simulator's black-oil thermal formulation.

This keyword can only be used if OPM Flow's thermal option has been activated by the THERMAL keyword in the RUNSPEC section. Note this is different to the commercial simulator that uses the TEMP keyword in the RUNSPEC section to activate the black-oil thermal model.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	TEMP	TEMP is a real positive value greater than zero that defines the absolute reference temperature used with TEXP1 and TEXP2 to estimate the change in water density with respect to temperature.			
		°R 527.67	K 293.15	K 293.15	Defined
2	TEXP1	TEXP1 is a real positive value greater than zero that defines the water thermal expansion coefficient of the first order.			
		1/°R 1.67×10^{-4}	1/K 3.0×10^{-4}	1/K 3.0×10^{-4}	Defined
3	TEXP2	TEXP2 is a real positive value greater than zero that defines the water thermal expansion coefficient of the second order.			
		1/°R2 9.26×10^{-7}	1/K2 3.0×10^{-6}	1/K2 3.0×10^{-6}	Defined

Notes:

1) The keyword is followed by NTPVT records as declared on the TABDIMS keyword in the RUNSPEC section.
2) Each data set is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.198: WATDENT Keyword Description

The water density at a given pressure and temperature is calculated from its value at surface conditions and the water shrinkage factor (the reciprocal of the water formation volume factor) as shown in the following equation:

$$\rho_w(p,T) = \rho_w(p_s,T_s)\, b_w(p,T) \tag{8.102}$$

Where the temperature dependence of the water shrinkage factor relative to its value at the reference temperature is calculated as shown in the following equation:

$$b_w(p,T)=\frac{b_w(p,T_{ref})}{1+c_1(T-T_{ref})+c_2(T-T_{ref})^2} \quad (8.103)$$

Where:

ρ_w = water density
b_w = water shrinkage factor
p = pressure
T = temperature
c_1, c_2 = thermal expansion coefficients to first and second order
s = subscript indicating surface conditions
ref = subscript indicating reference conditions

Example

The following example shows the WATDENT keyword using the default values, for when the thermal option has been activated by the THERMAL keyword in the RUNSPEC section and for when NTPVT on the TABDIMS keyword in the RUNSPEC section is set to two.

```
--
--       WATER DENSITY TEMPERATURE COEFFICIENTS (OPM FLOW EXTENSION KEYWORD)
--
--       WATER      DENSITY   DENSITY
--       TEMP       COEFF1    COEFF2
--       --------   -------   -------
WATDENT
         1*         1*        1*                          / TABLE NO. 01
         1*         1*        1*                          / TABLE NO. 02
```

There is no terminating "/" for this keyword.

8.3.366 WATJT – DEFINE WATER JOULE-THOMSON COEFFICIENT

Description

WATJT activates the water Joule-Thomson effect[279] in temperature calculations, and defines the water Joule-Thomson Coefficient ("JTC") at a given reference pressure, for when OPM Flow's thermal option has been activated by the THERMAL keyword in the RUNSPEC.

Note

This is an OPM Flow keyword used with OPM Flow's black-oil thermal model, that is not available in the commercial simulator's black-oil thermal formulation.

This keyword can only be used if OPM Flow's thermal option has been activated by the THERMAL keyword in the RUNSPEC section. Note this is different to the commercial simulator that uses the TEMP keyword in the RUNSPEC section to activate the black-oil thermal model, and does not include the Joule-Thomson effect in temperature calculations.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	PRESS	A real positive value that defines the reference pressure for the corresponding Joule-Thomson Coefficient, WATJTC.			
		psia	barsa	atma	None
2	WATJTC	WATJTC is a real positive or negative value that defines the water phase Joule-Thomson Coefficient. If the value is defaulted (1*) or set to 0, then WATJTC is internally calculated using thermal water density data on the WATSDENT keyword in the PROPS section. If a non-zero value is specified, then the WATJTC is assumed to be constant and equal to that value.			
		oF/psia	°C/barsa	°C/atma	0

Notes:

1) The keyword is followed by NTPVT records as declared on the TABDIMS keyword in the RUNSPEC section.
2) Each data set is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.199: WATJT Keyword Description

The Joule–Thomson effect is when a real gas, as oppose to an ideal gas, expands, resulting in the temperature of the gas dropping[280]. For liquids the effect is the opposite, that is the internal energy is transferred to kinetic energy with a corresponding increase in temperature as velocity increases.

Thermodynamically, the Joule–Thomson coefficient is defined as the isenthalpic[281] change in temperature in a fluid caused by a unitary pressure drop, as shown in the following equation:

[279] *The Joule–Thomson coefficient is defined as the change in temperature with respect to an increase in pressure at constant enthalpy.*

[280] *Natural Gas Engineering (McGraw-Hill chemical engineering series), Donald L. Katz, Robert I Lee, McGraw-Hill Education, 1990 (ISBN 0071007776, 9780071007771).*

$$\eta = \left(\frac{\partial T}{\partial P}\right) \tag{8.104}$$

Which can also express as[282]:

$$\eta = (T\alpha - 1)\frac{1}{(\rho C_p)} - \left(\frac{g}{C_p}\frac{dp}{dz}\right)^{-1} \tag{8.105}$$

Setting the gravity term, g, to zero we have:

$$\eta = (T\alpha - 1)\frac{1}{(\rho C_b)} \tag{8.106}$$

Where:

η	=	Joule–Thomson coefficient (oC/Pa),
α	=	thermal expansivity at constant pressure (1/oC),
C_p	=	specific heat at constant pressure (J/kg oC),
g	=	gravitational acceleration (m/s2)
P	=	pressure (Pa),
T	=	temperature (oC), and
z	=	height (m).

Example

The following example shows the WATJT keyword for when the thermal option has been activated by the THERMAL keyword in the RUNSPEC section, and for when NTPVT on the TABDIMS keyword in the RUNSPEC section is set equal to two.

```
--
--       WATER JOULE-THOMSON COEFFICIENT (OPM FLOW EXTENSION KEYWORD)
--
--       REF         OIL
--       PRESS       JTC
--       --------    -------
WATJT
         20.0        1*                  / TABLE NO. 01
         20.0        -0.20               / TABLE NO. 02
```

Here the first entry is defaulted, and the simulator will therefore calculate the water JTC internally using the data on the WATDENT keyword in the PROPS section.

There is no terminating "/" for this keyword.

[281] *An isenthalpic process or isoenthalpic process, is a process that proceeds without any change in enthalpy, H; or specific enthalpy, h.*

[282] *Pippard, A.B.: Elements of Classical Thermodynamics: For Advanced Students of Physics. Cambridge University Press, Cambridge, UK (1957)*

8.3.367 WATVISCT – DEFINE WATER VISCOSITY VERSUS TEMPERATURE FUNCTIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

WATVISCT defines the water viscosity as a function of temperature for when thermal option has been activated by the THERMAL keywords in the RUNSPEC. The reference pressure for this table is given by the VISCREF keyword in the PROPS section.

This keyword can only be used if OPM Flow's thermal option has been activated by the THERMAL keyword in the RUNSPEC section. Note this is different to the commercial simulator that uses the TEMP keyword in the RUNSPEC section to activate the black-oil thermal model.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	TEMP	A columnar vector of real monotonically increasing down the column values that defines the temperature values.			
		°F	°C	°C	None
2	VIS	A columnar vector of real decreasing down the column values that defines the water viscosity for the corresponding temperature values (TEMP). VIS should be given at the reference pressure defined by the PRESS variable on the VISCREF keyword.			
		cP	cP	cP	None

Notes:

1) The keyword is followed by NTPVT tables as declared on the TABDIMS keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NPPVT rows as declared on the TABDIMS keyword in the RUNSPEC section.
3) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.200: WATVISCT Keyword Description

Example

The following example shows the WATVISCT keyword for when the thermal option has been activated by the THERMAL keyword in the RUNSPEC section and for when NTPVT on the TABDIMS keyword in the RUNSPEC section is set equal to one.

```
--
--       WATER VISCOSITY VERSUS TEMPERATURE TABLES
--
--       WATER      WATER
--       TEMP       VISC
--       --------   -------
WATVISCT
           100.0    0.625
           110.0    0.620
           120.0    0.580
           150.0    0.550
           165.0    0.500                                  / TABLE NO. 01
```

There is no terminating "/" for this keyword.

8.3.368 WSF - WATER SATURATION TABLES VERSUS WATER SATURATION (GAS-WATER AND CO2STORE SYSTEMS)

Description

The WSF keyword defines the water relative permeability data versus water saturation tables for when gas and water are only present in the input deck. This keyword should only be used if the gas and water phases are present in the run, and can therefore also be used with the CO2STORE model. In addition, the keyword must be used in conjunction with the GSF keyword in the PROPS section, that defines the gas relative permeability and gas-water capillary pressure data versus gas saturation for gas-water systems.

Note

WSF is a compositional keyword in the commercial compositional simulator, and will therefore cause an error in the commercial black-oil simulator.

Currently, both the GSF and WSF keywords can only be used with the CO2STORE model.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SWAT	A columnar vector of real monotonically increasing down the column values starting from zero and terminating at one, that defines the water saturation. The first entry is the connate water saturation Swc and the last entry should be 1.0.			
		dimensionless	dimensionless	dimensionless	None
2	KRW	A columnar vector of real values that are either equal or increasing down the column and that are greater than or equal to zero and less than or equal to one that defines the water relative permeability with respect to gas saturation. Note that the first entry in the column must be zero.			
		dimensionless	dimensionless	dimensionless	None

Notes:

1) The keyword is followed by NTSFUN tables as declared on the TABDIMS keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NSSFUN rows as declared on the TABDIMS keyword in the RUNSPEC section.
3) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 8.201: WSF Keyword Description

See also the *GSF - Gas Saturation Function Tables (Gas-Water Systems)* keyword in the PROPS section, that defines the gas relative permeability and gas-water capillary pressure data as a function of gas saturation, for when only the gas and water water phases are present in the model.

Example

```
--
--       WATER RELATIVE PERMEABILITY TABLES (OPM FLOW KEYWORD)
--
WSF
--       SWAT       KRW
--       FRAC
--       --------   --------
         0.200000   0.0000
         0.238616   0.0002
         0.245309   0.0004
         0.261989   0.0010
         0.303091   0.0044
         0.368269   0.0191
         0.435026   0.0519
         0.486387   0.0940
         0.550683   0.1725
         0.575342   0.2115
         0.599076   0.2542
         0.621294   0.2991
         0.642171   0.3458
         0.658984   0.3868
         0.671123   0.4183
         0.679268   0.4403
         0.684963   0.4562
         0.688893   0.4674
         0.692025   0.4765
         0.694641   0.4841
         0.696976   0.4910
         0.699099   0.4973
         0.700000   0.5000
         1.000000   0.9000             / TABLE NO. 01
--       --------   --------
         0.200000   0.0000
         0.238616   0.0002
         0.245309   0.0004
         0.261989   0.0010
         0.303091   0.0044
         0.368269   0.0191
         0.435026   0.0519
         0.486387   0.0940
         0.550683   0.1725
         0.575342   0.2115
         0.599076   0.2542
         0.621294   0.2991
         0.642171   0.3458
         0.658984   0.3868
         0.671123   0.4183
         0.679268   0.4403
         0.684963   0.4562
         0.688893   0.4674
         0.692025   0.4765
         0.694641   0.4841
         0.696976   0.4910
         0.699099   0.4973
         0.700000   0.5000
         1.000000   0.9000             / TABLE NO. 01
```

The example defines two WSF tables for use when only gas and water are present in the run.

8.3.369 ZMFVD – DEFINE COMPOSITIONAL COMPONENTS VERSUS DEPTH

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The ZMFVD keyword defines the compositional component mole fractions, for each component, as a function of depth, as such the keyword should have the same number of component columnar vectors as that declared via the COMPS keyword in the RUNSPEC section, and the NCOMPS keyword in the PROPS section. The keyword should only be used if the CO2STORE and GASWAT keywords in the RUNSPEC section have also be activated for the gas-water two component model.

Note

This is an OPM Flow keyword used with OPM Flow's CO2STORE and GASWAT keywords in the RUNSPEC section, and should not be confused with the more general version of the ZMFVD keyword used in the commercial compositional simulator.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	DEPTH	A columnar vector of real monotonically increasing down the column values that defines the depth values for the corresponding compositional component mole fractions. The default number of DEPTH values is 20, as defined by the NDRXVD parameter on the EQLDIMS keyword in the RUNSPEC section, and which may be used to reset the number of DEPTH values.			
		feet	m	cm	None
2	ZCOMP	A series of columnar vectors, with each columnar vector representing a compositional component mole fraction as a function of DEPTH. In addition, the sum of the compositional component mole fractions must sum to one for a given DEPTH value, otherwise an error will occur. Secondly, if the composition is independent of depth, then only one single row may be entered. Note that the number of columnar vectors, should be the same as that entered via the NCOMPS keyword in the PROPS section, and the COMPS keyword in the RUNSPEC section. Finally, only the default value of two components are currently supported by OPM Flow.			
		mole fraction	mole fraction	mole fraction	None

Notes:

1) The keyword is terminated by a "/".

Table 8.202: ZMFVD Keyword Description

Example

The following example defines how to confirm a two component formulation, together with defining the names of the composition components, as well as the compositional gradient, to be used with the CO2STORE and GASWAT options.

```
--
--       CONFIRM NUMBER OF COMPOSITIONAL COMPONENTS (OPM FLOW KEYWORD)
--
NCOMPS
        2                                                                    /
--
--       DEFINE COMPOSITIONAL COMPONENTS NAMES (OPM FLOW KEYWORD)
--
CNAMES
         'CO2'
         'H2O'                                                               /
--
--       COMPOSITIONAL COMPONENT MOLE FRACTIONS VS DEPTH (OPM FLOW KEYWORD)
--
--       DEPTH    CO2    H2O
ZMFVD
         2000     0.0    1.0
         2100     0.0    1.0
/
```

CHAPTER 9: REGIONS SECTION

9.1 INTRODUCTION

The REGIONS section defines how various properties in the PROPS and SOLUTION sections are allocated to individual cells within the model, as well as defining various fluid in-place reporting regions. This is accomplished by assigning an integer value to each cell that represents the data set of the property to be assigned to the grid block.

9.2 DATA REQUIREMENTS

OPM Flow, like most numerical modeling software, uses a default value of one for the various region arrays and thus if there is only one PVT data set for example, then there is no need to define the region array associated with allocating the PVT tables (via the PVTNUM keyword in the REGIONS section), as all cells will be allocated PVT table number one. However, if there are more than one PVT table entered in the PROPS section and PVTNUM is not defined in the model, then PVT tables greater than one will not be used and there will be no warning message indicting the fact.

Table 9.1 summarizes the various property region data and their associated keywords.

No.	REGIONS Keyword	Standard Property Regions	Property Section
1	EQLNUM	*EQLNUM – Define the Equilibration Region Numbers*. Equilibrium region allocation based on the EQUIL keyword in the SOLUTION section.	SOLUTION
2	FIPNUM	*FIPNUM – Define the Fluid In-Place Region Numbers*. Fluid In-Place reporting via the FIPNUM array that divides the model into different fluid in-place reporting regions.	REGION
3	PVTNUM	*PVTNUM – Define the PVT Regions*. PVT table allocation of the DENSITY, PVDG, PVDO, PVTG, PVTO, PVCO, PVTW and ROCK tables	PROPS
4	SATNUM	*SATNUM – Define the Saturation Table Region Numbers*. Saturation (relative permeability) table allocation of the SGFN, SWFN, SOF2, SOF3, SGOF, and SWOF tables. This array is for the standard drainage, as oppose to the imbibition curves, used in all models. There are separate region arrays for alternative relative permeability curves.	PROPS
	Function Specific Regions		
5	ENDNUM	*ENDNUM – Define the End-Point Scaling Depth Region Numbers*. ENPTVD and ENKRVD versus depth table allocation for when ENDSCALE option has been activated in the RUNSPEC section.	PROPS
6	FIP	*FIP – Define the Fluid In-Place Names and Region and Numbers*. Fluid In-Place reporting via the FIP array that divides the model into different fluid in-place reporting regions.	REGION
7	FIPOWG	*FIPOWG – Activate Oil, Gas, and Water FIP Zone Reporting*. Fluid In-Place reporting for the total oil, gas and water in the model. There is no region property array for this keyword, as the simulator calculates the total phase volumes based on the contacts entered via the EQUIL keyword in the SOLUTION section.	REGION
8	HBNUM	*HBNUM – Define Herschel-Bulkley Region Numbers*. Herschel-Bulkley rheological property table region numbers.	PROPS

No.	REGIONS Keyword	Standard Property Regions	Property Section
9	HWSNUM	*HWSNUM – Define the Saturation Table Region Numbers (High Salinity and Water Wet)*. High salinity water wet saturation table allocation using the high salinity water wet saturation SWFN and SOFN tables. This is an alias for the SURFWNUM property region which is also not supported by OPM Flow.	PROPS
10	IBMNUM	*IMBNUM – Define the Imbibition Saturation Table Region Numbers*. Imbibition saturation table allocation of the SWFN, SOF2, SOF3 or SWOF imbibition tables.	PROPS
11	IMBNUMMF	*IMBNUMMF – Define the Imbibition Saturation Table Region Numbers (Matrix-Fracture)*. Imbibition saturation tables region numbers for flow between the matrix and fracture blocks, for when the Hysteresis option has been activated in Dual Porosity or Dual Permeability runs.	PROPS
12	KRNUM	*KRNUM – Define the Directional Saturation Table Region Numbers*. Direction dependent saturation tables region numbers for when Directional Dependent Saturation Function option has been activated by the DIRECT parameter on the SATOPTS	PROPS
13	KRNUMMF	*KRNUMMF – Define the Saturation Table Region Numbers (Matrix-Fracture)*. Drainage saturation tables region numbers for flow between the matrix and fracture blocks in a Dual Porosity or Dual Permeability model.	PROPS
14	LSLTWNUM	*LSLTWNUM – Define the Low Salt Water Wet Saturation Table Region Numbers*. Low salt water wet saturation tables region numbers for when the Low Salinity option for the Brine model and the Surfactant Wettability option have been activated for the model.	PROPS
15	LSNUM	*LSNUM – Define the Low Salt Oil Wet Saturation Table Region Numbers*. Low salt oil wet saturation tables region numbers for the Low Salinity Brine model.	PROPS
16	LWSLTNUM	*LWSLTNUM – Define the Low Salt Oil Wet Saturation Table Region Numbers*. Low salt oil wet saturation table region numbers for the Low Salinity Brine model.	PROPS
17	LWSNUM	*LWSNUM – Define the Low Salt Water Wet Saturation Table Region Numbers*. Low salt water wet saturation table region numbers for when the Low Salinity option for the Brine model and the Surfactant Wettability option have been activated for the model.	PROPS
18	MISNUM	*MISNUM – Define the Miscibility Region Numbers*. Miscible regions based on the TLMIXPAR records when the MISCIBLE or SOLVENT keywords have been activated in the RUNSPEC section.	PROPS
19	OPERNUM	*OPERNUM – Define Regions for Mathematical Operations on Arrays*. OPERATER keyword region numbers that can also be used with the EQUALREG, ADDREG, COPYREG, MULTIREG, MULTREGP and MULTREGT keywords, as well as the OPERATER keyword in calculating various grid properties in the GRID and REGION section.	GRID EDIT PROPS REGION SOLUTION

No.	REGIONS Keyword	Standard Property Regions	Property Section
20	PENUM	*PENUM – Define the Petro-Elastic Region Numbers*. Petro-elastic region numbers for petro-elastic coefficients, bulk modulus functions and shear modulus functions.	PROPS
21	PLMIXNUM	*PLMIXNUM – Define the Polymer Region Numbers*. Polymer region numbers for the mixing tables, and the maximum polymer and salt concentrations for the Polymer option.	PROPS
22	RESIDNUM	*RESIDNUM – Define Vertical Equilibrium Residual Flow Region Numbers* . Vertical Equilibrium residual flow calculation saturation tables or when region numbers for the Vertical Equilibrium option.	PROPS
23	ROCKNUM	*ROCKNUM – Define Rock Compaction Table Region Numbers*. Rock compaction table allocation for when the ROCKCOMP keyword as been activated in the RUNSPEC section, that allocates the ROCKTAB series of tables to a cell.	PROPS
24	SURFNUM	*SURFNUM – Define the Surfactant Miscible Saturation Table Region Numbers*. Surfactant saturation (relative permeability) tables allocation allocating the SWFN, SOF2, SOF3 or SWOF as miscible tables.	PROPS
25	SURFWNUM	*SURFWNUM – Define the Saturation Table Region Numbers (High Salinity and Water Wet)*. High salinity water wet saturation table allocation using the high salinity water wet saturation SWFN and SOFN tables.	PROPS
26	TNUM	*TNUM – Define Passive Tracer Concentration Regions*. Partitioned Tracer option region numbers for this option.	PROPS
27	TRKPF	*TRKPF – Define Partitioned Tracer Regions*. Defines the regions associated with partition tracers and the partitioning tables allocated to grid blocks for when the Partitioned Tracer option has been enabled by the PARTTRAC keyword in the RUNSPEC section.	PROPS
28	WH2NUM	*WH2NUM – Define WAG Hysteresis Saturation Table Region Numbers (Two Phase)*. Two phase Water-Alternating-Gas ("WAG") hysteresis saturation table region numbers for when the hysteresis option has been activated by the WAGHYSTR variable on the SATOPTS keyword in the RUNSPEC.	PROPS
29	WH3NUM	*WH3NUM – Define WAG Hysteresis Saturation Table Region Numbers (Three Phase)*. Three phase WAG hysteresis saturation table region numbers for when the hysteresis option has been activated by the WAGHYSTR variable on the SATOPTS keyword in the RUNSPEC.	PROPS

Notes:

1) Note that not all keywords and features listed above are implemented in OPM Flow. Cells not colored in the No. column indicate the keyword is supported or partially supported by OPM Flow, cells colored gray indicate that the keyword is not applicable and will be ignored by the simulator, and finally, cells colored in orange indicate keywords that are not supported by OPM Flow and will be ignored by the simulator.

2) Note that is common to set the FIPNUM array to be equal to the EQLNUM to have fluid in-place reporting for each equilibrium region, this can be done by using the COPY keyword to copy the EQLNUM array to the FIPNUM array.

Table 9.1: REGION Section Allocation Array Summary

Note

Earth modeling software may export the region arrays to be loaded into the simulator with values starting from zero instead of one. This may result in an error message if these cells are active in the model, if the cells are inactive then the zero values will be ignored by the simulator.

The allocation is based on a complete property data set, that is all the property data associated with a given data set is allocated to the cell. For example, if the fluid properties for the model are the same, (for example, PVTO and PVDG keyword data) but the rock compressibility is varying with depth resulting in, say three different ROCK keyword records, then there has to be three complete data sets in order to allocate the three ROCK records. This would mean that the PVTO and PVDG keywords, in this instance, would have to be repeated three times to match the three ROCK keyword records.

Example SATNUM and EQUIL arrays from the Volve[283] field are displayed in Figure 9.1 and Figure 9.2, respectively.

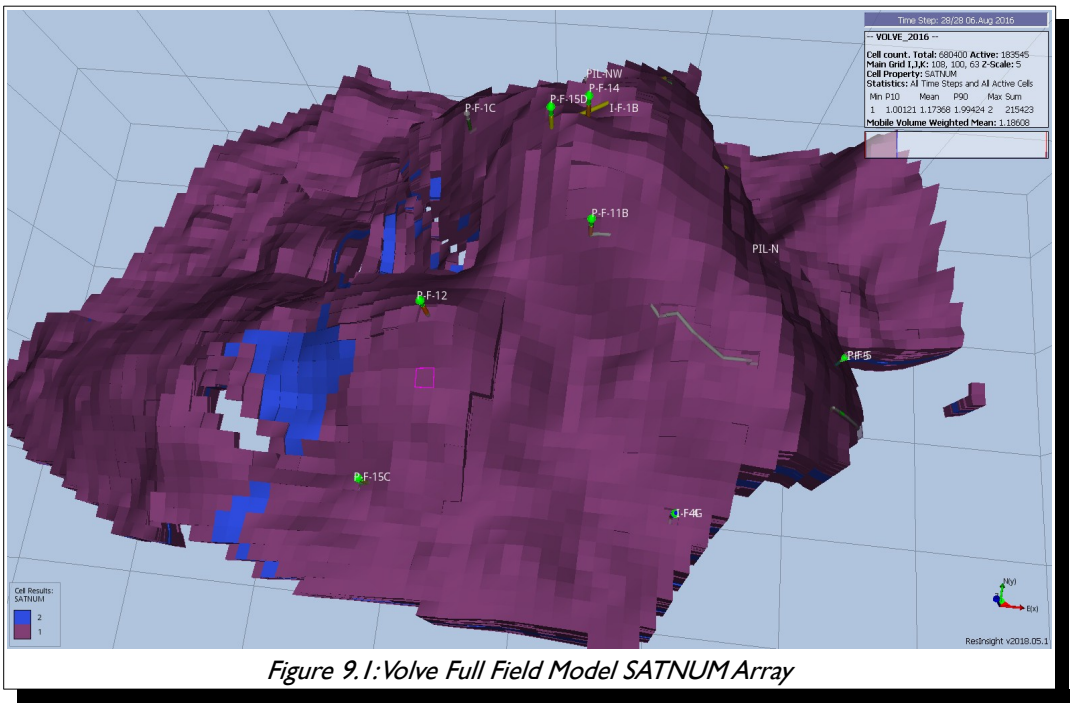


Figure 9.1: Volve Full Field Model SATNUM Array

[283] *The Volve Data was approved for data sharing in 2018 by the initiative of the last Operating company, Equinor and approved by the license partners ExxonMobil E&P Norway AS and Bayerngas Norge AS in the end of 2017.*

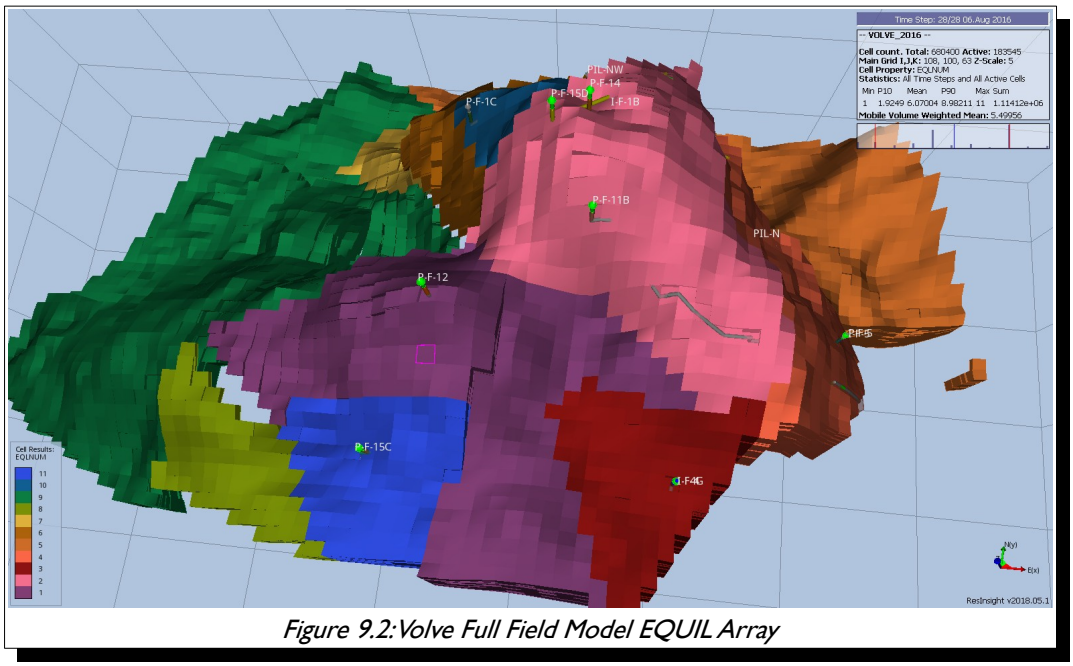


Figure 9.2:Volve Full Field Model EQUIL Array

9.3 KEYWORD DEFINITIONS

9.3.1 ADD – ADD A CONSTANT TO A SPECIFIED ARRAY

The ADD keyword adds a constant to a specified array or part of an array. The constant can be real or integer depending on the array type; however, the arrays that can be operated on is dependent on which section the ADD keyword is being applied.

See *ADD – Add a Constant to a Specified Array* in the GRID section for a full description.

9.3.2 ADDREG – ADD A CONSTANT TO AN ARRAY BASED ON A REGION NUMBER

The ADDREG keyword adds a constant to a specified array or part of an array based on cells with a specific region number. The region number array can be FLUXNUM, MULTNUM or OPERNUM and these arrays must be defined and be available before the ADDREG keyword is read by the simulator. The constant can be real or integer depending on the property array type; however, the property arrays that can be operated on is dependent on which section the ADDREG keyword is being used.

See *ADDREG – Add a Constant to an Array based on a Region Number* in the GRID section for a full description.

9.3.3 BOUNDARY – DEFINE A BOUNDARY BOX FOR PRINTING

The BOUNDARY keyword defines a rectangular grid for printing various arrays to the output print file (*.PRT); thus, avoiding printing all the elements in the selected array.

See *BOUNDARY – Define a Boundary Box for Printing* in the GRID section for a full description.

9.3.4 BOX - DEFINE A RANGE OF GRID BLOCKS TO ENTER PROPERTY DATA

BOX defines a range of grid blocks for which subsequent data is assigned for all the cells in the defined BOX. Note that the BOX grid is reset by the keyword ENDBOX by resetting the current defined BOX to be the whole grid. The keyword can be used for any array and for all grid types.

See *BOX - Define a Range of Grid Blocks to Enter Property Data* in the GRID section for a full description.

9.3.5 COLUMNS – DEFINE INPUT FILE COLUMN MARGINS

The COLUMNS keyword defines the input file column margins; characters outside the margins are ignored by the input parser.

See *COLUMNS – Define Input File Column Margins* in the GLOBAL section for a full description.

9.3.6 COPY – COPY ARRAY DATA TO ANOTHER ARRAY

The COPY keyword copies an array (or part of an array) to another array or part of an array. The arrays can be real or integer depending on the array type; however, the arrays that can be operated on is dependent on which section the COPY keyword is being used.

See *COPY – Copy Array Data to Another Array* in the GRID section for a full description.

9.3.7 COPYBOX – COPY ARRAY DATA DEFINED BY A BOX

The COPYBOX keyword copies an array (or part of an array) to part of the same array. The array can be real or integer depending on the array type; however, the arrays that can be operated on is dependent on which section the COPYBOX keyword is being used.

See *COPYBOX – Copy Array Data Defined by a Box* in the GRID section for a full description.

9.3.8 COPYREG – COPY AN ARRAY TO ANOTHER ARRAY BASED ON A REGION NUMBER

The COPYREG keyword copies a specified array or part of an array based on cells with a specific region number to another array. The region number array can be FLUXNUM, MULTNUM or OPERNUM and these arrays must be defined and be available before the COPYREG keyword is read by the simulator. The property arrays can be real or integer depending on the property array type; however, the property arrays that can be operated on is dependent on which section the COPYREG keyword is being used.

See *COPYREG – Copy an Array to Another Array based on a Region Number* in the GRID section for a full description.

9.3.9 DEBUG – DEFINE THE DEBUG DATA TO BE PRINTED TO FILE

This keyword defines the debug data to be written to the debug file (*.DBG). This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

See *DEBUG – Define the Debug Data to be Printed to File* in the GLOBAL section for a full description.

9.3.10 ECHO – ACTIVATE ECHOING OF USER INPUT FILES TO THE PRINT FILE

Turns on echoing of all the input files to the print file; note that this keyword is activated by default and can subsequently be switched off by the NOECHO activation keyword.

See *ECHO – Activate Echoing of User Input Files to the Print File* in the GLOBAL section for a full description.

9.3.11 END – DEFINE THE END OF THE INPUT FILE

This keyword marks the end of the input file and can occur in any section. Any keywords and data after the END keyword are ignored.

See *END – Define the End of the Input File* in the GLOBAL section for a full description.

9.3.12 ENDBOX – DEFINE THE END OF THE BOX DEFINED GRID

This keyword marks the end of a previously defined BOX sub-grid as defined by a previously entered BOX keyword. The keyword resets the input grid to be the full grid as defined by the NX, NY, and NZ variables on the DIMENS keyword in the RUNSPEC section.

See *ENDBOX – Define the End of the BOX Defined Grid* in the GRID section for a full description.

9.3.13 ENDFIN – END THE DEFINITION OF A LOCAL GRID REFINEMENT

ENDFIN defines the end of a Cartesian or radial local grid refinement ("LGR") definition and a LGR property definition data set.

See *ENDFIN – End the Definition of a Local Grid Refinement* in the GRID section for a full description.

9.3.14 ENDINC – DEFINE THE END OF AN INCLUDE FILE

This keyword marks the end of an include file specified on the INCLUDE keyword. When the ENDINC keyword is encountered in the INCLUDE file, input data is read from the next keyword in the current file. Any keywords and data after the ENDINC keyword in the INCLUDE file are ignored.

See *ENDINC – Define the End of an Include File* in the GLOBAL section for a full description.

9.3.15 ENDNUM – DEFINE THE END-POINT SCALING DEPTH REGION NUMBERS

Description

The ENDNUM keyword defines the end-point scaling depth table region numbers for each grid block. The end-point scaling depth tables for various regions are defined by the ENPVTD[284] and the ENKRVD[285] keywords in the PROPS section. In the RUNSPEC section the NTENDP variable on the ENDSCALE keyword defines the maximum number of depth tables.

No.	Name	Description	Default
1	ENDNUM	ENDNUM defines an array of positive integers assigning a grid cell to a particular end-point scaling depth table region. The maximum number of ENDNUM regions is set by the NTENDP variable on the ENDSCALE keyword in the RUNSPEC section.	1

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) If a cell is not assigned a ENDNUM region number then the default value of one will be used.
3) The keyword is terminated by a "/".

Table 9.2: ENDNUM Keyword Description

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

Examples

The example below sets three ENDNUM regions for a 4 x 5 x 2 model.

```
--
--        DEFINE ENDNUM REGIONS FOR ALL CELLS
--
ENDNUM
          2  2  1  1  2  2  1  1  1  1  1  1  1  1  1  1  1  1  1  1
          3  3  1  1  3  3  1  1  1  1  1  1  1  1  1  1  1  1  1  1
/
```

Alternatively the EQUALS keyword could be employed to accomplish the same task, that is:

```
--
--        ARRAY       CONSTANT      ---------- BOX ---------
--                                  I1  I2   J1  J2   K1  K2
EQUALS
          ENDNUM      1             1*  1*   1*  1*   1*  1* / SET REGION 1
          ENDNUM      2             1   2    1   2    1   1  / SET REGION 2
          ENDNUM      3             1   2    1   2    2   2  / SET REGION 3
/
```

[284] *This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.*

[285] *This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.*

9.3.16 ENDSKIP – DEACTIVATE SKIPPING OF KEYWORDS AND INPUT DATA

Turns off skipping of keywords that was activated by the SKIP, SKIP100, or SKIP300 keywords. Note that for each SKIP keyword activated there must be a corresponding ENDSKIP keyword.

See *ENDSKIP – DeActivate Skipping of Keywords and Input Data* in the GLOBAL section for a full description.

9.3.17 EQLNUM – DEFINE THE EQUILIBRATION REGION NUMBERS

Description

The EQLNUM keyword defines the equilibration region numbers for each grid block. The equilibration data for various regions are defined in the SOLUTION section. For example, the EQUIL keyword in the SOLUTION section defines the initial pressures and fluid contacts for each equilibration region identified by the EQLNUM region array.

No.	Name	Description	Default
1	EQLNUM	EQLNUM defines an array of positive integers assigning a grid cell to a particular equilibration region. The maximum number of EQLNUM regions is set by the NTEQUL variable on the EQLDIMS keyword in the RUNSPEC section.	1

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) The EQLNUM and PVTNUM arrays need to be consistent, that is the all cells with the same EQLNUM can only belong to one PVTNUM region.
3) If a cell is not assigned a EQLNUM region number then the default value will be used.
4) The keyword is terminated by a "/".

Table 9.3: EQLNUM Keyword Description

Examples

The example below sets three EQLNUM regions for a 4 x 5 x 2 model.

```
--
--        DEFINE EQLNUM REGIONS FOR ALL CELLS
--
EQLNUM
          2  2  1  1  2  2  1  1  1  1  1  1  1  1  1  1  1  1  1  1
          3  3  1  1  3  3  1  1  1  1  1  1  1  1  1  1  1  1  1  1
/
```

Alternatively the EQUALS keyword could be employed to accomplish the same task, that is:

```
--
--        ARRAY       CONSTANT      ---------- BOX ---------
--                                  I1  I2   J1  J2   K1  K2
EQUALS
          EQLNUM'     1             1*  1*   1*  1*   1*  1* / SET REGION 1
          EQLNUM'     2             1   2    1   2    1   1  / SET REGION 2
          EQLNUM'     3             1   2    1   2    2   2  / SET REGION 3
/
```

9.3.18 EQUALREG – SETS AN ARRAY TO A CONSTANT BY REGION NUMBER

The EQUALREG keyword sets a specified array to a constant for cells with a specific region number. The region number array can be FLUXNUM, MULTNUM or OPERNUM and these arrays must be defined and be available before the EQUALREG keyword is read by the simulator. The constant can be real or integer depending on the property array type; however, the property arrays that can be operated on is dependent on which section the EQUALREG keyword is being used.

See *EQUALREG – Sets an Array to a Constant by Region Number* in the GRID section for a full description.

9.3.19 EQUALS – SETS A SPECIFIED ARRAY TO A CONSTANT

The EQUALS keyword sets a specified array or part of an array to a constant. The constant can be real or integer depending on the array type; however, the arrays that can be operated on is dependent on which section the EQUALS keyword is being used.

See *EQUALS – Sets a Specified Array to a Constant* in the GRID section for a full description.

9.3.20 EXTRAPMS – ACTIVATE EXTRAPOLATION WARNING MESSAGES

The EXTRAPMS keyword activates extrapolation warning messages for when OPM Flow extrapolates the PVT or VFP tables. Frequent extrapolation warning messages should be investigated and resolved as this would indicate possible incorrect data and may result in the simulator extrapolating to unrealistic values.

See *EXTRAPMS – Activate Extrapolation Warning Messages* in the GLOBAL section for a full description.

9.3.21 FILEUNIT – ACTIVATE UNIT CONSISTENCY VERIFICATION

The FILEUNIT keyword defines the units of the of the data set, and is used to verify that the units in the input deck and any associated include files are consistent grid data. The keyword does not provide for the conversion between different sets of units.

See *FILEUNIT – Activate Unit Consistency Checking* in the GRID section for a full description.

9.3.22 FIP – DEFINE THE FLUID IN-PLACE NAMES AND REGION AND NUMBERS

Description

The FIP keyword defines the fluid in-place name and the associated region numbers for each grid block. The simulator can print out summaries of the fluid in-place in each region, the current flow rates between regions, and the cumulative flows between regions. This keyword is not in the standard keyword format due to the fluid in-place name being concatenated to the keyword FIP to fully define the keyword.

Note that the total number of FIP and FIPNUM regions must be defined by the NMFIPR variable on the REGDIMS keyword, or the NTFIP variable on the TABDIMS keyword. Both keywords are in the RUNSPEC section.

No.	Name	Description	Default
1	FIPNAME	A character string of up to eight characters, consisting of FIP as the first three characters followed by up to a five letter character string defining the fluid in-place's name.	None
2	FIPNUM	FIPNUM defines an array of positive integers greater than or equal to one, that assigns a grid cell to a particular fluid in-place region named by FIPNAME. The maximum number of FIP and FIPNUM regions is set by the REGDIMS(NMFIPR) or the TABDIMS(NTFIP) keywords(variables) in the RUNSPEC section. If both REGDIMS(NMFIPR) and TABDIMS(NTFIP) have been defined then the maximum of the two is used.	1

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) If a cell is not assigned a FIP region number by the end of the REGION section then the fault value of one will be used.
3) The keyword is terminated by a "/".

Table 9.4: FIP Keyword Description

The keyword behaves the same as the FIPNUM keyword except the full name of the keyword, including the concatenated characters, are used as the property region name. For example, if we wish define a fluid in-place region name called UNIT, then the keyword would be FIPUNIT.

Note

The commercial simulator prints out a fluid in-place report if the FIP option on the RPTSCHED keyword is set equal to three, that is: FIP=3. This option is currently not available in OPM Flow.

The region property data for FIP arrays can be written to the SUMMARY file, and the RSM file if requested, similar to the FIPNUM regions, with some caveats:

1) Only SUMMARY keywords for regions may be used, that is the SUMMARY variable name must begin with the letter R.
2) The SUMMARY variable name must consist of a character string length of exactly five characters, if less than five characters, then the "_" character should be used to fill out the SUMMARY variable name. For example, instead of RPRUNIT, use RPR__UNI, or instead of ROIPUNIT, use ROIP_UNI.

3) Only the first three characters of the FIP region name should be concatenated with the SUMMARY variable name. This means if the FIP region name is UNIT, the FIP keyword would be FIPUNIT; however, to access the regional pressures for FIPUNIT, one should use RPR__UNI, or to access the regional oil in-place one would use ROIP_UNI.

See also the FIPOWG keyword in the REGIONS section that automatically defines the fluid-in-place regions at the start of the run based the gas, oil and water zones at the time the model was initialized.

Example

The example below defines a region name of UNIT and sets three FIPUNIT regions for a 4 x 5 x 2 model.

```
--
--       DEFINE FIPUNIT FIP REGIONS FOR ALL CELLS
--
FIPUNIT
         2  2  1  1  2  2  1  1  1  1  1  1  1  1  1  1  1  1  1  1
         3  3  1  1  3  3  1  1  1  1  1  1  1  1  1  1  1  1  1  1
/
```

The second example is based on the Norne model that defines two FIP regions based on geological layers and numerical layers using the EQUALS keyword.

```
--       FIPGL BASED ON GEOLOGICAL LAYERS
--       ARRAY       CONSTANT     ---------- BOX ---------
--                                I1  I2   J1  J2   K1  K2
EQUALS
         FIPGL       1             1  46    1 112    1   3   /  Garn
         FIPGL       2             1  46    1 112    4   4   /  Not
         FIPGL       3             1  46    1 112    5   5   /  Ile 2.2
         FIPGL       4             1  46    1 112    6   8   /  Ile 2.1
         FIPGL       5             1  46    1 112    9  11   /  Ile 1
         FIPGL       6             1  46    1 112   12  12   /  Tofte 2.2
         FIPGL       7             1  46    1 112   13  15   /  Tofte 2.1
         FIPGL       8             1  46    1 112   16  18   /  Tofte 1
         FIPGL       9             1  46    1 112   19  22   /  Tilje
--
--       FIPNL BASED ON NUMERICAL LAYERS
--
         FIPNL       1             1  46    1 112    1   1   /  Garn 3
         FIPNL       2             1  46    1 112    2   2   /  Garn 2
         FIPNL       3             1  46    1 112    3   3   /  Garn 1w
         FIPNL       4             1  46    1 112    4   4   /  Not
         FIPNL       5             1  46    1 112    5   5   /  Ile 2.2
         FIPNL       6             1  46    1 112    6   6   /  Ile 2.1.3
         FIPNL       7             1  46    1 112    7   7   /  Ile 2.1.2
         FIPNL       8             1  46    1 112    8   8   /  Ile 2.1.1
         FIPNL       9             1  46    1 112    9   9   /  Ile 1.3
         FIPNL      10             1  46    1 112   10  10   /  Ile 1.2
         FIPNL      11             1  46    1 112   11  11   /  Ile 1.1
         FIPNL      12             1  46    1 112   12  12   /  Tofte 2.2
         FIPNL      13             1  46    1 112   13  13   /  Tofte 2.1.3
         FIPNL      14             1  46    1 112   14  14   /  Tofte 2.1.2
         FIPNL      15             1  46    1 112   15  15   /  Tofte 2.1.1
         FIPNL      16             1  46    1 112   16  16   /  Tofte 1.2.2
         FIPNL      17             1  46    1 112   17  17   /  Tofte 1.2.1
         FIPNL      18             1  46    1 112   18  18   /  Tofte 1.1
         FIPNL      19             1  46    1 112   19  19   /  Tilje 4
         FIPNL      20             1  46    1 112   20  20   /  Tilje 3
         FIPNL      21             1  46    1 112   21  21   /  Tilje 2
         FIPNL      22             1  46    1 112   22  22   /  Tilje 1
/
```

Then in order to get the reservoir pressure for the regions and the in-place oil volumes SUMMARY variables written to the SUMMARY file, one would use the following SUMMARY variable names in the SUMMARY section

```
-- ==========================================================================
--
-- SUMMARY SECTION
--
-- ==========================================================================
SUMMARY
--       FIP REGION REPORTING
--
ROIP_GL
/
ROIP_NL
/
RPR__GL
/
RPR__NL
/
```

Notice how the "_" character has been used to extend the SUMMARY variable name to five characters.

9.3.23 FIPNUM – DEFINE THE FLUID IN-PLACE REGION NUMBERS

Description

The FIPNUM keyword defines the fluid in-place region numbers for each grid block. The simulator can print out summaries of the fluid in-place in each region, the current flow rates between regions, and the cumulative flows between regions.

Note that the total number of FIPNUM and FIP regions must be defined by the NMFIPR variable on the REGDIMS keyword, or the NTFIP variable on the TABDIMS keyword. Both keywords are in the RUNSPEC section.

No.	Name	Description	Default
1	FIPNUM	FIPNUM defines an array of positive integers greater than or equal to one, that assigns a grid cell to a particular fluid in-place region. The maximum number of FIPNUM and FIP regions is set by the REGDIMS(NMFIPR) or the TABDIMS(NTFIP) keywords(variables) in the RUNSPEC section. If both REGDIMS(NMFIPR) and TABDIMS(NTFIP) have been defined then the maximum of the two is used.	1

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) If acell is not assigned a FIPNUM region then the default value will be used.
3) The keyword is terminated by a "/".

Table 9.5: FIPNUM Keyword Description

Note

In most simulation models the FIPNUM array is used to define various regions in the model for fluid in-place reporting and to identify (or report) the flow between the different regions. When calibrating a model's in-place volumes it would be useful to use the FIPNUM array combined with the MULTREGP keyword to accomplish this. However, the FIPNUM array cannot be used in the GRID section.

A work around is to:

1) Use the FIPNUM array but change the keyword to MULTNUM and incorporate this keyword or INCLUDE file in the GRID section.
2) Use the MULTREGP to calibrate the fluid in-place volumes for the various regions.
3) In the REGIONS section, use the COPY keyword to copy the MULTNUM array to the FIPNUM array.

The above work flow will ensure that both arrays and the reporting of fluid in-place regions are consistent.

Examples

The example below sets three FIPNUM regions for a 4 x 5 x 2 model.

```
--
--       DEFINE FIPNUM REGIONS FOR ALL CELLS
--
FIPNUM
         2  2  1  1  2  2  1  1  1  1  1  1  1  1  1  1  1  1  1  1
         3  3  1  1  3  3  1  1  1  1  1  1  1  1  1  1  1  1  1  1
/
```

Alternatively the EQUALS keyword could be employed to accomplish the same task, that is:

```
--
--       ARRAY       CONSTANT     ---------- BOX ---------
--                                I1  I2   J1  J2   K1  K2
EQUALS
         FIPNUM     1            1*  1*   1*  1*    1*  1* / SET REGION 1
         FIPNUM     2            1   2    1   2     1   1  / SET REGION 2
         FIPNUM     3            1   2    1   2     2   2  / SET REGION 3
/
```

9.3.24 FIPOWG – ACTIVATE OIL, GAS, AND WATER FIP ZONE REPORTING

Description

The FIPOWG keyword activates automatic fluid in-place reporting based on the initial oil, gas and water zones defined by the initial equilibration. The fluid contacts on the EQUIL keyword in the SOLUTION section determine the reporting fluid category a grid cell belongs to. For example all grid cells with depths above the gas-oil contact on the EQUIL keyword will be assigned to the gas zone and reported accordingly. Similarly, grid cells with depths between the gas-oil contact and the water-oil contact will be assigned to the oil zone. And finally, grid cells with depths below the oil-water contact will be assigned to the water zone. The simulator can print out summaries of the fluid in-place in each region, the current flow rates between regions, and the cumulative flows between regions.

Note that the total number of FIP and FIPNUM regions must be defined by the NMFIPR variable on the REGDIMS keyword in the RUNSPEC section.

There is no data required for this keyword.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

Example

```
--
--       ACTIVATE OIL, GAS, AND WATER FIP ZONE REPORTING
--
FIPOWG
```

The above example switches on automatic fluid in-place reporting based on the initial oil, gas and water zones defined by the initial equilibration.

9.3.25 FORMFEED – DEFINED THE PRINT FILE FORM-FEED CHARACTER

The FORMFEED keyword defines the form-feed character, or carriage control character, for the output print (*.PRT) run summary (*.RSM) files. The keyword should be place at the very top of the input file.

See *FORMFEED – Defined the Print File Form-Feed Character* in the GLOBAL section for a dull description.

9.3.26 GETDATA – LOAD AND ASSIGN DATA ARRAY FROM INIT OR RESTART FILE

The GETDATA keyword loads a data array from a previously generated INIT or RESTART file and assigns the loaded array to either same array in the run or another array name.

See *GETDATA – Load and Assign Data Array from INIT or RESTART Files* in the GRID section for a full description.

9.3.27 HA – HISTORY MATCH END-POINT GRADIENT ADDITIVE MODIFIER

The HA series of keywords defines the history match end-point gradient parameters used to set the additive cumulative end point data, for when the History Match Gradient option has been activated by the HMDIMS keyword in the RUNSPEC section.

See *HA – History Match End-Point Gradient Additive Modifier* in the PROPS section for a complete description.

9.3.28 HBNUM – DEFINE HERSCHEL-BULKLEY REGION NUMBERS

Description

The HBNUM keyword defines the Herschel-Bulkley rheological property table region numbers for each grid block, as such there must be one entry for each cell in the model. The region number specifies which table of Herschel-Bulkley rheological property data is assigned to a grid block, for when the Polymer option has been invoked via the POLYMER keyword in the RUNSPEC section and the Non-Newtonian Fluid phase has been declared active by the NNEWTF keyword, also in the RUNSPEC section. The FHERCHBL keyword in the PROPS section is used to specify the Herschel-Bulkley rheological property table data.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

9.3.29 HM – HISTORY MATCH REGION GRADIENT PARAMETERS

Description

The HM series of keywords in the REGION section defines the history match gradient regions and sub-regions, for when the History Match Gradient option has been activated by the HMDIMS keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

For grid properties, the region name (or region property array) is based on the property arrays defined in Table 9.6.

Property Array	Region Name	Grid Property Data Desription
PERMX	HMPERMX	Permeability multipliers in the x-direction.
PERMXY	HMPRMXY	Permeability multipliers in the x-direction and y-direction
PERMY	HMPERMY	Permeability multipliers in the y-direction.
PERMZ	HMPERMZ	Permeability multipliers in the z-direction.
PORV	HMPORV	Pore volume multiplier
SIGMA	HMSIGMA	Dual porosity and/or dual permeability SIGMA multiplier
TRANX	HMTRANX	Transmissibility multipliers in the x-direction.
TRANXY	HMTRNXY	Transmissibility multipliers in the x-direction and y-direction
TRANY	HMTRANY	Transmissibility multipliers in the y-direction.
TRANZ	HMTRANZ	Transmissibility multipliers in the z-direction.

Table 9.6: HM Region Grid Gradient Parameter Keyword List

In addition, if the End-Point Scaling option has been activated by the ENDSCALE keyword in the RUNSPEC section, then the history match gradient regions and sub-regions for the end-point data can be specified. In this the keyword consists of the first two characters of "HM" followed by the end-point keyword (Table 9.7), for example, HMSWL.

Type	End-Point Keyword	Oil-Water End-Point Definitions
Saturation	HMSWL	Connate water saturation, that is the smallest water saturation in a water saturation function table.
	HMSWCR	Critical water saturation, that is the largest water saturation for which the water relative permeability is zero.
	HMSOWCR	Critical oil-in-water saturation, that is the largest oil saturation for which the oil relative permeability is zero in an oil-water system.

Type	End-Point Keyword	Oil-Water End-Point Definitions
Relative Permeability	HMKRW	Relative permeability of water at the maximum water saturation (normally the maximum water saturation is one).
	HMKRO	Relative permeability of oil at the maximum oil saturation.
	HMKRWR	Relative permeability of water at the residual oil saturation or the residual gas saturation in a gas-water run.
	HMKRORW	Relative permeability of oil at the critical water saturation.
Capillary Pressure	HMSWLPC	Capillary pressure connate water saturation, that is the smallest water saturation in a water saturation function table.
Type	**End-Point Keyword**	**Gas-Oil End-Point Definitions**
Saturation	HMSGL	Connate gas saturation, that is the smallest gas saturation in a gas saturation function table.
	HMSGCR	Critical gas saturation, that is the largest gas saturation for which the gas relative permeability is zero.
	HMSOGCR	Critical oil-in-gas saturation, that is the largest oil saturation for which the oil relative permeability is zero in an oil-gas-connate water system.
Relative Permeability	HMKRG	Relative permeability of gas at the maximum gas saturation.
	HMKRGR	Relative permeability of gas at the residual oil saturation or the critical water saturation in a gas-water run.
	HMKRORG	Relative permeability of oil at the critical gas saturation.
Capillary Pressure	HMSGLPC	Capillary pressure connate gas saturation, that is the smallest gas saturation in a gas saturation function table.

Table 9.7: HM Region End-Point Gradient Parameter Keyword List

9.3.30 HMPROPS – HISTORY MATCH END-POINT SECTION START

HMPROPS defines the start of a history match end-points section, for when the History Match Gradient option has been activated by the HMDIMS keyword in the RUNSPEC section. In addition, the End-Point Scaling option must also be activated by the ENDSCALE keyword which is also in the RUNSPEC section. The keyword allows for the BOX, EQUALS, COPY, MINVALUE, MAXVALUE and ADD keywords to be used with the HM series of keywords that reference the end-point scaling arrays, that is: HMKRG, HMKRGR, HMKRO, HMKRORG, HMKRORW, HMKRW, HMKRWR, HMPCW, HMPCG, HMSGCR, HMSOWCR, HMSOGCR, HMSWCR, and HMSWL.

See *HMPROPS – History Match End-Point Section Start* in the PROPS section for a full description.

9.3.31 HWSNUM – DEFINE THE SATURATION TABLE REGION NUMBERS (HIGH SALINITY AND WATER WET)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The HWSNUM keyword defines the saturation tables (relative permeability and capillary pressure tables) region numbers for each grid block, as such there must be one entry for each cell in the mode, for when the Surfactant Wettability option has been selected. The keyword may also be used with the Low Salt Brine option, in this case the water wet curves are calculated as a function of the low and high water wet salinity curves. The region number specifies which set of relative permeability tables are used to calculate the relative permeability and capillary pressure in a grid block. Note that the keyword is obligatory if the SURFACTW keyword in the RUNSPEC section has been used to invoke the Surfactant Wettability option.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description	Default
1	HWSNUM	HWSNUM defines an array of positive integers assigning a grid cell to a particular saturation table region. The maximum number of HWSNUM regions is set by the NTSFUN variable on the TABDIMS keyword in the RUNSPEC section.	1

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) If a cell is not assigned a SATNUM region then the default value will be used.
3) The keyword is terminated by a "/".

Table 9.8: HWSNUM Keyword Description

Examples

The example below sets three HWSNUM regions for a 4 x 5 x 2 model.

```
--
--        DEFINE HWSNUM REGIONS FOR ALL CELLS
--
HWSNUM
         2  2  1  1  2  2  1  1  1  1  1  1  1  1  1  1  1  1  1  1
         3  3  1  1  3  3  1  1  1  1  1  1  1  1  1  1  1  1  1  1
/
```

Alternatively the EQUALS keyword could be employed to accomplish the same task, that is:

```
--
--        ARRAY       CONSTANT     ---------- BOX ---------
--                                 I1  I2   J1  J2   K1  K2
EQUALS
         HWSNUM      1            1*  1*   1*  1*   1*  1* / SET REGION 1
         HWSNUM      2            1   2    1   2    1   1  / SET REGION 2
         HWSNUM      3            1   2    1   2    2   2  / SET REGION 3
/
```

9.3.32 IMBNUM – DEFINE THE IMBIBITION SATURATION TABLE REGION NUMBERS

Description

The IMBNUM keyword defines the imbibition saturation tables (relative permeability and capillary pressure tables) region numbers for each grid block, as such there must be one entry for each cell in the model. The region number specifies which set of relative permeability tables (SGFN, SWFN, SOF2, SOF3, SOF32D, SGOF, SLGOF and SWOF) are used to calculate the relative permeability and capillary pressure in a grid block.

No.	Name	Description	Default
1	IMBNUM	IMBNUM defines an array of positive integers assigning a grid cell to a particular saturation table region. The maximum number of IMBNUM regions is set by the NTSFUN variable on the TABDIMS keyword in the RUNSPEC section.	1

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) The keyword is terminated by a "/".
3) If a cell is not assigned an IMBNUM region then the default value of one will be used.

Table 9.9: IMBNUM Keyword Description

In addition, saturation table assignment may be directional dependent in which case the directional dependent versions of the aforementioned array should be used, that is IMBNUMX, IMBNUMY and IMBNUMZ instead of IMBNUM. There is also the facility to make the directional end-point scaling reversible or non-reversible and if the non-reversible option is selected, the non-reversible versions of the aforementioned arrays should be used, that is IMBNUMX, IMBNUMX-, IMBNUMY, IMBNUMY-, IMBNUMZ and IMBNUMZ-, instead of the IMBNUM keyword. For reference, see *Error: Reference source not found*, that lists the various keywords that may be used with directional dependent relative permeability tables.

Note, currently IMBNUMX-, IMBNUMY-, and IMBNUMZ- are not supported.

Example

The example below sets three IMBNUM regions for a 4 x 5 x 2 model using the EQUALS keyword.

```
--
--        ARRAY        CONSTANT      ---------- BOX ---------
--                                    I1   I2    J1   J2    K1   K2
EQUALS
          IMBNUM       1             1*   1*    1*   1*    1*   1* / SET REGION 1
          IMBNUM       2             1    2     1    2     1    1  / SET REGION 2
          IMBNUM       3             1    2     1    2     2    2  / SET REGION 3
/
```

9.3.33 IMBNUMMF – DEFINE THE IMBIBITION SATURATION TABLE REGION NUMBERS (MATRIX-FRACTURE)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The IMBNUMMF keyword defines the imbibition saturation tables (relative permeability and capillary pressure tables) region numbers for flow between the matrix and fracture blocks, for when the HYSTER option on the SATOPTS keyword has been invoked to activate the Hysteresis option, and the Dual Porosity or Dual Permeability models have been activated via the DUALPORO or DUALPERM keywords. All keywords are in the RUNSPEC section.

The region number specifies which set of relative permeability tables (SGFN, SWFN, SOF2, SOF3, SOF32D, SGOF, SLGOF and SWOF) are used to calculate the relative permeability and capillary pressure between the matrix and fracture blocks. The keyword is optional and any cell not assigned a value will use the assignment from the IMBNUM array.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

9.3.34 IMPORT – IMPORT GRID FILE DATA AT THE CURRENT POSITION

The IMPORT keyword informs the simulator to continue reading input data from the specified IMPORT file. When the end of the IMPORT file is reached, input data is read from the next keyword in the current file. Normally IMPORT files are generated by grid pre-processing software and the keyword allows for both formatted and unformatted (binary) files to be loaded.

See *IMPORT – Import Grid File Data at the Current Position* in the GRID section for a full description.

9.3.35 INCLUDE – LOAD ANOTHER DATA FILE AT THE CURRENT POSITION

The INCLUDE keyword informs OPM Flow to continue reading input data from the specified INCLUDE file. When the end of the INCLUDE file is reached, or the ENDINC keyword is encountered in the included file, input data is read from the next keyword in the current file.

See *INCLUDE – Load Another Data File at the Current Position* in the GLOBAL section for a full description.

9.3.36 KRNUM – DEFINE THE DIRECTIONAL SATURATION TABLE REGION NUMBERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The KRNUM keyword defines the direction dependent saturation tables (relative permeability and capillary pressure tables) region numbers for each grid block face, as such there must be one entry for each cell in the model. The region number specifies which set of relative permeability tables (SGFN, SWFN, SOF2, SOF3, SOF32D, SGOF, SLGOF and SWOF) are used to calculate the relative permeability and capillary pressure in a grid block. The keyword should only be used if Directional Dependent Saturation Function option has been activated by the DIRECT parameter on the SATOPTS keyword in the RUNSPEC section. Otherwise the standard none directional relative permeability curves should be assigned by the SATNUM keyword in the REGIONS section.

This keyword is not in the standard keyword format due to the cell face (X, X+, Y, Y+, Z, and Z+ for Cartesian grids and R, R+, T, T+ for radial grids) being concatenated to the keyword KRNUM to fully define the keyword.

No.	Name	Description	Default
1	KRNUM	KRNUM defines an array of positive integers assigning a grid cell to a particular saturation table region. The maximum number of KRNUM regions is set by the NTSFUN variable on the TABDIMS keyword in the RUNSPEC section.	1

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) If a cell is not assigned a KRNUM region then the default value of one will be used.
3) The keyword is terminated by a "/".

Table 9.10: KRNUM Keyword Description

If the Directional Dependent Saturation Function option has been activated by the DIRECT parameter on the SATOPTS keyword in the RUNSPEC section, then the KRNUMX, KRNUMY and KRNUMZ form of the keyword should be used. Secondly, if the Non-Reversible End-Point Scaling option is selected by the IRREVERS parameter on the SATOPTS keyword in the RUNSPEC section, then the non-reversible versions of the KRNUM should be used, that is KRNUMX, KRNUMX-, KRNUMY, KRNUMY-, KRNUMZ and KRNUMZ-. For reference, see *Error: Reference source not found*, that lists the various keywords that may be used with directional dependent relative permeability tables.
Note, currently KRNUMX-, KRNUMY-, and KRNUMZ- are not supported.

Example

The example below sets the directional saturation tables in all three directions using the EQUALS keyword.

```
--
--        ARRAY       CONSTANT      ---------- BOX ---------
--                                  I1  I2   J1  J2   K1  K2
EQUALS
          KRNUMX      1             1*  1*   1*  1*   1*  1* / SET X-DIR TABLES
          KRNUMY      2             1*  1*   1*  1*   1*  1* / SET Y-DIR TABLES
          KRNUMZ      3             1*  1*   1*  1*   1*  1* / SET Z-DIR TABLES
/
```

9.3.37 KRNUMMF – DEFINE THE SATURATION TABLE REGION NUMBERS (MATRIX-FRACTURE)

Description

The KRNUMMF keyword defines the drainage saturation tables (relative permeability and capillary pressure tables) region numbers for flow between the matrix and fracture blocks, for when the Dual Porosity or Dual Permeability models have been activated via the DUALPORO or DUALPERM keywords in the RUNSPEC section.

The region number specifies which set of relative permeability tables (SGFN, SWFN, SOF2, SOF3, SOF32D, SGOF, SLGOF and SWOF) are used to calculate the relative permeability and capillary pressure between the matrix and fracture blocks. The keyword is optional and any cell not assigned a value will use the assignment from the SATNUM array.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

9.3.38 LSLTWNUM – DEFINE THE LOW SALT WATER WET SATURATION TABLE REGION NUMBERS

Description

The LSLTWNUM keyword defines the saturation tables (relative permeability and capillary pressure tables) region numbers for each grid block, as such there must be one entry for each cell in the model. The region number specifies which set of relative permeability tables (SWFN, SOF3 and related keywords) are used to calculate the relative permeability and capillary pressure in a grid block. The keyword should only be used if the Low Salinity option for the Brine model and the Surfactant Wettability option have been activated by the LOWSALT and SURFACTW keywords, respectively, in the RUNSPEC section.

The water wet curves are calculated as a weighted average of the low salinity saturation tables (allocated by this keyword) and the high salinity water wet saturation tables (allocated by the SURFWNUM keyword in the REGIONS section), using the weights provided by the LSALTFNC keyword in the PROPS section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description	Default
1	LSLTWNUM	LSLTWNUM defines an array of positive integers assigning a grid cell to a particular saturation table region. The maximum number of LSLTWNUM regions is set by the NTSFUN variable on the TABDIMS keyword in the RUNSPEC section.	1

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) If a cell is not assigned a LSLTWNUM region then the default value will be used.
3) The keyword is terminated by a "/".

Table 9.11: LSLTWNUM Keyword Description

Example

The example below sets three LSLTWNUM regions for the model.

```
--
--        ARRAY       CONSTANT      ---------- BOX ---------
--                                  I1  I2   J1  J2   K1  K2
EQUALS
          LSLTWNUM    1             1*  1*   1*  1*   1*  1* / SET REGION 1
          LSLTWNUM    2             1   2    1   2    1   1  / SET REGION 2
          LSLTWNUM    3             1   2    1   2    2   2  / SET REGION 3
/
```

9.3.39 LSNUM – DEFINE THE LOW SALT OIL WET SATURATION TABLE REGION NUMBERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The LSNUM keyword defines the saturation tables (relative permeability and capillary pressure tables) region numbers for each grid block, as such there must be one entry for each cell in the model. The region number specifies which set of relative permeability tables (SWFN, SOF3 and related keywords) are used to calculate the relative permeability and capillary pressure in a grid block. The keyword should only be used if the Low Salinity option for the Brine model has been activated in the RUNSPEC section using the LOWSALT keyword.

The oil wet curves are calculated as a weighted average of the low salinity saturation tables (allocated by this keyword) and the high salinity oil wet saturation tables (allocated by the SATNUM keyword), using the weights provided by the LSALTFNC keyword in the PROPS section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description	Default
1	LSNUM	LSNUM defines an array of positive integers assigning a grid cell to a particular saturation table region. The maximum number of LSNUM regions is set by the NTSFUN variable on the TABDIMS keyword in the RUNSPEC section.	1

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) If a cell is not assigned a LSNUM region then the default value will be used.
3) The keyword is terminated by a "/".

Table 9.12: LSNUM Keyword Description

If the Surfactant Wettability option have been activated by the SURFACTW keyword, the LSNUM tables correspond to the immiscible low salinity curves.

Example

The example below sets three LSNUM regions for the model.

```
--
--        ARRAY       CONSTANT     ---------- BOX ---------
--                                 I1  I2   J1  J2   K1  K2
EQUALS
          LSNUM       1            1*  1*   1*  1*   1*  1* / SET REGION 1
          LSNUM       2            1   2    1   2    1   1  / SET REGION 2
          LSNUM       3            1   2    1   2    2   2  / SET REGION 3
/
```

9.3.40 LWSLTNUM – DEFINE THE LOW SALT OIL WET SATURATION TABLE REGION NUMBERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The LWSLTNUM keyword defines the saturation tables (relative permeability and capillary pressure tables) region numbers for each grid block, as such there must be one entry for each cell in the model. The region number specifies which set of relative permeability tables (SWFN, SOF3 and related keywords) are used to calculate the relative permeability and capillary pressure in a grid block. The keyword should only be used if the Low Salinity option for the Brine model has been activated by the LOWSALT kwyword in the RUNSPEC section.

The oil wet curves are calculated as a weighted average of the low salinity saturation tables (allocated by this keyword) and the high salinity oil wet saturation tables (allocated by the SATNUM keyword in the REGIONS section), using the weights provided by the LSALTFNC keyword in the PROPS section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description	Default
1	LWSLTNUM	LWSLTNUM defines an array of positive integers assigning a grid cell to a particular saturation table region. The maximum number of LSLTWNUM regions is set by the NTSFUN variable on the TABDIMS keyword in the RUNSPEC section.	1

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) If a cell is not assigned a LWSLTNUM region then the default value will be used.
3) The keyword is terminated by a "/".

Table 9.13: LWSLTNUM Keyword Description

Example

The example below sets three LWSLTNUM regions for the model.

```
--
--        ARRAY       CONSTANT      ---------- BOX ---------
--                                  I1  I2   J1  J2   K1  K2
EQUALS
          LWSLTNUM    1             1*  1*   1*  1*   1*  1* / SET REGION 1
          LWSLTNUM    2             1   2    1   2    1   1  / SET REGION 2
          LWSLTNUM    3             1   2    1   2    2   2  / SET REGION 3
/
```

9.3.41 LWSNUM – DEFINE THE LOW SALT WATER WET SATURATION TABLE REGION NUMBERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The LWSNUM keyword defines the saturation tables (relative permeability and capillary pressure tables) region numbers for each grid block, as such there must be one entry for each cell in the model. The region number specifies which set of relative permeability tables (SWFN, SOF3 and related keywords) are used to calculate the relative permeability and capillary pressure in a grid block. The keyword should only be used if the Low Salinity option for the Brine model and the Surfactant Wettability option have been activated by the LOWSALT and SURFACTW keywords, respectively, in the RUNSPEC section.

The water wet curves are calculated as a weighted average of the low salinity saturation tables (allocated by this keyword) and the high salinity water wet saturation tables (allocated by the HWSNUM keyword), using the weights provided by the LSALTFNC keyword in the PROPS section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description	Default
1	LWSNUM	LWSNUM defines an array of positive integers assigning a grid cell to a particular saturation table region. The maximum number of LWSNUM regions is set by the NTSFUN variable on the TABDIMS keyword in the RUNSPEC section.	1

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) If a cell is not assigned a LWSNUM region then the default value will be used.
3) The keyword is terminated by a "/".

Table 9.14: LWSNUM Keyword Description

The HWSNUM allocated tables correspond to the immiscible high salinity water wet curves.

Example

The example below sets three LWSNUM regions for the model.

```
--
--        ARRAY       CONSTANT      ---------- BOX ---------
--                                  I1  I2   J1  J2   K1  K2
EQUALS
          LWSNUM      1             1*  1*   1*  1*   1*  1* / SET REGION 1
          LWSNUM      2             1   2    1   2    1   1  / SET REGION 2
          LWSNUM      3             1   2    1   2    2   2  / SET REGION 3
/
```

9.3.42 MESSAGE – OUTPUT USER MESSAGE

The MESSAGE keyword outputs a user message to the terminal, as well as to the print (*.PRT) and debug (*.DBG) files. Note this is different to the MESSAGES keyword, that defines OPM Flows message print limits and stop limits generated by the simulator.

See *MESSAGE – Output User Message* in the GLOBAL section for a full description.

9.3.43 MESSAGES – DEFINE MESSAGE PRINT LIMITS AND STOP LIMITS

The MESSAGES keyword defines the print and stops levels for various messages. The "print limits" set the maximum number of messages that will be printed, after which no more messages will be printed and the "stop limits" terminate the run when these limits are exceeded. There are six levels of message that increase in severity from informative all the way to programming errors, as outlined in Error: Reference source not found.

See *MESSAGES – Define Message Print Limits and Stop Limits* in the GLOBAL section for a full description.

9.3.44 MISNUM – DEFINE THE MISCIBILITY REGION NUMBERS

Description

The MISNUM keyword defines the miscibility region number mixing tables as defined by the TLMIXPAR keyword in the PROPS section, for when the miscibility option has been activated by the MISCIBLE keyword in the RUNSPEC section. MISNUM also allocates miscible residual oil saturation versus water saturation tables (SORWMIS keyword in the PROPS section) used to calculate the relative permeability and PVT properties for a grid cell.

Note that although this keyword can only be used when the miscibility option is active, it is not necessary to use this keyword even if the MISCIBLE keyword in the RUNSPEC has been activated as the default value of one will be applied to all grid blocks. Secondly, a value of zero for a grid cell results in immiscible fluids in that grid cell.

No.	Name	Description	Default
1	MISNUM	MISNUM defines an array of positive integers greater than or equal to zero, that assign a grid cell to a particular table of mixing parameters as defined by the TLMIXPAR and SORWMIS keywords. A value of zero sets the fluids within a grid cell to be immiscible. The maximum number of MISNUM regions is set by the NTMIS variable on the MISCIBLE keyword in the RUNSPEC section.	1

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) If a cell is not assigned a MISNUM region then the default value of one will be used.
3) The keyword is terminated by a "/".

Table 9.15: MISNUM Keyword Description

See also the TLMIXPAR and SORWMIS keyword in the PROPS section.

Example

The example below sets three MISNUM regions in the model on a layer by layer basis, using the EQUALS keyword.

```
--
--        ARRAY        CONSTANT      ---------- BOX ---------
--                                    I1   I2    J1   J2    K1   K2
EQUALS
          MISNUM       1             1*   1*    1*   1*    1    12 / SET REGION 1
          MISNUM       2             1*   1*    1*   1*    13   55 / SET REGION 2
          MISNUM       3             1*   1*    1*   1*    56  120 / SET REGION 3
/
```

9.3.45 MULTIPLY – MULTIPLY A SPECIFIED ARRAY BY A CONSTANT

The MULTIPLY keyword multiplies a specified array or part of an array by a constant. The constant can be real or integer depending on the array type; however, the arrays that can be operated on is dependent on which section the keyword is being used.

See *MULTIPLY – Multiply a Specified Array by a Constant* in the GRID section for a full description.

9.3.46 MULTIREG – MULTIPLY AN ARRAY BY A CONSTANT BASED ON A REGION NUMBER

The MULTIREG keyword multiplies an array or part of an array by a constant for cells with a specific region number. The region number array can be FLUXNUM, MULTNUM or OPERNUM and these arrays must be defined and be available before the MULTIREG keyword is read by the simulator. The constant can be real or integer depending on the property array type; however, the property arrays that can be operated on is dependent on which section the MULTIREG keyword is being used.

See *MULTIREG – Multiply an Array by a Constant based on a Region Number* in the GRID section for a full description.

9.3.47 NOECHO – DEACTIVATE ECHOING OF USER INPUT FILES TO THE PRINT FILE

Turns off echoing of all the input files to the print file. Note by default echoing of the inputs files is active. but can subsequently be switched off by the NOECHO activation keyword.

See *NOECHO – Deactivate Echoing of User Input Files to the Print File* in the GLOBAL section for a full description.

9.3.48 NOWARN – DEACTIVATE WARNING MESSAGES

Turns off warning messages to be printed to the print file; note that this keyword is deactivated by default and can subsequently be switched off by the WARN activation keyword. The warning messages may be turned on and off using keywords WARN and NOWARN.

See *NOWARN – Deactivate Warning Messages* in the GLOBAL section for a full description.

9.3.49 OPERATE – DEFINE MATHEMATICAL OPERATIONS ON ARRAYS

This keyword, OPERATE, defines mathematical operations on property arrays (NTG, PORO etc.) and optionally using another property array as input to the function. The keyword allows for various mathematical functions and their associated variables to be defined and applied to the selected array data. Input constants can be real or integer depending on the array type; however, the arrays that can be operated on is dependent on which section the keyword is being used.

See *OPERATE – Define Mathematical Operations on Arrays* in the GRID section for a full description.

9.3.50 OPERATER – DEFINE MATHEMATICAL OPERATIONS ON ARRAYS BY REGION

This keyword defines the mathematical operations on arrays for specific regions in the commercial simulator and is currently not supported by OPM Flow. However, similar functionality is provided by the ADD and MULTIPLY keywords.

See *OPERATER – Define Mathematical Operations on Arrays by Region* in the GRID section for a full description.

9.3.51 OPERNUM – DEFINE REGIONS FOR MATHEMATICAL OPERATIONS ON ARRAYS

This keyword defines the OPERATE region numbers for each grid block. The OPERATE keyword defines mathematical operations on arrays in the commercial simulator and is currently not supported by OPM Flow. However, similar functionality is provided by the ADD and MULTIPLY keywords combined with MULTNUM region array.

See *OPERNUM – Define Regions for Mathematical Operations on Arrays* in the GRID section for a full description.

9.3.52 PENUM – DEFINE THE PETRO-ELASTIC REGION NUMBERS

Description

The PENNUM keyword defines the petro-elastic region number for each grid block that is used to assign the of petro-elastic coefficients, bulk modulus functions and shear modulus functions as defined by the PECOEFS, PEKTAB and PEGTAB series of keywords in the PROPS section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

9.3.53 PLMIXNUM – DEFINE THE POLYMER REGION NUMBERS

Description

The PLMIXNUM keyword defines the polymer region number for each grid block that is used to assign the mixing tables as well as the maximum polymer and salt concentrations, as defined by the PLMIXPAR and PLYMAX keywords in the PROPS section, for when the polymer option has been activated by the POLYMER keyword in the RUNSPEC section.

The maximum polymer concentration and the associated salt concentration are declared on the PLYMAX keyword.

No.	Name	Description	Default
1	PLMIXNUM	PLMIXNUM defines an array of positive integers greater than or equal to one, that assign a grid cell to a particular table of mixing parameters as defined by the PLMIXPAR and PLYMAX keywords. The maximum number of PLMIXNUM regions is set by the NPLMIX variable on the REGDIMS keyword in the RUNSPEC section.	1

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) If a cell is not assigned a PLMIXNUM region then the default value of one will be used.
3) The keyword is terminated by a "/".

Table 9.16: PLMIXNUM Keyword Description

See also the PLYADS, PLYADSS, PLYDHLF, PLYMAX, PLYROCK, PLYSHEAR, PLYSHLOG and PLYVISC keywords in the PROPS section.

Example

The example below sets three PLMIXNUM regions in the model on a layer by layer basis, using the EQUALS keyword.

```
--
--        ARRAY       CONSTANT      ---------- BOX ---------
--                                  I1  I2   J1  J2   K1  K2
EQUALS
          PLMIXNUM      1           1*  1*   1*  1*   1   12 / SET REGION 1
          PLMIXNUM      2           1*  1*   1*  1*   13  55 / SET REGION 2
          PLMIXNUM      3           1*  1*   1*  1*   56 120 / SET REGION 3
/
```

9.3.54 PVTNUM – DEFINE THE PVT REGIONS

Description

The PVTNUM keyword defines the PVT region numbers for each grid block, as such there must be one entry for each cell in the model. The region number specifies which set of PVT tables (DENSITY, PVDG, PVDO, PVTG, PVTO, PVCO, PVTW and ROCK) are used to calculate the PVT properties in a grid block.

No.	Name	Description	Default
1	PVTNUM	PVTNUM defines an array of positive integers assigning a grid cell to a particular PVT region. The maximum number of PVTNUM regions is set by the NTPVT variable on the TABDIMS keyword in the RUNSPEC section.	1

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) The EQULNUM and PVTNUM arrays need to be consistent, that is the all cells with the same PVTNUM can only belong to one EQLNUM region.
3) If a cell is not assigned a PVTNUM region then the default value will be used.
4) The keyword is terminated by a "/".

Table 9.17: PVTNUM Keyword Description

Note

Care should be taken that cells in different PVTNUM regions are not in communication, since the fluid properties are associated with a cell. If for example, a rbbl or a rm3 of oil flows from PVTNUM region 1 to PVTNUM region 2, then the oil properties of that oil will change from the PVT 1 data set to the PVT data set 2. This will result in material balance errors, that may or may not cause numerical issues.

To avoid this one should use the MULTNUM (or FLUXNUM, or OPERNUM) array with the MULTREGT array to ensure that the various PVTNUM regions are not in communication.

Examples

The example below sets three PVTNUM regions for a 4 x 5 x 2 model.

```
--
--       DEFINE PVTNUM REGION FOR ALL CELLS
--
PVTNUM
         2  2  1  1  2  2  1  1  1  1  1  1  1  1  1  1  1  1  1  1
         3  3  1  1  3  3  1  1  1  1  1  1  1  1  1  1  1  1  1  1
/
```

Alternatively the EQUALS keyword could be employed to accomplish the same task, that is:

```
--
--       ARRAY       CONSTANT     ---------- BOX ---------
--                                I1  I2   J1  J2   K1  K2
EQUALS
         PVTNUM      1            1*  1*   1*  1*   1*  1* / SET REGION 1
         PVTNUM      2            1   2    1   2    1   1  / SET REGION 2
         PVTNUM      3            1   2    1   2    2   2  / SET REGION 3
/
```

There third example shows how to ensure the various PVT regions are isolated. First of all define the MULTNUM array in the GRID section and ensure all the regions are isolated.

```
-- ======================================================================
--
-- GRID SECTION
--
-- ======================================================================
GRID
--
--       ARRAY       CONSTANT     ---------- BOX ---------
--                                I1  I2   J1  J2   K1  K2
EQUALS
         MULTNUM     1            1*  1*   1*  1*   1*  1* / SET REGION 1
         MULTNUM     2            1   2    1   2    1   1  / SET REGION 2
         MULTNUM     3            1   2    1   2    2   2  / SET REGION 3
/
--
--       SET TRANSMISSIBILITES ACROSS DIFFERENT RESERVOIRS TO ZERO TO ISOLATE
--       RESERVOIRS
--
--       REGION   REGION   TRANS   DIREC   NNC    REGION ARRAY
--       FROM     TO       MULT    OPT     OPTS   M / F / O
MULTREGT
         1*       1*       0.0     1*     'ALL'   M        / ALL REGIONS SEALED
/
```

Then in the REGIONS section copy the MULTNUM array to the PVTNUM array.

```
-- ======================================================================
--
-- REGIONS SECTION
--
-- ======================================================================
REGIONS

--
--       COPY AN ARRAY TO ANOTHER ARRAY BASED ON A REGION NUMBER
--
--       ARRAY     ARRAY     REGION   REGION ARRAY
--       FROM      TO        NUMBER    M / F / O
COPYREG
         MULTNUM   PVTNUM    1         M                  / COPY MULT TO PVT 1
         MULTNUM   PVTNUM    2         M                  / COPY MULT TO PVT 2
         MULTNUM   PVTNUM    3         M                  / COPY MULT TO PVT 3
/
```

All the separate PVT regions are now isolated.

9.3.55 PYEND – END THE DEFINITION OF A PYINPUT SECTION

The PYINPUT and PYEND keywords are a part of OPM Flow's Python scripting facility that processes standard Python commands that can be used to manipulate and define the simulators input parameters during processing of the input deck. The main purpose of the facility is to script the construction of the various keywords. A PYINPUT Definition Section is terminated by a PYEND keyword on a separate single line.

See *PYEND – End the Definition of a PYINPUT Section* in the GRID section for a full description.

9.3.56 PYINPUT – DEFINE THE START OF A PYINPUT SECTION

The PYINPUT and PYEND keywords are a part of OPM Flow's Python scripting facility that processes standard Python commands that can be used to manipulate and define the simulators input parameters during processing of the input deck. The main purpose of the facility is to script the construction of the various keywords used by the simulator. PYINPUT declares the start of a PYINPUT Definition Section on a single separate line, which is then followed by various standard Python commands, with one command per line.

See *PYINPUT – Define the Start of a PYINPUT Section* in the GRID section for a full description.

9.3.57 REFINE – START THE DEFINITION OF A LOCAL GRID REFINEMENT

The REFINE keyword defines the start of a Cartesian or radial local grid refinement ("LGR") definition that sets the properties of the selected LGR. The keyword is then followed by the property keywords associated with the section where the keyword is being invoked. For example, if the REFINE keyword is used in the GRID section then most of the keywords in that section can be used to set the grid properties for the LGR.

See *REFINE – Start the Definition of a Local Grid Refinement* in the GRID section for a full description.

9.3.58 REGIONS - DEFINE THE START OF THE REGIONS SECTION OF KEYWORDS

Description

The REGIONS activation keyword marks the end of the PROPS section and the start of the REGIONS section that defines how various fluid and rock property data defined in the PROPS section are allocated to the individual cells in the model.

There is no data required for this keyword.

Example

```
-- ================================================================================
--
-- REGIONS SECTION
--
-- ================================================================================
REGIONS
```

The above example marks the end of the PROPS section and the start of the REGIONS section in the OPM Flow data input file.

9.3.59 RESIDNUM – DEFINE VERTICAL EQUILIBRIUM RESIDUAL FLOW REGION NUMBERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The RESIDNUM keyword defines the Vertical Equilibrium ("VE") residual flow calculation saturation tables (relative permeability and capillary pressure tables) region numbers for each grid block, as such there must be one entry for each cell in the model. The region number specifies which set of relative permeability tables (SGFN, SWFN, SOF2, SOF3, SOF32D, SGOF, SLGOF and SWOF) are used to calculate the relative permeability and capillary pressure in a grid block. The keyword should only be used if the Vertical Equilibrium option has been invoked via the VE keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description	Default
1	RESIDNUM	RESIDNUM defines an array of positive integers assigning a grid cell to a particular saturation table region. The maximum number of RESIDNUM regions is set by the NTSFUN variable on the TABDIMS keyword in the RUNSPEC section.	1

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) If a cell is not assigned a RESIDNUM region then the default value will be used.
3) The keyword is terminated by a "/".

Table 9.18: RESIDNUM Keyword Description

Note that any capillary pressure data on the relative permeability tables (SGFN, SWFN, SOF2, SOF3, SOF32D, SGOF, SLGOF and SWOF) will be ignored by the VE option.

Example

The example below sets three RESIDNUM regions for the model, by first setting all values to one, then setting layers 2 to 10 to two, and finally setting layers 30 to 50 to three.

```
--
--        ARRAY        CONSTANT      ---------- BOX ---------
--                                    I1  I2    J1  J2    K1  K2
EQUALS
          SATNUM       1             1*  1*    1*  1*    1*  1* / SET REGION 1
          SATNUM       2             1   2     1   2     2   10 / SET REGION 2
          SATNUM       3             1   2     1   2     30  50 / SET REGION 3
/
```

9.3.60 ROCKNUM – DEFINE ROCK COMPACTION TABLE REGION NUMBERS

Description

The ROCKNUM keyword defines the rock compaction table region numbers for each grid block, as such there must be one entry for each cell in the model. The region number specifies which set of rock compaction tables defined by the ROCKTAB keyword are used to calculate the rock compaction in a grid block.

No.	Name	Description	Default
1	ROCKNUM	ROCKNUM defines an array of positive integers assigning a grid cell to a particular rock compaction table region. The maximum number of ROCKNUM regions is set by the NTROCC variable on the ROCKCOMP keyword in the RUNSPEC section.	1

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) If a cell is not assigned a ROCKNUM region then the default value will be used.
3) The keyword is terminated by a "/".

Table 9.19: ROCKNUM Keyword Description

Examples

The example below sets three ROCKNUM regions for a 4 x 5 x 2 model.

```
--
--        DEFINE ROCKNUM REGION FOR ALL CELLS
--
ROCKNUM
          2  2  1  1  2  2  1  1  1  1  1  1  1  1  1  1  1  1  1  1
          3  3  1  1  3  3  1  1  1  1  1  1  1  1  1  1  1  1  1  1
/
```

Alternatively the EQUALS keyword could be employed to accomplish the same task, that is:

```
--
--        ARRAY       CONSTANT     ---------- BOX ---------
--                                 I1  I2   J1  J2   K1  K2
EQUALS
          ROCKNUM     1            1*  1*   1*  1*   1*  1* / SET REGION 1
          ROCKNUM     2            1   2    1   2    1   1  / SET REGION 2
          ROCKNUM     3            1   2    1   2    2   2  / SET REGION 3
/
```

9.3.61 RPTREGS – DEFINE REGIONS SECTION REPORTING

Description

This keyword defines the data in the REGIONS section that is to be printed to the output print file in human readable format. The keyword has two distinct forms, the first of which consists of the keyword followed by a series of integers on the next line indicating the data to be printed (see the first example). This is the original formal in the commercial simulator and was subsequently superseded by the second format. The second format consists of the keyword followed by a series of character strings that indicate the data to be printed. In most cases the character string is the keyword used to load the data in the OPM Flow input deck, for example FIPNUM for the fluid in-place array. Its is anticipated that OPM Flow will eventually support the functionality of the second format only, the first format although recognized will be completely ignored.

No.	Name	Description	Default
1	EQLNUM	Print the equilibration region array.	N/A
2	FIPNUM	Print the fluid in-place array.	N/A
3	PVTNUM	Print the PVT table assignment array.	N/A
4	SATNUM	Print the saturation function (relative permeability) assignment array.	N/A
….		….	N/A

Notes:

1) The keyword is terminated by a "/".

Table 9.20: RPTREGS Keyword Description

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

Note

This keyword has the potential to produce very large print files that some text editors may have difficulty loading, coupled with the fact that reviewing the data in this format is very cumbersome.

A more efficient solution is to load the *.INIT file into OPM ResInsight to view the data graphically, this also has the benefit of being able to filter the grid based on I, J, K ranges and grid properties.

Examples

The first example shows the original format of this keyword; although the keyword and format are recognized by OPM Flow, the format is ignored and is unlikely to be implemented in in the simulator.

```
--
--       DEFINE REGIONS SECTION REPORT OPTION (ORIGINAL FORMAT)
-
RPTREGS
         1       2*0      1       3*1                                  /
```

The next example shows the second format of the keyword which may be supported in a future release of OPM Flow.

```
--        DEFINE REGIONS SECTION REPORT OPTIONS
--
RPTREGS
          FIPMUM    EQLNUM    PVTNUM    SATNUM                                  /
```

9.3.62 SATNUM – DEFINE THE SATURATION TABLE REGION NUMBERS

Description

The SATNUM keyword defines the saturation tables (relative permeability and capillary pressure tables) region numbers for each grid block, as such there must be one entry for each cell in the model. The region number specifies which set of relative permeability tables (SGFN, SWFN, SOF2, SOF3, SOF32D, SGOF, SLGOF and SWOF) are used to calculate the relative permeability and capillary pressure in a grid block.

No.	Name	Description	Default
1	SATNUM	SATNUM defines an array of positive integers assigning a grid cell to a particular saturation table region. The maximum number of SATNUM regions is set by the NTSFUN variable on the TABDIMS keyword in the RUNSPEC section.	1
Notes: 1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement. 2) If a cell is not assigned a SATNUM region then the default value will be used. 3) The keyword is terminated by a "/".			

Table 9.21: SATNUM Keyword Description

Examples

The example below sets three SATNUM regions for a 4 x 5 x 2 model.

```
--
--        DEFINE SATNUM REGIONS FOR ALL CELLS
--
SATNUM
          2  2  1  1  2  2  1  1  1  1  1  1  1  1  1  1  1  1  1  1
          3  3  1  1  3  3  1  1  1  1  1  1  1  1  1  1  1  1  1  1
/
```

Alternatively the EQUALS keyword could be employed to accomplish the same task, that is:

```
--
--        ARRAY       CONSTANT      ---------- BOX ---------
--                                  I1  I2   J1  J2   K1  K2
EQUALS
          SATNUM      1             1*  1*   1*  1*   1*  1* / SET REGION 1
          SATNUM      2             1   2    1   2    1   1  / SET REGION 2
          SATNUM      3             1   2    1   2    2   2  / SET REGION 3
/
```

9.3.63 SKIP – ACTIVATE SKIPPING OF ALL KEYWORDS AND INPUT DATA

The SKIP keyword turns on skipping of keywords until the ENDSKIP activation keyword is encountered. All keywords and between the SKIP and ENDSKIP keywords are ignored by OPM Flow.

See *SKIP – Activate Skipping of All Keywords and Input Data* in the GLOBAL section for a full description.

9.3.64 SKIP100 – ACTIVATE SKIPPING OF BLACK-OIL KEYWORDS AND INPUT DATA

This keyword turns on skipping of black-oil keywords until the ENDSKIP activation keyword is encountered. All black-oil keywords between the SKIP100 and ENDSKIP keywords are ignored by OPM Flow.

See *SKIP100 – Activate Skipping of Black-Oil Keywords and Input Data* in the GLOBAL section for a full description.

9.3.65 SKIP300 – ACTIVATE SKIPPING OF "COMPOSITIONAL" KEYWORDS AND INPUT DATA

Turns on skipping of "compositional" keywords until the ENDSKIP activation keyword is encountered. All "compositional" keywords between the SKIP300 and ENDSKIP keywords are ignored by OPM Flow.

See *Error: Reference source not found* in the GLOBAL section for a full description.

9.3.66 SURFNUM – DEFINE THE SURFACTANT MISCIBLE SATURATION TABLE REGION NUMBERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The SURFNUM keyword defines the saturation tables (relative permeability and capillary pressure tables) region numbers for each grid block, as such there must be one entry for each cell in the model. The region number specifies which set of oil-water relative permeability tables (SWFN, SOF2, SOF3, and SWOF) are used to calculate the relative permeability and capillary pressure in a grid block. In this case the SURFNUM allocated tables assume that oil and water are miscible, whereas the SATNUM allocated tables are used to allocate the immiscible saturation tables. To use this keyword the Surfactant option must have been activated by the SURFACT keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description	Default
1	SURFNUM	SURFNUM defines an array of positive integers assigning a grid cell to a particular saturation table region. The maximum number of SURFNUM regions is set by the NTSFUN variable on the TABDIMS keyword in the RUNSPEC section.	1

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) If a cell is not assigned a SURFNUM region then the default value will be used.
3) The keyword is terminated by a "/".

Table 9.22: SURFNUM Keyword Description

Example

The example below sets three SURFNUM for the model.

```
--
--        ARRAY       CONSTANT      ---------- BOX ---------
--                                  I1  I2   J1  J2   K1  K2
EQUALS
          SURFNUM     1             1*  1*   1*  1*   1*  1* / SET REGION 1
          SURFNUM     2             1*  1*   1*  1*   1   1  / SET REGION 2
          SURFNUM     3             1*  1*   1*  1*   2   2  / SET REGION 3
/
```

9.3.67 SURFWNUM – DEFINE THE SATURATION TABLE REGION NUMBERS (HIGH SALINITY AND WATER WET)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The SURFWNUM keyword defines the saturation tables (relative permeability and capillary pressure tables) region numbers for each grid block, as such there must be one entry for each cell in the mode, for when the Surfactant Wettability option has been selected. The keyword may also be used with the Low Salt Brine option, in this case the water wet curves are calculated as a function of the low and high water wet salinity curves. The region number specifies which set of relative permeability tables are used to calculate the relative permeability and capillary pressure in a grid block. Note that the keyword is obligatory if the SURFACTW keyword in the RUNSPEC section has been used to invoke the Surfactant Wettability option.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description	Default
1	SURFWNUM	SURFWNUM defines an array of positive integers assigning a grid cell to a particular saturation table region. The maximum number of SURFWNUM regions is set by the NTSFUN variable on the TABDIMS keyword in the RUNSPEC section.	1

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) If a cell is not assigned a SATNUM region then the default value will be used.
3) The keyword is terminated by a "/".

Table 9.23: SURFWNUM Keyword Description

Example

The example below sets three SURFWNUM regions for the model.

```
--
--        ARRAY       CONSTANT      ---------- BOX ---------
--                                  I1  I2   J1  J2   K1  K2
EQUALS
          SURFWNUM    1             1*  1*   1*  1*   1*  1* / SET REGION 1
          SURFWNUM    2             1   2    1   2    1   1  / SET REGION 2
          SURFWNUM    3             1   2    1   2    2   2  / SET REGION 3
/
```

9.3.68 TNUM – DEFINE PASSIVE TRACER CONCENTRATION REGIONS

Description

The TNUM keyword defines the regions associated with the series of tracers associated with a phase (oil, water, or gas) in the model. The maximum number of tracers for each phase are declared on the TRACER keyword in the RUNSPEC section. Unlike other keywords, the TNUM keyword must be concatenated with the phase and the name of the tracer declared by TRACER keyword in the PROPS section. Table 9.24 outlines the format of the TNUM keyword name.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description	Default
1	TNUM	A four letter character equal to TNUM that is the root keyword name for this data set array.	None
2	PHASE	A one letter character string that must be equal to F or S, that is concatenated to TNUM. The letter F states that the tracer is for the "free" phase, for example oil or water, as well as gas cap gas (free gas). The letter S signifies that the tracer is a "solution" phase tracer, for example gas dissolved in oil (as activated by the DISGAS keyword in the RUNSPEC section), or condensate (vaporized oil) in the gas (as per the VAPOIL keyword in the RUNSPEC section). Note tracers that are defined by the letter S to be in the "solution" phase, must also be initialized by the "free" phase as well.	None
3	NAME	A three letter character string defining the tracer's name, which is concatenate to TNUM and PHASE to given the full name of the keyword Note it is best to void names beginning with the letters F, S, and T as these names may great naming issues in post-processing software.	None

Table 9.24: TNUM Keyword Name Format

Following the declaration of the full keyword name, TNUMPHASENAME, the keyword is followed by the data as outlined below.

No.	Name	Description	Default
1	TNUMREG	TUNREG defines an array of positive integers assigning a grid cell to a particular tracer table region. The maximum number of TNUMREG regions is set by the NTTRVD variable on the EQLDIMS keyword in the RUNSPEC section.	1

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) If a cell is not assigned a TNUMREG region then the default value will be used.
3) The keyword is terminated by a "/".

Table 9.25: TNUM Keyword Data Description

See also the TRACER keyword in the PROPS section and the TBLK keyword in the SOLUTION section.

Example

First define four passive tracers one for a free gas, one for dissolved gas, one for oil and one to track the water.

```
--
--        DEFINE TRACER NAMES
--
--        TRACER   TRACER
--        NAME     PHASE
--        ------   ------
TRACER
         'GCG'     'GAS'                                  / GAS CAP GAS
         'DGS'     'GAS'                                  / DISOLVED GAS
         'OIL'     'OIL'                                  / OIL
         'WAT'     'WAT'                                  / WAT
/
```

Given a 100 x 100 x 5 grid with DISGAS activated in the RUNSPEC section, then the following TNUM keywords define the various tracer regions given that NTTRVD equals four on the EQLDIMS keyword in the RUNSPEC section.

```
--
--        DEFINE PASSIVE TRACER CONCENTRATION REGIONS
--
TNUMFGCG
          1000*1
          1000*2
          1000*2
          1000*2
          1000*2
/
TNUMSDGS
          1000*1
          1000*1
          1000*1
          1000*1
          1000*1
/
TNUMFOIL
          1000*3
          1000*3
          1000*3
          1000*3
          1000*3
/
TNUMFWAT
          1000*4
          1000*4
          1000*4
          1000*4
          1000*4
/
```

The keyword name is derived from the TNUM keyword, plus either F or S, plus the tracer name declared in the TRACER keyword. For example for the gas cap (free gas) this would be TNUM+F+GAS to give the TNUMFGAS keyword. And for the dissolved (solution) gas this would be TNUM+S+DGS resulting in the TNUMSDGS keyword.

9.3.69 TRKPF – DEFINE PARTITIONED TRACER REGIONS

Description

The TRKPF keyword defines the regions associated with the series of partition tracers and the partitioning tables allocated to grid blocks in the model, for when the Partitioned Tracer option has been enabled by the PARTTRAC keyword in the RUNSPEC section. The maximum number of tracers for each phase are declared on the TRACERS keyword in the RUNSPEC section. Unlike other keywords, the TRKPF keyword must be concatenated with the name of the tracer declared by TRACER keyword in the PROPS section. Table 9.26 outlines the format of the TRKPF keyword name.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description	Default
1	TRKPF	A five letter character string equal to TRKPF that is the root keyword name for this data set array.	None
2	NAME	A three letter character string defining the tracer's name, as declared by the TRACER keyword, which is concatenate to TRKPF to given the full name of the keyword Note it is best to void names beginning with the letters F, S, and T as these names may great naming issues in post-processing software.	None

Table 9.26: TRKPF Keyword Name Format

Following the declaration of the full keyword name, TRKPFNAME, the keyword is followed by the data as outlined below.

No.	Name	Description	Default
1	TRKPFREG	TRKPFREG defines an array of positive integers assigning a grid cell to a particular tracer table region. The maximum number of TRKPFREG regions is set by the NTTRVD variable on the EQLDIMS keyword in the RUNSPEC section.	1
Notes: 1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement. 2) If a cell is not assigned a TRKPFREG region then the default value will be used. 3) The keyword is terminated by a "/".			

Table 9.27: TRKPF Keyword Data Description

See also the TRACER and TRACERKP keywords in the PROPS section and the TBLK keyword in the SOLUTION section.

Example

First define one mult-partitioned tracer for the water phase.

```
--
--        DEFINE TRACER NAMES
--
--        TRACER   TRACER   TRACER  PARTITION  NUM   ADSOR
--        NAME     PHASE    VOLUME  PHASE      K(P)  PHASE
--        ------   ------   ------  ---------  ----  -----
TRACER
         'WAT'     'WAT'    1*      MULT       2     ALL    / WAT
/
```

Then for a given a 100 x 100 x 5 grid assign the partitioned tracer regions and K(P) tables, based on two regions.

```
--
--        DEFINE PARTITIONED TRACER REGIONS
--
TRKPFWAT
          1000*1
          1000*1
          1000*2
          1000*2
          1000*2
/
```

The keyword name is derived from the TRKPF keyword, plus the tracer name declared in the TRACER keyword, in this case the keyword name is TRKPFWAT.

9.3.70 WARN – ACTIVATE WARNING MESSAGES

Turns on warning messages to be printed to the print file (*.PRT); note that this keyword is activated by default and can subsequently be switched off by the NOWARN activation keyword. The warning messages may be turned on and off using keywords WARN and NOWARN. OPM Flow always prints error messages.

See *WARN – Activate Warning Messages* in the GLOBAL section for a full description.

9.3.71 WH2NUM – DEFINE WAG HYSTERESIS SATURATION TABLE REGION NUMBERS (TWO PHASE)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The WH2NUM keyword defines the two phase Water-Alternating-Gas ("WAG") hysteresis tables (relative permeability and capillary pressure tables) region numbers for each grid block, for when the hysteresis option has been activated by the WAGHYSTR variable on the SATOPTS keyword in the RUNSPEC section. The region number specifies which set of relative permeability tables (SGFN, SWFN, SOF2, SOF3, SOF32D, SGOF, SLGOF and SWOF) are used to calculate the relative permeability and capillary pressure in a grid block. Note that this keyword if the two phase water relative permeabilities WAG option.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description	Default
1	WH2NUM	WH2NUM defines an array of positive integers assigning a grid cell to a particular saturation table region. The maximum number of WH2NUM regions is set by the NTSFUN variable on the TABDIMS keyword in the RUNSPEC section.	Taken from cell allocated SATNUM

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) If a cell is not assigned a WH2NUM region then the default value will be used.
3) The keyword is terminated by a "/".

Table 9.28: WH2NUM Keyword Description

Example

The example below sets three WH2NUM regions for a model.

```
--
--        ARRAY       CONSTANT      ---------- BOX ---------
--                                  I1  I2   J1  J2   K1  K2
EQUALS
          WH2NUM      1             1*  1*   1*  1*   1*  1* / SET REGION 1
          WH2NUM      2             1   2    1   2    1   1  / SET REGION 2
          WH2NUM      3             1   2    1   2    2   2  / SET REGION 3
/
```

9.3.72 WH3NUM – DEFINE WAG HYSTERESIS SATURATION TABLE REGION NUMBERS (THREE PHASE)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The WH3NUM keyword defines the three phase Water-Alternating-Gas ("WAG") hysteresis tables (relative permeability and capillary pressure tables) region numbers for each grid block, for when the hysteresis option has been activated by the WAGHYSTR variable on the SATOPTS keyword in the RUNSPEC section. The region number specifies which set of relative permeability tables (SGFN, SWFN, SOF2, SOF3, SOF32D, SGOF, SLGOF and SWOF) are used to calculate the relative permeability and capillary pressure in a grid block. Note that this keyword if the three phase water relative permeabilities WAG option.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description	Default
1	WH3NUM	WH3NUM defines an array of positive integers assigning a grid cell to a particular saturation table region. The maximum number of WH3NUM regions is set by the NTSFUN variable on the TABDIMS keyword in the RUNSPEC section.	Taken from cell allocated SATNUM

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) If a cell is not assigned a WH3NUM region then the default value will be used.
3) The keyword is terminated by a "/".

Table 9.29: WH3NUM Keyword Description

Example

The example below sets three WH3NUM regions for a model.

```
--
--        ARRAY       CONSTANT      ---------- BOX ---------
--                                  I1  I2   J1  J2   K1  K2
EQUALS
          WH3NUM      1             1*  1*   1*  1*   1*  1* / SET REGION 1
          WH3NUM      2             1   2    1   2    1   1  / SET REGION 2
          WH3NUM      3             1   2    1   2    2   2  / SET REGION 3
/
```

CHAPTER 10: SOLUTION SECTION

10.1 INTRODUCTION

The SOLUTION section defines the initial conditions for the model, that is the conditions at time zero, before any flow occurs in the reservoir. As such, sufficient information from the previous sections is required for the simulator to calculate the initial saturation and pressure distributions through out the model, such that the fluids are in gravity/hydrostatic equilibrium. This means that if the simulator is run for a period of time without any production or injection then the there should be no saturation or pressure changes in the model. In fact this "test" of running the simulator for a few years without any production or injection should always be performed to ensure that the model is indeed in hydrostatic equilibrium.

There are basically three methods to initialize a model:

1) **Equilibrium** Initialization: This is the primary method to initialize a model, and is in fact, the only manner that should be used to set the model at equilibrium conditions, unless a restart run is being initialized. Here, OPM Flow uses the previously entered PVT data and the fluid saturation tables (SWFN, SGFN, etc.), combined with the EQUIL keyword in the SOLUTION section to compute the capillary and fluid gradients, and hence the fluid saturations and fluid densities in each cell. The EQUIL keyword basically sets the pressure at a datum depth together with fluid contacts for an equilibrium region, from which all the required grid block properties can be calculated. Note that it not uncommon for hydrostatic equilibrium not be be achieved initially, due for example different PVT regions being in communication, aquifer properties incorrectly defined, etc. Which it is why it is important to run the simulator for a few years without any production or injection to ensure that the model is indeed in hydrostatic equilibrium, and to rectify any issues.

2) **Enumeration** Initialization: This is the option where one defines the pressures for all the grid cells together with the cells initial gas and water saturations, the oil saturation in this case is back calculated by the simulator using $S_o = 1.0 - S_g - S_w$. Thus, in three phase runs only two phase saturations need be entered and for two phase run only one saturation array needs to be provided. This type of initialization is rarely used in the industry due to the complexity of ensuring that the model is in gravity equilibrium; however, the facility has been used by some companies in the past, using in-house proprietary software, to generate suitable input to the simulator. Enumeration of course can also be used to initialize the model in non-equilibrium state, similar to the restart initialization described next.

3) **Restart** Initialization: This is a form of enumeration initialization but in this case the condition of gravity equilibrium is not applied; in other words the model is initialized at a point in time when after the model has produced or injected fluids. This type of initialization is very common, as it enables the run to "restart" from a previously run case. For example, once a model has been history matched, all the prediction cases can be "restarted" from the end of the final history match run, as oppose to running the cases from the start of the history match to the end of the predication cases. In some fields, the reservoir may not be in hydrostatic equilibrium, due for example, a pressure differential across the field caused by a re-charging aquifer. In these rare cases the model is initialized using the hydrostatic equilibrium approach but then run until the present day conditions match the measured present day conditions by adjusting the aquifer influx. The actual history match / predication runs are then restarted from the resulting present day restart file.

Equilibrium Initialization uses the PVT data and saturation tables entered in the PROPS section combined with data entered on the EQUIL keyword in the SOLUTION section. As capillary forces are generally small relative to viscous forces in field scale simulations, one may expect the errors introduced by use of the drainage capillary pressure curves to simulate an imbibition process to be small. Thus, it is common practice in field scale simulations to use the capillary pressure drainage curves in initializing a model, mainly as drainage curves are routinely measured in the laboratory, whereas imbibition curve are not. The exception of course is when hysteresis is applicable or when simulating small scale systems. If the hysteresis option has been activated via HYSTER variable on the SATOPTS keyword in the RUNSPEC section, then the simulator will use the appropriate imbibition capillary pressure data entered via the saturation table keywords (SWFN, SGFN etc.) in determining the fluid saturation.

The initial vertical distribution of fluids will be governed by the fluid densities and capillary forces, with a gas zone, a gas-oil transition zone, an oil zone, an oil-water transition zone, finally the water zone within a hydrostatic unit or region. Not all zones may be present in a hydrostatic unit, for example in dry gas reservoirs there will be only gas, gas-water transition and water zones. This is fundamental construct given the density of the in situ phases, there are rare exceptions to this principle; however, when there is an exception the more likely explanation is the various fluids are not in the same hydrostatic unit.

In the methodology outline below, it has been assumed that the initial oil pressure has been specified at some depth, z_{init} within the oil zone. Note it is not necessary that the pressure be specified within the oil zone, but it makes describing the analysis somewhat simpler. Thus we have:

$$p = p^{oil}_{init} \quad at \quad z = z_{init} \tag{10.1}$$

The oil pressure at any other point in the oil column can be calculated as shown below:

$$p_{oil} = p^{oil}_{init} + \gamma^{oil} \left(z - z_{init} \right) \tag{10.2}$$

Where ɣoil is the oil density and z is the given depth. We can then calculate the oil pressure at the Water-Oil Contact ("WOC"), that is p^{oil}_{woc}, from:

$$p^{oil}_{woc} = p^{oil}_{init} + \gamma^{oil} \left(z_{woc} - z_{init} \right) \tag{10.3}$$

And employing the definition of water-oil capillary pressure, one can write an expression for the water pressure at the water-oil contact:

$$p^{water}_{woc} = p^{oil}_{woc} + p_{cwoc} \tag{10.4}$$

The water pressure at any other point in the reservoir can now be related to the water pressure at the water-oil contact as shown below:

$$p_{water} = p^{water}_{woc} + \gamma^{water} \left(z - z_{woc} \right) \tag{10.5}$$

Where ɣwater is the water density and z is the given depth.

Similarly, for gas we have the oil phase pressure at the Gas-Oil Contact ("GOC").

$$p^{oil}_{goc} = p^{oil}_{init} + \gamma^{oil} \left(z_{goc} - z_{init} \right) \tag{10.6}$$

Again, we can use gas-oil capillary pressure to obtain the gas phase pressure at the GOC, that is pgasgoc.

$$p^{gas}_{goc} = p^{oil}_{goc} + p_{cgoc} \tag{10.7}$$

and the gas phase pressure at any point in the reservoir is therefore determined from:

$$p_{gas} = p^{gas}_{goc} + \gamma^{gas} \left(z - z_{goc} \right) \tag{10.8}$$

Where ɣwas is the water density.

Having calculated the phase pressures at all depths in the reservoir (all grid block cells in the model), we can then calculate the capillary pressures, as the difference between the oil and water pressures, and the oil and gas pressures, depending on the phases present, that is:

$$p_{cwo} = p_{oil} - p_{water} \tag{10.9}$$

$$p_{cgo} = p_{gas} - p_{oil} \tag{10.10}$$

Under the assumption that the reservoir is initially at capillary equilibrium, we now calculate the average water saturation for each grid block in the computational grid on the basis of the capillary pressure at the center of that grid block.

$$S_w = S_w \left(P_{cow_{mid-point\ of\ grid\ cell}} \right) \tag{10.11}$$

The calculation of initial gas saturations is exactly analogous to the above treatment of water saturations.

$$S_g = S_g \left(P_{cog_{mid-point\ of\ grid\ cell}} \right) \tag{10.12}$$

Note also that summation of all fluids in a grid block should sum to one, that is, $S_g + S_o + S_w = 1.0$.

With grid blocks of sufficiently small size, the procedure described above will yield accurate saturation distributions for the static equilibrium situation. Unfortunately, in practice we are often forced to used grid blocks that are much larger than we would like because of computer time and storage considerations. One of the results of using such large blocks is that the saturation distributions and hence the oil- and gas-in-place volumes can sometimes be substantially in error.

This is easily demonstrated by imagining a case where the water-oil contact lies infinitesimally above the center of a 20-foot thick grid block. In this case, the procedure described above would cause the water saturation to be calculated as 1.0, whereas in actual fact it should be very nearly 0.5 (1.0 - Swc). Hence, we would make an error approaching half the pore volume of the grid block in the oil and water in-place in that block. To overcome this we can sub-divide each grid block in the vertical plane, and calculate the saturations for the sub-blocks, with the number of sub-blocks define via the EQLOPT3 option on the EQLNUM keyword. The simulator can than sum the sub-blocks to obtain the average simulator grid block saturation and thus obtain a more accurate estimate of the volumes in-place. Note this is less of a concern these days due to fact that most models now use 0.5 to 1.0 metres (one to three ft.) in the vertical resolution due to the improvement in computer efficiency.

There are variations to the above general methodology depending on the data entered, for example:

1) If the SWATINIT array, that stipulates the initial water saturation for each cell, has been entered in the PROPS section then the entered capillary pressure data on the saturation tables are re-scaled to honor the SWATINIT array whenever possible, and the re-scaled capillary pressure data are used in the equilibrium initialization calculations. Thus, each cell will have its own scaling of the capillary pressure that needs to be taken into account during the simulation.

2) For reservoirs with both oil and dissolved gas phases (OIL and DISGAS keywords in the RUNSPEC section), the dissolved gas ratio, Rs, or the saturation pressure Pb, may also be entered as a function of depth. See the RSVD and PBVD keywords in the SOLUTION section. If neither of these keywords have been entered then Rs is set to equal to the Rs value at the GOC. This also means that the datum depth on the EQUIL keyword in the SOLUTION section, must equal the GOC depth, in order for the saturated Rs to be calculated from the datum pressure and the associated PVT table.

3) Similarly, for reservoirs with both wet gas and vaporized oil (GAS and VAPOIL keywords in the RUNSPEC section), the dissolved vaporized oil (more commonly referred to as condensate) gas ratio, Rv, or the saturation pressure Pd, may also be entered as a function of depth. See the RVVD and PDVD keywords in the SOLUTION section. If neither of these keywords have been entered then Rv is set to equal to the Rv value at the GOC. Again, this also means that the datum depth must set to the GOC depth in order for the saturated Rv to be calculated from the datum pressure and the associated PVT table.

In OPM Flow[286] initial conditions are calculated in a similar manner by using the pressure at a reference depth, the depths of the water-oil and oil-gas contacts, and the capillary pressures at those contact depths. It then solves the following ordinary differential equations:

$$\frac{dp_{oil}}{dz} = \gamma_{oil}(z P_{oil})g$$

$$\frac{dp_{gas}}{dz} = \gamma_{gas}(z P_{gas})g \qquad (10.13)$$

$$\frac{dp_{wat}}{dz} = \gamma_{wat}(z P_{wat})g$$

Where

γ_i	=	oil, gas, water density,
z	=	the depth and,
Pi	=	the reference pressure depth, and g the gravity term.

The equations are solved numerically using a 4th-order Runge-Kutta[287] and [288] method, with the order of the phases decided by the location of the datum depth. First, the simulator solves for the phase in whose zone the datum is located, using the datum pressure to bind the pressure solution. Then the simulator solves for phases corresponding to the zones above and below. To tie the phase pressures, the simulator uses the already computed phase pressure(s) evaluated at the zone contact depth and the input capillary pressures.

The same approach is used for the water saturation but the capillary pressure is scaled to match the initial water saturation. Each cell will thus have its own scaling of the capillary pressure that needs to be taken into account during the simulations. Initial fluid properties (GOR, formation volume factors, etc.) are calculated from the cell equilibrium pressure and the entered fluid property tables, as described earlier.

[286] *Atgeirr Flø Rasmussen, Tor Harald Sandve, Kai Bao, Andreas Lauser, Joakim Hove, Bård Skaflestad, Robert Klöfkorn, Markus Blatt, Alf Birger Rustadf, Ove Sævareid, Knut-Andreas Lie, Andreas Thune: The Open Porous Media Flow Reservoir Simulator (21019). Computers & Mathematics with Applications October 2019.*

[287] *Runge, Carl David Tolmé (1895), "Über die numerische Auflösung von Differentialgleichungen", Mathematische Annalen, Springer, 46 (2): 167–178, doi:10.1007/BF01446807.*

[288] *Kutta, Martin (1901), "Beitrag zur näherungsweisen Integration totaler Differentialgleichungen", Zeitschrift für Mathematik und Physik, 46: 435–453.*

10.2 DATA REQUIREMENTS

10.2.1 EQUILIBRIUM INITIALIZATION

For the standard Equilibrium Initialization the EQUIL keyword in the SOLUTION section is used to initialize the model. The keyword contains the datum depth and pressure, oil-water and gas-oil contacts, the capillary pressure values at the fluid contacts, as well as various equilibrium initialization options, including how the dissolve gas and vaporized oil should be calculated, and if a refined in-place volume calculation should be performed. There should be one record for each equilibration region with the number of regions defined by the NTEQUL parameter on the EQLDIMS keyword in the RUNSPEC section.

Note that if a fluid contact is not present in the reservoir, for example if for two phase runs containing only gas and water, the water oil-water contact on the EQUIL keyword should be set to the gas-water contact. Also, in cases where there is no gas cap (or free gas) then gas-oil should be set to a value shallower than the top of the reservoir, and for three phases cases where there is initially no oil zone, as for a gas condensate field for example, the gas-oil contact should be set to the same depth as water-oil contact. Table 10.1 outlines the minimum set of keywords required by OPM Flow in order for the simulator to initialize the model For the standard Equilibrium Initialization option.

No.	SOLUTION Keyword	Equilibrium Initialization Description	Type Of Initialization
1	EQUIL	*EQUIL – Define the Equilibration Initialization Data* - this is main keyword for the Equilibration initialization that sets the datum depth, the pressure and fluid contacts for an equilibrium region.	Equilibration
2	PBVD	*PBVD – Equilibration Bubble-Point versus Depth Tables* - used for Live Black-Oil equilibration regions to define the saturation pressure versus depth relationship.	Equilibration
3	PDVD	*PDVD – Define Equilibration Dew-Point versus Depth Tables* - used for Wet Gas equilibration regions to define the saturation pressure versus depth relationship.	Equilibration
4	RSVD	*RSVD – Equilibration Dissolved Gas-Oil Ratio (Rs) versus Depth Tables* - used for Live Black-Oil equilibration regions to define the gas-oil ratio versus depth relationship.	Equilibration
5	RVVD	*RVVD – Equilibration Vaporized Oil-Gas Ratio (Rv) versus Depth Tables* - use for Wet Gas equilibration regions to declare the condensate-gas ratio versus depth relationship.	Equilibration
6	SALTVD	*SALTVD – Equilibration Salt Concentration versus Depth Tables* - this is a standard commercial simulator keyword used with the Brine model, but may also be used with OPM Flow's Vaporized Water and Salt Precipitation models.	Equilibration
7	THPRES	*THPRES - Define Equilibration Region Threshold Pressures*	Equilibration Enumeration
		Vaporized Water Model (OPM Flow Specific Keywords)	
1	RVWVD	*RVWVD – Equilibration Vaporized Water-Gas Ratio (Rvw) versus Depth Tables*	Equilibration

No.	SOLUTION Keyword	Equilibrium Initialization Description	Type Of Initialization
2	SALTPVD	*SALTPVD – Initial Precipitated Salt Volume Fraction versus Depth Tables*	Equilibration
3	SALTVD	*SALTVD – Equilibration Salt Concentration versus Depth Tables* - this is a standard commercial simulator keyword used with the Brine model, but may also be used with OPM Flow's Vaporized Water and Salt Precipitation models.	Equilibration

Notes:

1) Only the Equilibration and Restart initialization type of initialization are used in practice.
2) For the Equilibration option the PBVD, PDVD, RSVD and RSVVD keywords may be optional if the datum depth on he EQUIL keyword is set equal to the gas-oil contact ("GOC"), otherwise one of these keywords must be used to set the initial RS ("GOR") or RV ("GCR") values for the equilibrium region being defined.
3) For Live Black-Oil equilibrium regions if the datum depth is not equal to the GOC then one can either use the PBVD keyword to set the saturation pressure (bubble-point) versus depth, or the RSVD keyword to set the GOR versus depth. Similarly for Wet Gas equilibrium regions, one can either use the PDVD keyword to set the saturation pressure (dew-point) versus depth, or RVVD keyword to set the CGR versus depth.
4) The THPRES keyword is entirely optional but is commonly used to isolate various equilibrium regions.

Table 10.1: Minimum Set of SOLUTION Equilibrium Keywords Required by OPM Flow

The minimum set of SOLUTION keywords is for general three phases runs, that is where the OIL, DISGAS, GAS, VAPOIL and/or WATER keywords in the RUNSPEC section have been invoked. If additional phases or options have been invoked in the RUNSPEC section, then additional SOLUTION keywords may be required in order to initialize the model. For example, if the BRINE keyword in the RUNSPEC section has been used to activate the BRINE model then the SALTVD keyword in the SOLUTION section may or may not be used to set the initial salt concentration versus depth relationship.

10.2.2 ENUMERATION INITIALIZATION

Enumeration Initialization is based on user supplying all the required data for the simulator to determine the pressures, fluid saturations, and fluid properties for each cell in the model. This similar to entering the grid property array data in the GRID section, in that all the data is required for all the cells in the model.

To define the cell pressures for all the grid blocks the PRESSURE keyword in the SOLUTION section should be employed. There is also the PRVD keyword in the SOLUTION section that can be used to define a pressure versus depth relationship to enable the simulator to calculate the cell pressures instead of entering the data via the PRESSURE keyword; however, this keyword is not implemented in OPM Flow.

For the fluid saturations, it is only necessary to enter the saturations sufficient to define the system, so for a two phase system it is only necessary to define one phase, normally the water phase, as the other phase will be calculated by the simulator. Similarly, for three phases only two phases must be entered with the water phase always normally entered, and the other phase dependent on the type of reservoir. The data entered is using the SGAS, SOIL and SWAT keywords in the SOLUTION section.

Finally the initial fluid property data needs to be defined based on the fluid phases active in the model. For oil reservoirs either the RS keyword that defines a cell's initial gas-oil ratio, or the PBUB keyword that defines the oil's initial bubble point pressure for each cell are required. For gas reservoirs, either the RV keyword that defines vaporize oil-gas ratio (or condensate gas-oil ratio), or the PDEW keyword that defines the initial dew point pressure for each cell must be entered. Table 10.2 outlines the minimum set of keywords required by OPM Flow in order for the simulator to initialize the model For the Enumeration Initialization option -all the keywords are in the SOLUTION section.

No.	SOLUTION Keyword	Enumeration Initialization Description	Type Of Initialization
1	PRESSURE	*PRESSURE – Define the Initial Equilibration Pressures for All Grid Blocks*. This keyword must be present for all Enumeration equilibration regions.	Enumeration
2	PBUB	*PBUB – Define the Initial Equilibration Bubble-Point Pressure for All Grid Blocks* - used for Live Black-Oil equilibration regions to define the saturation pressure for each grid cell.	Enumeration
3	PDEW	*PDEW – Define the Initial Equilibration Dew-Point Pressure for All Grid Blocks* - used for Wet Gas equilibration regions to set the saturation pressure for each cell.	Enumeration
4	RS	*RS – Define the Initial Equilibration GOR (Rs) for All Grid Blocks* - used for Live Black-Oil equilibration regions to define the gas-oil ratio for each grid blocks.	Enumeration
5	RV	*RV – Define the Initial Equilibration CGR (Rv) for All Grid Blocks* - use for Wet Gas equilibration regions to set the condensate-gas ratio for each cell.	Enumeration
6	SALT	*SALT – Define the Initial Equilibration Salt Concentration for All Grid Blocks* - used to set the initial salt concentration in the Brine model and OPM Flow's Salt Precipitation model.	Enumeration
7	SGAS	*SGAS – Define the Initial Equilibration Gas Saturation for All Grid Blocks* - if gas is present in the model then the keyword is used to set the initial gas saturation.	Enumeration
8	SOIL	*SOIL – Define the Initial Equilibration Oil Saturation for All Grid Blocks* - if oil is present in the model then the keyword is used to set the initial gas saturation.	Enumeration
9	SWAT	*SWAT – Define the Initial Equilibration Water Saturation for All Grid Blocks* - since water is normally always present in petroleum reservoirs, this keyword is always used to define the initial water saturation. For two-phase runs, oil-water or gas-water, then it is only necessary to use this keyword, as the second phase saturations are automatically calculated by the simulator.	Enumeration
10	THPRES	*THPRES - Define Equilibration Region Threshold Pressures*	Equilibration Enumeration
		Microbial Induced Calcite Precipitation Model Enumeration (OPM Flow Specific Keywords)	
1	PRESSURE	*PRESSURE – Define the Initial Equilibration Pressures for All Grid Blocks*. This keyword must be present for all Enumeration equilibration regions.	Enumeration
2	SBIOF	*SBIOF - Define The Initial Equilibration Biofilm Volume Fraction For All Grid Blocks*	Enumeration
3	SCALC	*SCALC – Define The Initial Equilibration Calcite Volume Fraction For All Grid Blocks*	Enumeration

No.	SOLUTION Keyword	Enumeration Initialization Description	Type Of Initialization
4	SMICR	*SMICR – Define The Initial Equilibration Microbial Concentration For All Grid Blocks*	Enumeration
5	SOXYG	*SOXYG - Define The Initial Equilibration Oxygen Concentration For All Grid Blocks*	Enumeration
6	SUREA	*SUREA - Define The Initial Equilibration Urea Concentration For All Grid Blocks*	Enumeration
		Vaporized Water Model (OPM Flow Specific Keywords)	
1	RVW	*RVW – Define the Initial Equilibration Vaporized Water in Gas Ratio for All Grid Blocks* - used in both the Vaporized Water model and the Salt Precipitation model.	Enumeration
2	SALT	*SALT – Define the Initial Equilibration Salt Concentration for All Grid Blocks* - this is a standard commercial simulator keyword used with the Brine model, but may also be used with OPM Flow's Vaporized Water and Salt Precipitation models.	Enumeration
3	SALTP	*SALTP – Define the Initial Precipitated Salt Volume Fraction for All Grid Blocks*	Enumeration

1) In an Enumeration type of initialization only sufficient data to define the pressures and fluid distributions are required, for example, in a two phase gas-water run it is only necessary to use the PRESSURE, RV and either SGAS or SWAT keywords to formulate the solution
2) The THPRES keyword is entirely optional but is commonly used to isolate various equilibrium regions.
3) Microbial Induced Calcite Precipitation ("MICP") model is OPM Flow's implementation of MICP used to investigate CO_2 leakage remediation. Note currently the model assumes 100% water saturation and therefore it is not necessary to use the SWAT keyword.

Table 10.2: Minimum Set of SOLUTION Enumeration Keywords Required by OPM Flow

Again, the minimum set of SOLUTION keywords is for general three phases runs, that is where the OIL, DISGAS, GAS, VAPOIL and/or WATER keywords in the RUNSPEC section have been invoked. If additional phases or options have been invoked, then additional SOLUTION keywords may be required in order to initialize the model.

10.2.3 RESTART INITIALIZATION

Restart Initialization is performed by the simulator by loading a previously generated RESTART file and equilibrating the model accordingly. In nearly all cases the model will not satisfy the condition of gravity equilibrium; in other words the model is initialized at a point in time when after the model has produced or injected fluids.

To write a restart record to the RESTART file, either the RPTSOL or RPTRST keyword in the SOLUTION section can be used to write the initial restart record at time equal to zero. To write restart records at various time points as the simulation progeess through time, then either the RPTRST or RPTSCHED keywords may be used. Both the RPTSOL and RPTSCHED keywords enable just the basic restart record to be written to the RESTART file, which is generally sufficient in most cases. In comparison with the dedicated RPTRST keyword, RPTRST provides additional functionality on the data to be written to the RESTART file and is therefor the preferred keyword to use. For example to request the standard restart data be written out every month using the RPTRST keyword, one would use the following:

```
--
--       RESTART CONTROL BASIC = 4 (YEARLY) 5 (MONTHLY)
--
RPTRST
        BASIC=5                                                          /
```

Subsequently one of the monthly restart points can be used to restart the run.

The restart keywords in the in the various section used to created restart points and to initialize a model initialization are shown in Table 10.3.

No.	Keyword	Restart Creation Description	Keyword Section
1	RPTRST	*RPTRST – Define Data to be Written to the RESTART File*	SOLUTION
2	RPTSOL	*RPTSOL – Define SOLUTION Section Reporting*	
3	RPTRST	*RPTRST – Define Data to be Written to the RESTART File*	SCHEDULE
4	RPTSCHED	*RPTSCHED – Define SCHEDULE Section Reporting*	
No.	**Keyword**	**Restart Initialization Description**	**Keyword Section**
5	RESTART	*RESTART – Restart Run From an Existing Restart File* - this keyword is used to initialization the model from a previously run OPM Flow run or from the commercial simulator. Note that due to the complexities of the RESTART file, OPM Flow may not always be able to restart from the commercial simulators RESTART file.	SOLUTION

Notes:

1) OPM Flow can only restart runs from a RESTART file, the commercial simulator's SAVE file format is not supported.
2) The THPRES keyword is not required as the data is included in the RESTART file.
3) SCHEDULE section keywords not stored in the RESTART and must therefore remain in the input deck if required include:
   - GLOBAL section keywords: COLUMNS, DEBUG, ECHO, EXTRAPMS, FORMFEED, INCLUDE, MESSAGES, NOECHO, NOWARN and the OPTIONS keyword.
   - SCHEDULE section keywords: PIMULTAB, RPTSCHED, RPTRST, SCDPTAB, SCDATAB, TUNING, VFPPROD, and VFPINJ keywords.

Table 10.3: SOLUTION Restart Initialization Keywords Required by OPM Flow

In the input deck that is being employed to restart the run, all the equilibration keywords (EQUIL, RSVD, etc.) or the enumeration equilibration keywords (PRESSURE, SGAS, SOIL. SWAT, etc.) in the SOLUTION section used to initialize the model., should be deleted. Secondly, it is good practice to insert the SKIPREST keyword at the very beginning of the SCHEDULE section. The SKIPREST keyword causes the simulator to only read in data it requires for restarting the run up to the RESTART point. This is because certain keywords always need to be present in a restart run in the SCHEDULE section, as the data is not stored on the RESTART file, for example the VFP tables (VFPPROD and VFPINJ keywords in the SCHEDULE section). The SKIPREST keyword automatically processes the input deck and reads the required data.

10.2.4 SECTION KEYWORDS

A complete list of SOLUTION keywords in alphabetic order is shown in Table 10.4 together with a generalized Topic column that classifies the functionality of the keyword. Note that not all keywords and features listed in Table 10.4 are implemented in OPM Flow. Cells not colored in the No. column indicate the keyword is supported by OPM Flow, cells colored gray indicate that the keyword is not applicable, and finally, cells colored in orange indicate keywords that are not currently supported by OPM Flow.

No.	RUNSPEC Keyword	Description	Topic
1	ADD	*ADD – Add a Constant to a Specified Array*.	Input
2	ADDREG	*ADDREG – Add a Constant to an Array based on a Region Number*.	Input
3	APIVD	*APIVD - Equilibration Oil API Gravity versus Depth Tables*.	API
4	AQANCONL	*Error: Reference source not found*.	Aquifer
5	AQANNC	*AQANNC – Define Analytic Aquifer Non-Neighbor Connections*.	Aquifer
6	AQANTRC	*AQANTRC - Define Analytic Aquifer Initial Tracer Concentrations*.	Aquifer
7	AQUALIST	*AQUALIST – Define An Analytic Aquifer Name to Aquifer Numbers* .	Aquifer
8	AQUANCON	*AQUANCON – Define Analytical Connections to the Grid*.	Aquifer
9	AQUCHGAS	*AQUCHGAS – Define Constant Pressure Gas Analytical Aquifer Properties*.	Aquifer
10	AQUCHWAT	*AQUCHWAT – Define Constant Pressure Water Analytical Aquifer Properties*.	Aquifer
11	AQUCON	*AQUCON – Define Numerical Aquifer Connections to the Grid*.	Aquifer
12	AQUCT	*AQUCT – Define Carter-Tracy Analytical Aquifers*.	Aquifer
13	AQUFET	*AQUFET – Define Fetkovich Analytical Aquifer and Connections* .	Aquifer
14	AQUFETP	*AQUFETP – Define Fetkovich Analytical Aquifers* .	Aquifer
15	AQUFLUX	*AQUFLUX - Define Constant Flux Analytical Aquifer*.	Aquifer
16	BC	*BC – Define Boundary Conditions* This is an OPM Flow specific keyword defines the boundary conditions for the model, and can be used to set boundary conditions for when external influx or efflux volumes are influencing the reservoir pressure and production hi	Equilibration
17	BOUNDARY	*BOUNDARY – Define a Boundary Box for Printing*.	Output
18	BOX	*BOX - Define a Range of Grid Blocks to Enter Property Data*.	Input
19	COLUMNS	*COLUMNS – Define Input File Column Margins*.	Input
20	COPY	*COPY – Copy Array Data to Another Array*.	Input

No.	RUNSPEC Keyword	Description	Topic
21	COPYREG	*COPYREG – Copy an Array to Another Array based on a Region Number*.	Input
22	DATUM	*DATUM – Define the Datum Depth for the Model*.	Output
23	DATUMR	*DATUMR – Define Datum Depths for the FIPNUM Regions*.	Output
24	DATUMRX	*DATUMRX – Define Datum Depths for the FIP Allocated Regions* .	Output
25	DEBUG	*DEBUG – Define the Debug Data to be Printed to File*.	Output
26	DYNAMICR	*DYNAMICR – Start of Dynamic Region Parameter Definition*.	Input
27	ECHO	*ECHO – Activate Echoing of User Input Files to the Print File*.	Output
28	END	*END – Define the End of the Input File*.	Input
29	ENDBOX	*ENDBOX – Define the End of the BOX Defined Grid*.	Input
30	ENDDYN	*ENDDYN– End of Dynamic Region Parameter Definition*.	Input
31	ENDFIN	*ENDFIN – End the Definition of a Local Grid Refinement*.	LGR
32	ENDINC	*ENDINC – Define the End of an Include File*.	Input
33	ENDSKIP	*ENDSKIP – DeActivate Skipping of Keywords and Input Data*.	Input
34	EQUALREG	*EQUALREG – Sets an Array to a Constant by Region Number*.	Input
35	EQUALS	*EQUALS – Sets a Specified Array to a Constant*.	Input
36	EQUIL	*EQUIL – Define the Equilibration Initialization Data*.	Equilibration
37	EXTRAPMS	*EXTRAPMS – Activate Extrapolation Warning Messages*.	Output
38	FILEUNIT	*FILEUNIT – Activate Unit Consistency Checking*.	Input
39	FORMFEED	*FORMFEED – Defined the Print File Form-Feed Character*.	Output
40	GASCONC	*GASCONC – Define the Initial Equilibration Coal Gas Concentration for All Grid Blocks*.	CBM
41	GASSATC	*GASSATC – Define the Initial Equilibration Saturated Coal Gas Concentration for All Grid Blocks*.	CBM
42	GCVD	*GCVD – Define Equilibration Coal Gas Concentration versus Depth Tables*.	CBM
43	GETDATA	*GETDATA – Load and Assign Data Array from INIT or RESTART Files*.	Input
44	GETGLOB	*GETGLOB – Activate Loading of Global Grid Restart Data Option*.	Input
45	GI	*GI - Define the Initial Equilibration Gi Values for All Grid Blocks*.	Enumeration

No.	RUNSPEC Keyword	Description	Topic
46	HMAQUCT	*HMAQUCT – History Match Carter-Tracy Aquifer Gradient Parameters*.	Gradient
47	HMAQUFET	*HMAQUFET – History Match Fetkovich Aquifer Gradient Parameters*.	Gradient
48	HMMLCTAQ	*HMMLCTAQ – History Match Carter-Tracy Aquifer Gradient Multipliers*.	Gradient
49	HMMLFTAQ	*HMMLFTAQ – History Match Fetkovich Aquifer Gradient Multipliers*.	Gradient
50	HMMLTWCN	*HMMLTWCN – History Match Well Connection and Skin Multipliers*.	Gradient
51	HMWELCON	*HMWELCON – History Match Well Connection and Skin Parameters*.	Gradient
52	IMPORT	*IMPORT – Import Grid File Data at the Current Position*.	Input
53	INCLUDE	*INCLUDE – Load Another Data File at the Current Position*.	Input
54	MESSAGE	*MESSAGE – Output User Message*. The MESSAGE keyword outputs a user message to the terminal, as well as to the print (*.PRT) and debug (*.DBG) files.	Output
55	MESSAGES	*MESSAGES – Define Message Print Limits and Stop Limits*. The MESSAGES keyword defines the print and stops levels for various messages.	Output
56	MULTIPLY	*MULTIPLY – Multiply a Specified Array by a Constant*.	Input
57	MULTIREG	*MULTIREG – Multiply an Array by a Constant based on a Region Number*.	Input
58	NOECHO	*NOECHO – Deactivate Echoing of User Input Files to the Print File*.	Output
59	NOHMD	*NOHMD – Deactivate History Match Gradient Derivative Calculations*.	Gradient
60	NOHMO	*NOHMO – Deactivate History Match Gradient Derivative Calculations (Alias)*.	Gradient
61	NOWARN	*NOWARN – Deactivate Warning Messages*.	Output
62	OILAPI	*OILAPI – Define the Initial Equilibration Oil API for All Grid Blocks*.	API
63	OPERATE	*OPERATE – Define Mathematical Operations on Arrays*.	Input
64	OPERATER	*OPERATER – Define Mathematical Operations on Arrays by Region*.	Input
65	OUTSOL	*OUTSOL – Define Data to be Written to the RESTART File (Retired)*.	Output
66	PBUB	*PBUB – Define the Initial Equilibration Bubble-Point Pressure for All Grid Blocks*.	Enumeration
67	PBVD	*PBVD – Equilibration Bubble-Point versus Depth Tables*.	Equilibration
68	PDEW	*PDEW – Define the Initial Equilibration Dew-Point Pressure for All Grid Blocks*.	Enumeration

No.	RUNSPEC Keyword	Description	Topic
69	PDVD	*PDVD – Define Equilibration Dew-Point versus Depth Tables*.	Equilibration
70	PRESSURE	*PRESSURE – Define the Initial Equilibration Pressures for All Grid Blocks*.	Enumeration
71	PRVD	*PRVD – Define the Initial Equilibration Pressures versus Depth*.	Enumeration
72	PYEND	*PYEND – End the Definition of a PYINPUT Section*. This keyword is an OPM Flow specific keyword for the simulator's scripting facility using the standard Python interpreter.	Action
73	PYINPUT	*PYINPUT – Define the Start of a PYINPUT Section*. This keyword is an OPM Flow specific keyword for the simulator's scripting facility using the standard Python interpreter.	Action
74	RAINFALL	*RAINFALL – Constant Flux Aquifer Rainfall Flux by Month*.	Aquifer
75	RBEDCONT	*RBEDCONT – Define River Grid Block Contact Area versus Depth*.	Rivers
76	REFINE	*REFINE – Start the Definition of a Local Grid Refinement*.	LGR
77	RESTART	*RESTART – Restart Run From an Existing Restart File*.	Restart
78	RIVERSYS	*RIVERSYS - Define River System (Branch Structure and Boundary Conditions)*.	Rivers
79	RPTRST	*RPTRST – Define Data to be Written to the RESTART File*.	Output
80	RPTSOL	*RPTSOL – Define SOLUTION Section Reporting*.	Output
81	RS	*RS – Define the Initial Equilibration GOR (Rs) for All Grid Blocks*.	Enumeration
82	RSVD	*RSVD – Equilibration Dissolved Gas-Oil Ratio (Rs) versus Depth Tables*.	Equilibration
83	RTEMP	*RTEMP - Define the Initial Reservoir Temperature for the Model*.	Thermal
84	RTEMPA	*RTEMPA - Define the Initial Reservoir Temperature for the Model*.	Thermal
85	RTEMPVD	*RTEMPVD - Define the Initial Reservoir Temperature versus Depth Tables*.	Thermal
86	RV	*RV – Define the Initial Equilibration CGR (Rv) for All Grid Blocks*.	Enumeration
87	RVVD	*RVVD – Equilibration Vaporized Oil-Gas Ratio (Rv) versus Depth Tables*.	Equilibration
88	RVW	*RVW – Define the Initial Equilibration Vaporized Water in Gas Ratio for All Grid Blocks* This is an OPM Flow specific keyword for the simulator's Water Vaporization Model that is activated by the VAPWAT in the RUNSPEC section.	Water Vaporization

No.	RUNSPEC Keyword	Description	Topic
89	RVWVD	*RVWVD – Equilibration Vaporized Water-Gas Ratio (Rvw) versus Depth Tables* This is an OPM Flow specific keyword for the simulator's Water Vaporization Model that is activated by the VAPWAT in the RUNSPEC section.	Water Vaporization
90	SALT	*SALT – Define the Initial Equilibration Salt Concentration for All Grid Blocks*. A standard commercial simulator keyword used with the Brine Model, but may also be used with OPM Flow's Vaporized Water and Salt Precipitation models.	Brine
91	SALTP	*SALTP – Define the Initial Precipitated Salt Volume Fraction for All Grid Blocks* This is an OPM Flow specific keyword for the simulator's Salt Precipitation Model that is activated by the PRECSALT keyword and declaring that vaporized water is present in the run via the VAPWAT in the RUNSPEC section.	Water Vaporization
92	SALTPVD	*SALTPVD – Initial Precipitated Salt Volume Fraction versus Depth Tables*. This is an OPM Flow specific keyword for the simulator's Salt Precipitation Model that is activated by the PRECSALT keyword and declaring that vaporized water is present in the run via the VAPWAT in the RUNSPEC section.	Water Vaporization
93	SALTREST	*SALTREST – Define the Restart Salt Concentration for All Grid Blocks*.	Brine
94	SALTVD	*SALTVD – Equilibration Salt Concentration versus Depth Tables*. A standard commercial simulator keyword used with the Brine model, but may also be used with OPM Flow's Vaporized Water and Salt Precipitation models.	Brine
95	SBIOF	*SBIOF - Define The Initial Equilibration Biofilm Volume Fraction For All Grid Blocks*	MICP
96	SCALC	*SCALC – Define The Initial Equilibration Calcite Volume Fraction For All Grid Blocks*	MICP
97	SCVD	*SCVD – Define Equilibration Coal Solvent Concentration versus Depth Tables*.	CBM
98	SFOAM	*SFOAM – Define the Initial Equilibration Foam Concentration for All Grid Blocks*.	Enumeration
99	SGAS	*SGAS – Define the Initial Equilibration Gas Saturation for All Grid Blocks*.	Enumeration
100	SKIP	*SKIP – Activate Skipping of All Keywords and Input Data*.	Input
101	SKIP100	*SKIP100 – Activate Skipping of Black-Oil Keywords and Input Data*.	Input
102	SKIP300	*Error: Reference source not found*.	Input

No.	RUNSPEC Keyword	Description	Topic
103	SMICR	*SMICR – Define The Initial Equilibration Microbial Concentration For All Grid Blocks*	MICP
104	SOIL	*SOIL – Define the Initial Equilibration Oil Saturation for All Grid Blocks*.	Enumeration
105	SOLUTION	*SOLUTION - Define the Start of the SOLUTION Section of Keywords*.	Input
106	SOLVCONC	*SOLVCONC – Define the Initial Coal Solvent Concentration for All Grid Blocks*.	CBM
107	SOLVFRAC	*SOLVFRAC – Define the Initial Gas Solvent Fraction for All Grid Blocks*	Solvent
108	SOXYG	*SOXYG - Define The Initial Equilibration Oxygen Concentration For All Grid Blocks*	MICP
109	SPOLY	*SPOLY – Define the Initial Equilibration Polymer Concentration for All Grid Blocks*.	Polymer
110	SPOLYMW	*SPOLYMW – Define The Initial Equilibration Polymer Molecular Weights For All Grid Blocks* This is an OPM Flow specific keyword for the simulator's Polymer Molecular Weight Transport option that uses the polymer molecular weight in calculating the polymer viscosity, as well as accounting for formation damage due to the water and polymer injection, by adjusting the wellbore skin pressure.	Polymer
111	SSOL	*SSOL – Define the Initial Equilibration Solvent Saturation for All Grid Blocks*.	Solvent
112	SUREA	*SUREA - Define The Initial Equilibration Urea Concentration For All Grid Blocks*	MICP
113	SURF	*SURF – Define the Initial Equilibration Polymer Concentration for All Grid Blocks*.	Surfactant
114	SWAT	*SWAT – Define the Initial Equilibration Water Saturation for All Grid Blocks*.	Enumeration
115	TBLK	*TBLK – Define Tracer Initial Grid Block Concentrations*.	Tracers
116	TEMPI	*TEMPI – Define the Initial Temperature Values for All Cells*.	Thermal
117	TEMPVD	*TEMPVD - Define the Initial Reservoir Temperature versus Depth Tables*.	Thermal
118	THPRES	*THPRES - Define Equilibration Region Threshold Pressures*.	Equilibration
119	TVDP	*TVDP – Define the Initial Equilibration Tracer Saturation versus Depth Functions*.	Tracers
120	VAPPARS	*VAPPARS – Oil Vaporization Parameters*.	Phases

No.	RUNSPEC Keyword	Description	Topic
121	VISDATES	*VISDATES – Define External Reservoir Geo-Mechanics VISAGE Stress Dates*.	Geo-Mechanics
122	VISOPTS	*VISOPTS – Define External Reservoir Geo-Mechanics VISAGE Options*.	Geo-Mechanics
123	WARN	*WARN – Activate Warning Messages*.	Output

Notes:

1) Note that not all keywords and features listed above are implemented in OPM Flow. Cells not colored in the No. column indicate the keyword is supported, cells colored gray indicate that the keyword is not applicable, and finally, cells colored in orange indicate keywords that are not supported by OPM Flow.
2) For the Topic column the definition of the entries is defined as follows:

1) **Action:** Keywords associated with the ACTIONX or OPM Flow's Python facility.
2) **API:** API option equilibration keywords.
3) **Brine:** BRINE option equilibration keywords.
4) **CBM:** COAL BED METHANE equilibration keyword (not implemented in OPM Flow).
5) **Enumeration:** Enumeration equilibration keywords.
6) Equilibration: Equilibration keywords.
7) **Geo-Mechanics:** GEO_MECHANICS equilibration keywords (this option is not available in OPM Flow).
8) **Gradient:** GRADIENT equilibration keywords., used in history matching (this option is not available in OPM Flow).
9) Input: Input control and options.
10) **LGR:** LOCAL GRID REFINEMENT equilibration keywords (not implemented in OPM Flow).
11) **MICP: Microbial Induced Calcite Precipitation** model **equilibration keywords**.
12) **Output:** Output control and options.
13) **Phases:** Equilibration keywords related to the active phases.
14) **Polymer**: POLYMER equilibration keywords.
15) **Restart:** Restart equilibration keywords.
16) Rivers Keywords associated with the RIVERS facility (not implemented in OPM Flow).
17) **Solvent:** SOLVENT equilibration keywords.
18) **Thermal:** THERMAL equilibration keywords.
19) **Tracers:** TRACER equilibration keywords.
20) **Water Vaporization:** WATER VAPORIZATION equilibration keywords.

Table 10.4:Alphabetic List of SOLUTION Keywords

Note that a number of keywords can be classified under several topics, so the Topic column should only be used as a general reference guide.

Example

The first example is taken from the Norne field, in which the model is in SI units, and the NTEQUL varibale on the EQLDIMS keyword in the RUNSPEC section has been set to five for five equilibration regions in the model.

```
--
--        DATUM    DATUM    OWC      PCOW    GOC     PCGO    RS    RV    N
--        DEPTH    PRESS    DEPTH    ----    DEPTH   ----    OPT   OPT   OPT
EQUIL
          2582.0   268.56   2692.0    0.0   2582.0   0.0     1     0     0 / C+D:Garn
          2500.0   263.41   2585.5    0.0   2500.0   0.0     2     0     0 / G:  Garn
          2582.0   269.46   2618.0    0.0   2582.0   0.0     3     0     0 / E:  Garn
          2200.0   236.92   2400.0    0.0   2200.0   0.0     4     0     0 / G:Ile-Tilje
          2585.0   268.77   2693.3    0.0   2585.0   0.0     5     0     0 / C+D+E:Ile-Ti
--
--        DEPTH     RS
--                  sm3/sm3
--        ------    --------
RSVD
          2582.0     120.25
          2597.0     110.00
          2660.7     106.77
          2697.0     106.77                                  / RS VS DEPTH EQUIL REGN 01
--        ------    --------
          2500.0      94.50
          2510.0      94.30
          2590.0      94.10                                  / RS VS DEPTH EQUIL REGN 02
--        ------    --------
          2582.0     120.25
          2597.0     110.00
          2660.7     106.77
          2697.0     106.77                                  / RS VS DEPTH EQUIL REGN 03
--        ------    --------
          2500.0      94.50
          2510.0      94.30
          2590.0      94.10                                  / RS VS DEPTH EQUIL REGN 04
--        ------    --------
          2585.9     120.29
          2599.9     110.00
          2663.6     106.77
          2699.9     106.77                                  / RS VS DEPTH EQUIL REGN 05
--
--        EQLNUM   EQLNUM   THPRES
--        FROM     TO       VALUE
THPRES
          1        2        0.588031                        / REGN 1 TO REGN 2
          2        1        0.588031                        / REGN 2 TO REGN 1
          1        3        0.787619                        / REGN 1 TO REGN 3
          3        1        0.787619                        / REGN 3 TO REGN 1
          1        4        7.000830                        / REGN 1 TO REGN 4
          4        1        7.000830                        / REGN 4 TO REGN 1
/
```

Note that even though the datum depth are the gas-oil contact depths are the same the RSVD keyword has been used to defined the RS (GOR) versus depth relationship. Secondly, the THPRES keyword has been used to set a threshold pressure between the various equilibration regions to prevent flow between the regions until the threshold is exceeded.

The next example is taken from a field containing only dead[289] oil with one equilibration region and Fetkovich[290] analytical aquifer connected on the flanks of the field.

```
--
--       DATUM   DATUM   OWC     PCOW   GOC    PCGO   RS   RV   N
--       DEPTH   PRESS   DEPTH   ----   DEPTH  ----   OPT  OPT  OPT
EQUIL
         3707    1609    3707    0      1*      1*     0    0     0  /
--
--       DATUM
--       DEPTH
DATUM
         3500                                                    / DATUM DEPTH
--
--                       FETKOVICH AQUIFER DESCRIPTION
--
--      ID   DATUM   AQF    AQF    AQF     AQF    AQF   SALT
--      NUM  DEPTH   PRESS  VOLM   COMP    PI     PVT   CONC
--
AQUFETP
         1   1130.   1*    1.0E+12 3.0E-5  500E3  1     0.0        /
/
--
--                       ANALYTIC AQUIFER CONNECTION
--
--  ID     ---------- BOX ---------   CONNECT  AQF    AQF      ADJOIN
--  NUMBER I1  I2   J1  J2   K1  K2   FACE     INFLX  MULTI    CELLS
--
--                       ANALYTIC AQUIFER CONNECTION
--
--       ID     ---------- BOX ---------   CONNECT  AQF    AQF    ADJOIN
--       NUMBER I1  I2   J1  J2   K1  K2   FACE     INFLX  MULTI  CELLS
AQUANCON
         1     57  57    28  36   46  58    'I+'    1*     1*     'NO' /
         1      1   1    14  14    1  29    'J-'    1*     1*     'NO' / NORTHERN
         1      2   2    12  12    1  29    'J-'    1*     1*     'NO' /
         1      3   3    10  10    1  29    'J-'    1*     1*     'NO' /
         1      4   5     9   9    1  29    'J-'    1*     1*     'NO' /
         1      6   6     8   8    1  29    'J-'    1*     1*     'NO' /
         1     14  14     4   4    1  29    'J-'    1*     1*     'NO' /
         1     15  15     2   2    1  29    'J-'    1*     1*     'NO' /
         1     16  77     1   1    1  29    'J-'    1*     1*     'NO' /
         1      1   1    14 136    1  29    'I-'    1*     1*     'NO' / WESTERN
         1     13  13   137 141    1  29    'I-'    1*     1*     'NO' /
         1     12  12   142 143    1  29    'I-'    1*     1*     'NO' /
         1     11  11   144 144    1  29    'I-'    1*     1*     'NO' /
         1     10  10   145 145    1  29    'I-'    1*     1*     'NO' /
         1      9   9   146 146    1  29    'I-'    1*     1*     'NO' /
         1      8   8   147 147    1  29    'I-'    1*     1*     'NO' /
```

Notice how the gas-oil contact has been defaulted as there is no gas in the model only oil and water. Alternatively, one could located the gas-oil contact above the reservoir. Secondly the DATUM keyword has been used to reset the datum depth for pressure reporting.

[289] *"Dead" oil is oil that it contains no dissolved gas or a relatively thick oil or residue that has lost its volatile components.*

[290] *Fetkovich, M. J. "A Simplified Approach to Water Influx Calculations - Finite Aquifer Systems," Journal of Petroleum Technology, (1971) 23, No. 7, 814-828.*

10.3 KEYWORD DEFINITIONS

10.3.1 ADD – ADD A CONSTANT TO A SPECIFIED ARRAY

The ADD keyword adds a constant to a specified array or part of an array. The constant can be real or integer depending on the array type; however, the arrays that can be operated on is dependent on which section the ADD keyword is being applied.

See *ADD – Add a Constant to a Specified Array* in the GRID section for a full description.

10.3.2 ADDREG – ADD A CONSTANT TO AN ARRAY BASED ON A REGION NUMBER

The ADDREG keyword adds a constant to a specified array or part of an array based on cells with a specific region number. The region number array can be FLUXNUM, MULTNUM or OPERNUM and these arrays must be defined and be available before the ADDREG keyword is read by the simulator. The constant can be real or integer depending on the property array type; however, the property arrays that can be operated on is dependent on which section the ADDREG keyword is being used.

See *ADDREG – Add a Constant to an Array based on a Region Number* in the GRID section for a full description.

10.3.3 APIVD - EQUILIBRATION OIL API GRAVITY VERSUS DEPTH TABLES

Description

The APIVD keyword defines the oil API gravity versus depth tables for each equilibration region when API Tracking as been activated by the API keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	DEPTH	A columnar vector of real monotonically increasing down the column values that defines the depth values for the corresponding API gravity values, API.			
		feet	m	cm	None
2	API	A columnar vector of real values that defines the API gravity at the corresponding DEPTH. The American Petroleum Institute ("API") classifies oils based on an API gravity (γ_{API}), or degrees API (°API), the relationship between relative density (γ_o) of oil and API gravity (γ_{API}) is given by: $$\gamma_{API} = \frac{141.5}{\gamma_o} - 131.5$$			
		°API	°API	°API	None

Notes:

1) The keyword is followed by NTEQUL records as declared on the EQLDIMS keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NDRXVD rows as declared on the EQLDIMS keyword in the RUNSPEC section.
3) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 10.5:APIVD Keyword Description

Example

Given NTEQUL equals three and NDRXVD is greater than or equal to two on the EQLDIMS keyword in the RUNSPEC section, then the following example defines the bubble-point versus depth functions.

```
--
--        DEPTH     API
--                  GRAVITY
--        ------    --------
APIVD
          3000.0     41.10
          8000.0     41.10                              / API VS DEPTH EQUIL REGN 01
--        ------    --------
          3000.0     41.10
          8000.0     38.50                              / API VS DEPTH EQUIL REGN 02
--        ------    --------
          3000.0     41.10
          8000.0     38.50                              / API VS DEPTH EQUIL REGN 03
```

Here three tables are entered; the first table has a constant API gravity versus depth relationship for equilibration region number one and the other two equilibration regions have the API gravity varying with depth.

10.3.4 AQANCONL – DEFINE ANALYTICAL CONNECTIONS TO A LGR GRID

Description

The AQANCON keyword defines how analytical aquifers are connected to a Local Grid Refinement ("LGR") grid, this includes the Carter-Tracy, Fetkovich and Constant Flux analytical aquifers, all of which are implemented in OPM Flow. Carter-Tracy analytical aquifers are characterized by the AQUCT keyword in the GRID section and Fetkovich analytical aquifers are defined by either the AQUFET or AQUFETP keywords in the SOLUTION section. Finally, the Constant Flux aquifer is defined by the AQUFLUX keyword in SOLUTION section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	AQUID	AQUID is a positive integer greater than or equal to one and less than the maximum number of analytical aquifers as defined by the NANAQU variable on the AQUDIMS keyword in the RUNSPEC section, that defines the aquifer to be connected to the grid.			None
2	LGRNAME	A character string of up to eight characters in length that defines the name of the LGR that will connect to an analytical aquifer AQUID. The LGR must have been previously defined by the either the CARFIN (Cartesian LGR grid) keyword, or the RADIN/RADIN4 (radial LGR grid) keyword in the GRID section.			None
3	I1	A positive integer that defines the LGR's lower bound of the cells in the I-direction to be connected to the aquifer, and must be greater than or equal to one and less than or equal to I2 and NX on the CARFIN keyword for Cartesian grids.			1
4	I2	A positive integer that defines the LGR's upper bound of the of the cells in the I-direction to be connected to the aquifer, and must be greater than or equal to I1 and less than or equal to NX on the CARFIN keyword for Cartesian grids.			NX
5	J1	A positive integer that defines the LGR's lower bound of the cells in the J-direction to be connected to the aquifer, and must be greater than or equal to one and less than or equal to J2 and NY on the CARFIN keyword for Cartesian grids.			1
6	J2	A positive integer that defines the LGR's upper bound of the cells in the J-direction to be connected to the aquifer, and must be greater than or equal to J1 and less than or equal to NY on the CARFIN keyword for Cartesian grids.			NY
7	K1	A positive integer that defines the LGR's lower bound of the cells in the K-direction to be to be connected to the aquifer, and must be greater than or equal to one and less than or equal to K2 and NZ on the CARFIN keyword for Cartesian grids.			1

No.	Name	Description			Default
		Field	Metric	Laboratory	
8	K2	A positive integer that defines the LGR's upper bound of the cells in the K-direction to be connected to the aquifer, and must be greater than or equal to KI and less than or equal to NZ on the CARFIN keyword for Cartesian grids.			NZ
9	AQUFACE	AQUFACE is a character string that sets the connection "face" of the cells declared by this record and should be set to one of the following: 1) X+, Y+, or Z+ for the positive direction, or X-, Y- or Z- for the negative direction transmissibilities. 2) I+, J+, or K+ for the positive direction, or I-, J- or K- for the negative direction transmissibilities.			None
10	AQUFLUX	AQUFLUX is a positive real value that sets the fraction of the total influx between the aquifer and the defined cells declared on this keyword. If defaulted the cell face for each cell is applied and if a values is declared then this values is applied to all cells declared by this record.			
		ft^2	m^2	cm^2	1*
11	AQUCOEF	AQUCOEF is a real positive values that scales the calculated connection between the aquifer and the cells declared on this record.			
		dimensionless	dimensionless	dimensionless	1.0
12	AQUOPT	AQUOPT is a character string that sets the cell face connection and should be set to one of the following: 1) YES: Aquifer connections can adjoin to active cells allowing for connections inside the reservoir grid. It is not recommended to use this option without thoroughly checking the connections in the model. 2) NO: Aquifer connections cannot adjoin to active cells preventing connections inside the reservoir grid. This is the recommended and the default value.			NO

Notes:

1) Where NX, NY and NZ are the dimensions of the LGRNAME as defined on the CARFIN keyword in the GRID section, or the NR, NTHEATA and NZ dimensions on the RADFIN keyword in the GRID section.
2) Each record must be terminated by a "/" and the keyword is terminated by a "/".

Table 10.6: AQANCONL Keyword Description

Note

If the AQANCONL keyword has been utilized in the run deck then OPM Flow will write the AQUIFERA array to the *.INIT file in order to visualize the aquifer connections in OPM ResInsight.

This is accomplished by setting the AQUIFERA value to 2(AQUID-1) for cells connected to aquifer AQUID. If a cell is connected to multiple analytical aquifers then AQUIFERA is summed for all aquifers connected to a cell. Note that connecting cells to multiple aquifers is best avoided.

Example

The following example defines aquifer number one connected to the J- face of various cells in the LGROP001 LGR, and a second basal aquifer connected to the K+ face.

```
--
--                      LGR ANALYTIC AQUIFER CONNECTION
--
--       ID   LGR        ---------- BOX ---------    CONNECT  AQF    AQF    ADJOIN
--       NUM  NAME       I1  I2   J1  J2   K1  K2   FACE     INFLX  MULTI  CELLS
AQANCONL
         1    LGROP001    1  10   1   15   1   10    J-       1*     1.0     'NO' /
         1    LGROP001    1  10   1   15   12  18    J-       1*     1.0     'NO' /
         2    LGROP001    1  10   1   15   20  20    K+       1*     1.0     'NO' /
/
```

See the AQUCT keyword in the GRID section for a complete example on defining and connecting a Carter-Tracy aquifer to a simulation grid.

10.3.5 AQANNC – DEFINE ANALYTIC AQUIFER NON-NEIGHBOR CONNECTIONS

Description

AQANNC defines the analytic aquifer non-neighbor connections.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

10.3.6 AQANTRC - DEFINE ANALYTIC AQUIFER INITIAL TRACER CONCENTRATIONS

Description

The AQANTRC keyword defines the initial tracer concentrations for analytic aquifers that have previously been defined by the AQCT keyword in the GRID, PROPS, or SOLUTION sections for Carter-Tracy analytical aquifers, or the AQFET and AQFETP keywords in the SOLUTION section for Fetkovich analytical aquifers.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

10.3.7 AQUALIST – DEFINE AN ANALYTIC AQUIFER NAME TO AQUIFER NUMBERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, AQULIST, defines an analytic aquifer name to aquifer numbers for greater readability in the output.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

10.3.8 AQUANCON – DEFINE ANALYTICAL CONNECTIONS TO THE GRID

The AQUANCON keyword defines how analytical aquifers are connected to the simulation grid, this includes Carter-Tracy and Fetkovich analytical aquifers, both of which are implemented in OPM Flow.

See *AQUANCON – Define Analytical Connections to the Grid* in the GRID section for a full description.

10.3.9 AQUCHGAS – DEFINE CONSTANT PRESSURE GAS ANALYTICAL AQUIFER PROPERTIES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The AQUCHGAS keyword defines the properties of constant pressure gas analytical aquifers.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

10.3.10 AQUCHWAT – DEFINE CONSTANT PRESSURE WATER ANALYTICAL AQUIFER PROPERTIES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The AQUCHWAT keyword defines the properties of constant pressure water analytical aquifers.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

10.3.11 AQUCON – DEFINE NUMERICAL AQUIFER CONNECTIONS TO THE GRID

AQUCON keyword defines how numerical aquifers are connected to the simulation grid.

See *AQUCON – Define Numerical Aquifer Connections to the Grid* in the GRID section for a full description.

10.3.12 AQUCT – DEFINE CARTER-TRACY ANALYTICAL AQUIFERS

The AQUCT keyword defines a Carter Tracy aquifer, the properties of the aquifer, including the Carter-Tracy aquifer influence function associated with the aquifer, defined by the AQUTAB keyword in the PROPS section.

See *AQUCT – Define Carter-Tracy Analytical Aquifers* in the GRID section for a full description

10.3.13 AQUFET – DEFINE FETKOVICH ANALYTICAL AQUIFER AND CONNECTIONS

Description

The AQUFET keyword defines Fetkovich[291] analytical aquifers, the aquifer properties, together with the cell connections to the aquifer. Each row entry in the AQUFETP keyword defines one Fetkovich analytical aquifer and one cell face to be connected to the aquifer.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped; however, see the AQUFETP keyword in the SOLUTION section and AQUANCON keyword in the GRID section, on how to define and connect Fetkovich analytical aquifers.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	DATUM	DATUM is a single positive value that defines the Fetkovich reference datum depth for PRESS.			
		feet	m	cm	None
2	PRESS	PRESS is a single positive value that defines the aquifer pressure at DATUM. If PRESS is defaulted then the simulator will set the aquifer's initial reservoir pressure to be in equilibrium with the cells the aquifer is contacted to. Defaulting this parameter will avoid inconsistent equilibration pressures between the reservoir cells and the aquifer.			
		psia	barsa	atma	1*
3	PORV	A real positive value that defines the initial water volume of the aquifer.			
		stb	sm^3	scc	None
4	COMP	COMP is a real number defining the total compressibility (Ct) of the aquifer, that is the rock compressibility (Cf) plus the water compressibility (Cw) at the aquifer datum pressure (DATUM) and is defined as: $C_t = C_f + C_w$			
		1/psia	1/barsa	1/atma	None
5	PI	A real positive number that defines the aquifer productivity index based on the aquifer influx rate per unit pressure drop.			
		stb/d/psia	sm^3/barsa	scc/hr/atma	None
6	PVTW	A positive integer that defines the aquifer's PVTW water property table.			1
7	I1	A positive integer that defines the lower bound of the cells in the I-direction to be connected to the aquifer and must be greater than or equal to one and less than or equal to I2 and NX.			1

[291] *Fetkovich, M. J. "A Simplified Approach to Water Influx Calculations - Finite Aquifer Systems," Journal of Petroleum Technology, (1971) 23, No. 7, 814-828.*

No.	Name	Description			Default
		Field	Metric	Laboratory	
8	I2	A positive integer that defines the upper bound of the of the cells in the I-direction to be connected to the aquifer and must be greater than or equal to II and less than or equal to NX			NX
9	J1	A positive integer that defines the lower bound of the cells in the J-direction to be connected to the aquifer and must be greater than or equal to one and less than or equal to J2 and NY.			1
10	J2	A positive integer that defines the upper bound of the cells in the J-direction to be connected to the aquifer and must be greater than or equal to JI and less than or equal to NY.			NY
11	K1	A positive integer that defines the lower bound of the cells in the K-direction to be to be connected to the aquifer and must be greater than or equal to one and less than or equal to K2 and NZ.			1
12	K2	A positive integer that defines the upper bound of the cells in the K-direction to be connected to the aquifer and must be greater than or equal to KI and less than or equal to NZ.			NZ
13	AQUFACE	AQUFACE is a character string that sets the connection "face" of the cells declared by this record and should be set to one of the following: 1) X+, Y+, or Z+ for the positive direction, or X-, Y- or Z- for the negative direction transmissibilities. 2) I+, J+, or K+ for the positive direction, or I-, J- or K- for the negative direction transmissibilities.			None
14	SALTCON	SALTCON is a real positive number that defines the initial salt concentration in the aquifer. This variable is ignored by OPM Flow.			
		lb/stb	kg/sm^3	gm/scc	0.0

Notes:

1) Where NX, NY and NZ are the dimensions of the model as defined on the DIMENS keyword in the RUNSPEC section.
2) The keyword is followed by up to NANAQU records as defined on the AQUDIMS keyword in the RUNSPEC section
3) The keyword should be terminated by a "/".

Table 10.7:AQUFET Keyword Description

Note this keyword should only be used in equilibration and enumerated initialize runs, that is it should be omitted from RESTART runs.

Note

If the model is unstable then this may be due to an aquifer not being in equilibrium with the connecting reservoir blocks, for example the aquifer is connected to only hydrocarbon reservoir cells. Try commenting out the aquifer and see if this resolves the instabilities.

Example

Given the following grid and aquifer dimensions in the RUNSPEC section:

```
-- ==================================================================================
--
-- RUNSPEC SECTION
--
-- ==================================================================================
RUNSPEC   --
--       MAX     MAX     MAX
--       NDIVIX  NDIVIY  NDIVIZ
DIMENS
         20      1       5                                                       /

--       AQF     AQF     AQF     AQF     AQF     AQF     AQF     AQF
--       MXAQN   MXNAQC  NIFTBL  NRIFTB  NANAQU  NCAMAX MXNALI MXAAQL
AQUDIMS
         1*      1*      5       100     1       1*      1*      1*              /
```

The Fetkovich Analytical aquifer is defined in the SOLUTION sections as:

```
-- ==================================================================================
--
-- SOLUTION SECTION
--
-- ==================================================================================
SOLUTION --
--                        FETKOVICH AQUIFER DESCRIPTION AND CONNECTIONS
--
--       DATUM  AQF    AQF    AQF     AQF  AQF -------- BOX ------- CONNECT SALT
--       DEPTH  PRESS  VOLM   COMP    PI   PVT I1  I2  J1  J2  K1  K2 FACE  CONC
--
AQUFET
         1130.  1*    1.0E+12 3.0E-5  500E3 1   1   1   1   1   1   1  'J-'  /
```

Here one Fetkovich Analytical aquifer is connected to a single cell (1, 1, 1) at the J- face (or X- face) of the grid.

10.3.14 AQUFETP – DEFINE FETKOVICH ANALYTICAL AQUIFERS

Description

The AQUFETP keyword defines Fetkovich[292] analytical aquifers and the aquifer properties. Each row entry in the AQUFETP keyword defines one Fetkovich analytical aquifer. In order to fully define this type of aquifer, the aquifer must be connected to the reservoir using the AQUANCON keyword in the GRID or SOLUTION sections.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	AQUID	A positive integer greater than or equal to one and less than or equal to NANAQ on the AQUDIMS keyword in the RUNSPEC section, that defines the Fetkovich aquifer number.			1
2	DATUM	DATUM is a single positive value that defines the Fetkovich reference datum depth for PRESS.			
		feet	m	cm	None
3	PRESS	PRESS is a single positive value that defines the aquifer pressure at DATUM. If PRESS is defaulted then the simulator will set the aquifer's initial reservoir pressure to be in equilibrium with the cells the aquifer is contacted to. Defaulting this parameter will avoid inconsistent equilibration pressures between the reservoir cells and the aquifer.			
		psia	barsa	atma	1*
4	PORV	A real positive value that defines the initial water volume of the aquifer.			
		stb	sm^3	scc	None
5	COMP	COMP is a real number defining the total compressibility (Ct) of the aquifer, that is the rock compressibility (Cf) plus the water compressibility (Cw) at the aquifer datum pressure (DATUM) and is defined as: $C_t = C_f + C_w$			
		1/psia	1/barsa	1/atma	None
6	PI	A real positive number that defines the aquifer productivity index based on the aquifer influx rate per unit pressure drop.			
		stb/d/psia	sm^3/d/barsa	scc/hr/atma	None
7	PVTW	A positive integer that defines the aquifer's PVTW water property table.			1

[292] *Fetkovich, M. J. "A Simplified Approach to Water Influx Calculations - Finite Aquifer Systems," Journal of Petroleum Technology, (1971) 23, No. 7, 814-828.*

No.	Name	Description			Default
		Field	Metric	Laboratory	
8	SALTCON	SALTCON is a real positive number that defines the initial salt concentration in the aquifer, for when the simulator's Brine Model has been activated via the BRINE keyword in the RUNSPEC section. This variable is ignored by OPM Flow.			
		lb/stb	kg/sm^3	gm/scc	0.0
9	TEMP	TEMP is a real positive number that defines the initial temperature of the aquifer at DATUM for use with OPM Flow's thermal option. The THERMAL keyword in the RUNSPEC section must be activated to use this option.			
		°F	°C	°C	1*

Notes:

1) The keyword is followed by up to NANAQ records as defined on the AQUDIMS keyword in the RUNSPEC section

2) Each record is terminated by a "/" and the keyword should be terminated by a "/". Note the commercial simulator only requires a terminating "/" if the number of records on this keyword are less than NANAQ on the AQUDIMS keyword in the RUNSPEC section. If the number of records are equal to NANAQ and a terminating "/" has been entered then the commercial simulator will issue a warning message; however, the commercial simulator run will proceed as expected.

Table 10.8:AQUFETP Keyword Description

Note this keyword should only be used in equilibration and enumerated initialize runs, that is it should be omitted from RESTART runs.

Note

If the model is unstable then this may be due to an aquifer not being in equilibrium with the connecting reservoir blocks, for example if the aquifer is connected to some hydrocarbon reservoir cells. Try commenting out the aquifer and see if this resolves the instabilities, and if so amend the aquifer connections accordingly.

Example

Given the following grid and aquifer dimensions in the RUNSPEC section:

```
-- ======================================================================
--
-- RUNSPEC SECTION
--
-- ======================================================================
RUNSPEC
--
--       MAX     MAX     MAX
--       NDIVIX  NDIVIY  NDIVIZ
DIMENS
         20      1       5                                                /

--       AQF     AQF     AQF     AQF     AQF     AQF     AQF     AQF
--       MXAQN   MXNAQC  NIFTBL  NRIFTB  NANAQU  NCAMAX MXNALI MXAAQL
AQUDIMS
         1*      1*      5       100     1       1*      1*      1*              /
```

The Fetkovich analytical aquifer is defined in the SOLUTION sections as:

```
-- ======================================================================
--
-- SOLUTION SECTION
--
-- ======================================================================
SOLUTION --
--                       FETKOVICH AQUIFER DESCRIPTION
--
--       ID   DATUM   AQF    AQF    AQF     AQF    AQF   SALT
--       NUM  DEPTH   PRESS  VOLM   COMP    PI     PVT   CONC
--
AQUFETP
         1   1130.   1*    1.0E+12 3.0E-5  500E3  1     0.0                  /
/
```

And the connection of the aquifer is set in the GRID or the SOLUTION sections as:

```
--
--                       ANALYTIC AQUIFER CONNECTION
--
--       ID     ---------- BOX ---------   CONNECT  AQF    AQF      ADJOIN
--       NUMBER I1  I2   J1  J2   K1  K2   FACE     INFLX  MULTI    CELLS
AQUANCON
         1      1   1    1   1    1   1     J-      1.0    1.0      'NO'        /
/
```

Here one Fetkovich analytical aquifer is connected to a single cell (1, 1, 1) at the J- face (or Y- face) of the cell.

10.3.15 AQUFLUX - DEFINE CONSTANT FLUX ANALYTICAL AQUIFER

Description

The AQUFLUX keyword defines the properties of Constant Flux Analytical Aquifers, that allows for a constant water influx to the connected grid blocks. This type of aquifer is connected to the model using either the AQUANCON keyword for global cells, or the AQUANCONL keyword for cells belonging to a Local Grid Refinement ("LGR"), both the aforementioned keywords are in the GRID and SOLUTION sections. The keyword itself may be utilized in both the SOLUTION and SCHEDULE sections, with subsequent entries overwriting the previous entry.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	AQUID	A positive integer greater than or equal to one and less than or equal to NANAQ on the AQUDIMS keyword in the RUNSPEC section, that defines the AQUFLUX aquifer number.			1
2	AQFLUX	A real positive value the defines the aquifer water influx rate, per unit area of the connected grid cell face.			
		stb/day/ft2	sm^3/day/m2	scc/hour/cm2	None
3	SALTCON	SALTCON is a real positive number that defines the initial salt concentration in the aquifer, for when with simulator's Brine Model has been activated via the BRINE keyword in the RUNSPEC section. This variable is ignored by OPM Flow.			
		lb/stb	kg/sm^3	gm/scc	0.0
4	TEMP	TEMP is a real positive number that defines the initial temperature of the aquifer for use with OPM Flow's thermal option. The THERMAL keyword in the RUNSPEC section must be activated to use this option. If the THERMAL keyword is absent from the input deck, then the parameter is ignored.			
		°F	°C	°C	1*
5	PRESS	PRESS is a single positive value that defines the aquifer pressure for use with OPM Flow's thermal option. The THERMAL keyword in the RUNSPEC section must be activated to use this option. If the THERMAL keyword is absent from the input deck, then the parameter is ignored.			
		psia	barsa	atma	1*

Notes:

1) The keyword is followed by up to NANAQ records as defined on the AQUDIMS keyword in the RUNSPEC section

2) Each record is terminated by a "/" and the keyword should be terminated by a "/". Note the commercial simulator only requires a terminating "/" if the number of records on this keyword are less than NANAQ on the AQUDIMS keyword in the RUNSPEC section. If the number of records are equal to NANAQ and a terminating "/" has been entered then the commercial simulator will issue a warning message; however, the commercial simulator run will proceed as expected.

Table 10.9:AQUFLUX Keyword Description

The water flow rate into a connected grid cell for this type of aquifer, is calculated from:

$$q_w = AQUFLUX(AQFLUX) \times A_{(i,j,k)} \times AQUANCON(AQUCOEF) \quad (10.14)$$

Where:

AQUFLUX(AQFLUX) = the AQFLUX parameter on the AQUFLUX keyword,
AQUANCON(AQUCOEF) = the AQUCOEF parameter on the AQUANCON keyword

Note that $A_{(i,j,k)}$ is calculated from the connected cell geometry, and thus the AQUANCON(AQUFLUX) and the AQUANCONL(AQUFLUX) parameters are ignored for this type of aquifer.

The AQUFLUX keyword should only be used in equilibration and enumerated initialize runs, that is it should be omitted from RESTART runs.

See also the *BC – Define Boundary Conditions* keyword for OPM Flow's boundary model facilities that can be used instead of AQUFLUX, and allows for additional flexibility.

Example

Given a total of five analytical aquifers as declared by the NANAQ parameter on the AQUDIMS keyword being set to five, of which three are Constant Flux Aquifers and two Carter-Tracy Aquifers, then:

```
--
--                        CONSTANT FLUX AQUIFER DESCRIPTION
--
--       ID   WATER    SALT   AQF    AQU
--       NUM  INFLUX   CONC   TEMP   PRES
--
AQUFLUX
         1   0.0004   1*     1*     1*                    /
         2   0.0005   1*     1*     1*                    /
         3   0.0003   1*     1*     1*                    /
/
--
--                        CARTER-TRACY AQUIFER DESCRIPTION
--
--       ID   DATUM   AQF    AQF    AQF    AQF     AQF   AQF   INFL   PVT  AQU
--       NUM  DEPTH   PRESS  PERM   PORO   RCOMP   RE    DZ    ANGLE  NUM  TAB
--
AQUCT
         4   2000.0  269    100.0  0.30   3.0e-5  330   10.0  360.0   1    2   /
         5   2000.0  269    100.0  0.30   3.0e-5  330   10.0  360.0   1    2   /
/
```

Defines three Constant Flux Aquifers and three Carter-Tracy Aquifers, and the connection of the aquifers are set in the GRID or SOLUTION sections via:

```
--
--                        ANALYTIC AQUIFER CONNECTION
--
--       ID     ---------- BOX ---------   CONNECT  AQF    AQF      ADJOIN
--       NUMBER I1  I2   J1  J2   K1  K2   FACE     INFLX  MULTI    CELLS
AQUANCON
         1      1   198  1   14   1   10    J-       1*     1.0      'NO'        /
         2      1   198  1   14   12  22    J-       1*     1.0      'NO'        /
         3      1   198  1   14   23  54    J-       1*     1.0      'NO'        /
         4      1   198  1   14   61  70    J-       1*     1.0      'NO'        /
         5      1   198  15  172  71  71    K+       1*     1.0      'NO'        /
/
```

10.3.16 BC – DEFINE BOUNDARY CONDITIONS

Description

The BC keyword defines the boundary conditions for the model, and can be used to set boundary conditions for when external influx or efflux volumes are influencing the reservoir pressure and production history. For example, when the average reservoir pressure remains constant throughout the production period due to water influx, or gas migration from an external source.

Note

This is an OPM Flow specific keyword and will therefore cause an error in the commercial simulator.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	I1	A positive integer that defines the lower bound of the grid in the I-direction for which the boundary conditions are to be applied, must be greater than or equal 1 and less than or equal to I2 and NX.			1
2	I2	A positive integer that defines the upper bound of the grid in the I-direction for which the boundary conditions are to be applied, must be greater than or equal to I1 and less than or equal to NX			NX
3	J1	A positive integer that defines the lower bound of the grid in the J-direction for which the boundary conditions are to be applied, must be greater than or equal 1 and less than or equal to J2 and NY.			1
4	J2	A positive integer that defines the upper bound of the grid in the J-direction for which the boundary conditions are to be applied, must be greater than or equal to J1 and less than or equal to NY.			NY
5	K1	A positive integer that defines the lower bound of the grid in the K-direction for which the boundary conditions are to be applied, must be greater than or equal to one and less than or equal to K2 and NZ.			1
6	K2	A positive integer that defines the upper bound of the grid in the K-direction for which the boundary conditions are to be applied, must be greater than or equal to K1 and less than or equal to NZ.			NZ
7	DIRECT	A character string that defines the direction to apply the boundary conditions, and should be set to one of the following X, Y, or Z for the positive direction, or X-, Y- or Z- for the negative direction.			None

No.	Name	Description			Default
		Field	Metric	Laboratory	
8	TYPE	A defined character string that defines the type of boundary condition to be applied, and should be set to one of the following character strings: 1) DIRICHLET: for a user defined boundary conditions. In this case, items (1) through (7) must be set, together with PHASE for the fluid type, and PRESS and TEMP for the constant pressure and temperature boundary conditions. This option is currently not supported but will be available in the next release. 2) FREE: for the initial state of the boundary to be kept throughout the simulation, that is a constant boundary condition. For this option only items (1) through (7) need to be defined. 3) RATE: for the boundary to have a constant influx or efflux rate. Again items (1) through (7) are required, plus PHASE for the fluid type, and RATE to set the PHASE rate. Only the FREE and RATE options are currently supported; however the next release will support the DIRICHLET option.			None
9	PHASE	A defined character string that sets fluid type used in the boundary calculations, and should be set to one of the following character strings: 1) GAS: the gas phase will be used to control the boundary conditions for when the TYPE has been set to DIRICHLET or RATE. 2) OIL: the oil phase will be used to control the boundary conditions for when the TYPE has been set to DIRICHLET or RATE. 3) WATER or WAT: the water phase will be used to control the boundary conditions for when the TYPE has been set to DIRICHLET or RATE.			None
10	RATE	A real value that defines the constant surface oil, gas or water rate to be injected or withdrawn at the boundary, for when TYPE has been set to RATE. Note a negative value implies an injection rate, whereas, a positive value indicates a withdrawal rate.			
		Liquid: stb/day Gas: Mscf/day	Liquid: sm^3/day Gas: sm^3/day	Liquid: scc/hr Gas: scc/hr	0.0
11	PRESS	PRESS is a real positive value that defines the constant pressure boundary condition. PRESS should only be entered if TYPE has been set to DIRICHLET. If the pressure at the boundary is less than PRESS, then the fluid type declared via PHASE will flow across the boundary. The default value of 1* will use the simulator's calculated value based on data entered via the EQUIL keyword in the SOLUTION section.			
		psia	barsa	atma	1*
12	TEMP	TEMP is a real positive number that defines the constant temperature boundary condition. TEMP should only be entered if TYPE has been set to DIRICHLET. The default value of 1* will use the simulator's calculated value based on data entered via one of the following reservoir temperature keywords: RTEMP, RTEMPA, RTEMPVD, TEMPI, or TEMPVD, in the SOLUTION section. Note that all of the aforementioned reservoir temperature keywords, except for TEMPI, may also be used in the PROPS section as well.			1*

No.	Name	Description			Default
		Field	Metric	Laboratory	
		°F	°C	°C	

Notes:

1) Where NX, NY and NZ are the dimensions of the model as defined on the DIMENS keyword in the RUNSPEC section.
2) Each record must be terminated by a "/" and the keyword is terminated by a "/".

Table 10.10: BC Keyword Description

See also the *AQUFLUX - Define Constant Flux Analytical Aquifer* keyword that is supported by OPM Flow in both the SOLUTION and SCHEDULE sections, to define the aquifer influx with greater flexibility.

If the BC keyword is not present in the input deck, then the boundary conditions for the model are set to be no flow, which is the normal behavior in both OPM Flow and the commercial simulator.

Examples

The first example shows how to set a constant pressure boundary using TYPE equal to FREE:

```
--
--        DEFINE MODEL BOUNDARY CONDITIONS (OPM FLOW SOLUTION GRID KEYWORD)
--
--        ---------- BOX ---------    BC    BC         BC     BC     BC     BC
--        I1  I2   J1  J2   K1  K2   DIRC  TYPE       PHASE  RATE  PRESS  TEMP
BC
          1   1    1   1*   1   1*   X-    FREE       1*     1*    1*     1*    /
          1   1*   1   1    1   1*   Y     FREE                                /
/
```

With this option it is only necessary to define the boundary cells and all the other parameters (PHASE, RATE, PRESS, and TEMP) can be defaulted, as they are ignored for when TYPE equals FREE.

The next example is based on NX, NY and NZ equal to 20, 1, 10 respectively, on the DIMENS keyword in the RUNSPEC section, and shows how different boundary types can be assigned to different parts of the model.

```
--
--        DEFINE MODEL BOUNDARY CONDITIONS (OPM FLOW SOLUTION GRID KEYWORD)
--
--        ---------- BOX ---------    BC    BC         BC     BC     BC     BC
--        I1  I2   J1  J2   K1  K2   DIRC  TYPE       PHASE  RATE  PRESS  TEMP
BC
          1   1    1   1    1   10   X-    RATE       GAS    1*    256.0  100.0 /
          20  20   1   1    1   10   X     FREE       4*                        /
/
```

The last example shows how the DIRICHLET boundary condition option may be used:

```
--
--       DEFINE MODEL BOUNDARY CONDITIONS (OPM FLOW SOLUTION GRID KEYWORD)
--
--       ---------- BOX ---------    BC    BC         BC     BC    BC     BC
--       I1  I2   J1  J2   K1  K2    DIRC  TYPE       PHASE  RATE  PRESS  TEMP
BC
         1   1    1   1*   1   1*    X-    DIRICHLET  WAT    1*    256.0  100.0 /
         1   1*   1   1    1   1*    Y     DIRICHLET  WAT    1*    1*     100.0 /
/
```

Here, the first line sets both the pressure and temperature at the boundary, and the second line defaults the pressure entry, so that the simulator calculated initial boundary pressure will be used.

10.3.17 BOUNDARY – DEFINE A BOUNDARY BOX FOR PRINTING

The BOUNDARY keyword defines a rectangular grid for printing various arrays to the output print file (*.PRT); thus, avoiding printing all the elements in the selected array.

See *BOUNDARY – Define a Boundary Box for Printing* in the GRID section for a full description.

10.3.18 BOX - DEFINE A RANGE OF GRID BLOCKS TO ENTER PROPERTY DATA

BOX defines a range of grid blocks for which subsequent data is assigned for all the cells in the defined BOX. Note that the BOX grid is reset by the keyword ENDBOX by resetting the current defined BOX to be the whole grid. The keyword can be used for any array and for all grid types.

See *BOX - Define a Range of Grid Blocks to Enter Property Data* in the GRID section for a full description.

10.3.19 COLUMNS – DEFINE INPUT FILE COLUMN MARGINS

The COLUMNS keyword defines the input file column margins; characters outside the margins are ignored by the input parser.

See *COLUMNS – Define Input File Column Margins* in the GLOBAL section for a full description.

10.3.20 COPY – COPY ARRAY DATA TO ANOTHER ARRAY

The COPY keyword copies an array (or part of an array) to another array or part of an array. The arrays can be real or integer depending on the array type; however, the arrays that can be operated on is dependent on which section the COPY keyword is being used.

See *COPY – Copy Array Data to Another Array* in the GRID section for a full description.

10.3.21 COPYREG – COPY AN ARRAY TO ANOTHER ARRAY BASED ON A REGION NUMBER

The COPYREG keyword copies a specified array or part of an array based on cells with a specific region number to another array. The region number array can be FLUXNUM, MULTNUM or OPERNUM and these arrays must be defined and be available before the COPYREG keyword is read by the simulator. The property arrays can be real or integer depending on the property array type; however, the property arrays that can be operated on is dependent on which section the COPYREG keyword is being used.

See *COPYREG – Copy an Array to Another Array based on a Region Number* in the GRID section for a full description.

10.3.22 DATUM – DEFINE THE DATUM DEPTH FOR THE MODEL

Description

The DATUM keyword defines the datum depth for the model. This allows for all grid block pressures and potentials to be calculated at a common depth.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	DATUM	DATUM is a single positive value that defines the datum depth for the model.			
		feet	m	cm	None
Notes: 1) The keyword is terminated by a "/".					

Table 10.11: DATUM Keyword Description

See also the DATUMR and DATUMRX keywords in the SOLUTION section that also define the datum depth for the model.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

Example

```
--        DATUM
--        DEPTH
--        ------
DATUM
          5000.0                                    / DATUM DEPTH FOR REPORTING
```

The above example defines the datum for the model to be 5000.0

10.3.23 DATUMR – DEFINE DATUM DEPTHS FOR THE FIPNUM REGIONS

Description

The DATUMR keyword defines the datum depth for each fluid in-place region (FIPNUM) declared in the model. This allows for all grid block pressures and potentials to be calculated at a common depth within a FIPNUM region.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	DATUMR	DATUMR is a vector of positive values that defines the datum depth for each fluid in-place region.			
		feet	m	cm	None
Notes: 1) The keyword is followed by FIPNUM values as defined on REGDIMS keyword in the RUNSPEC section. 2) The keyword is terminated by a "/".					

Table 10.12: DATUMR Keyword Description

See also the DATUM and DATUMRX keywords in the SOLUTION section that also define the datum depth for the model.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

Example

```
--
--        DATUM
--        DEPTH
--        ------
DATUMR
          4800.0
          4900.0
          5000.0                                  / DATUM DEPTH FOR REPORTING
```

The above example defines the datum depth for three FIPNUM regions, for when FIPNUM has been set equal to three on the REGDIMS keyword in the RUNSPEC section.

10.3.24 DATUMRX – DEFINE DATUM DEPTHS FOR THE FIP ALLOCATED REGIONS

Description

The DATUMRX keyword defines the datum depth for each fluid in-place family region defined by the FIP keyword. This allows for all grid block pressures and potentials to be calculated at a common depth within a given FIP region. The FIP keyword in the REGION sections allows one to define additional sets of fluid in-place regions to the standard FIPNUM keyword. For example, one could use FIPNUM to define the reservoir layers as fluid in-place regions and the FIP keyword to define he fluid in-place region for fault blocks.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	FIPNAME	A character string of up to five characters in length that defines the FIP family name for which the datum depth data is being defined. The default value of 1* will set DATUMR to the standard FIPNUM region numbers.			1*
2	DATUMR	DATUMR is a vector of positive values that defines the datum depth for each fluid in-place family region. There must be one entry for each region in the FIP family name. A maximum of NTFIP, as declared by the REGDIMS keyword in the RUNSPEC, values may be entered for each FIPNAME entry.			
		feet	m	cm	None

Notes:

1) The keyword is followed any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".

Table 10.13: DATUMR Keyword Description

See also the FIP keyword in the REGIONS section to define FIP family regions, and the DATUM and DATUMR keywords in the SOLUTION section that also define the datum depth for the model.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

Example

```
--
--        FIP        DATUM
--        NAME       DEPTH
DATUMRX
          'FLTBL'    5000.0  5000.0  5000.0  5000.0    / DATUM DEPTH FOR REPORTING
          'LICBL'    5000.0  5050.0                    / DATUM DEPTH FOR REPORTING
/
```

The above example defines the datum depth for two FIP families, FLTBL and LICBL, with the datum set to a constant 5000.0 psia for FLTBL family and different values for each of the regions in the LICBL family of regions.

10.3.25 DEBUG – DEFINE THE DEBUG DATA TO BE PRINTED TO FILE

This keyword defines the debug data to be written to the debug file (*.DBG). This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

See *DEBUG – Define the Debug Data to be Printed to File* in the GLOBAL section for a full description.

10.3.26 DYNAMICR – START OF DYNAMIC REGION PARAMETER DEFINITION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The DYNAMICR keyword marks the start of a Dynamic Region section and defines the parameters used for Dynamic Regions that allows for property and reporting regions to vary as the run progresses, based on the parameters and logic defined by this keyword and section. A Dynamic Region section is terminated by the ENDDYN keyword in the SOLUTION or SCHEDULE sections.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

10.3.27 ECHO – ACTIVATE ECHOING OF USER INPUT FILES TO THE PRINT FILE

Turns on echoing of all the input files to the print file; note that this keyword is activated by default and can subsequently be switched off by the NOECHO activation keyword.

See *ECHO – Activate Echoing of User Input Files to the Print File* in the GLOBAL section for a full description.

10.3.28 END – DEFINE THE END OF THE INPUT FILE

This keyword marks the end of the input file and can occur in any section. Any keywords and data after the END keyword are ignored.

See *END – Define the End of the Input File* in the GLOBAL section for a full description.

10.3.29 ENDBOX – DEFINE THE END OF THE BOX DEFINED GRID

This keyword marks the end of a previously defined BOX sub-grid as defined by a previously entered BOX keyword. The keyword resets the input grid to be the full grid as defined by the NX, NY, and NZ variables on the DIMENS keyword in the RUNSPEC section.

See *ENDBOX – Define the End of the BOX Defined Grid* in the GRID section for a full description.

10.3.30 ENDDYN– END OF DYNAMIC REGION PARAMETER DEFINITION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The ENDDYN keyword marks the end of a Dynamic Region section that was started with the DYNAMICR keyword in the SOLUTION or SCHEDULE sections. Dynamic Regions allow for property and reporting regions to vary as the run progresses, based on the parameters and logic defined within the section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

10.3.31 ENDFIN – END THE DEFINITION OF A LOCAL GRID REFINEMENT

ENDFIN defines the end of a Cartesian or radial local grid refinement ("LGR") definition and a LGR property definition data set.

See *ENDFIN – End the Definition of a Local Grid Refinement* in the GRID section for a full description.

10.3.32 ENDINC – DEFINE THE END OF AN INCLUDE FILE

This keyword marks the end of an include file specified on the INCLUDE keyword. When the ENDINC keyword is encountered in the INCLUDE file, input data is read from the next keyword in the current file. Any keywords and data after the ENDINC keyword in the INCLUDE file are ignored.

See *ENDINC – Define the End of an Include File* in the GLOBAL section for a full description.

10.3.33 ENDSKIP – DEACTIVATE SKIPPING OF KEYWORDS AND INPUT DATA

Turns off skipping of keywords that was activated by the SKIP, SKIP100, or SKIP300 keywords. Note that for each SKIP keyword activated there must be a corresponding ENDSKIP keyword.

See *ENDSKIP – DeActivate Skipping of Keywords and Input Data* in the GLOBAL section for a full description.

10.3.34 EQUALREG – SETS AN ARRAY TO A CONSTANT BY REGION NUMBER

The EQUALREG keyword sets a specified array to a constant for cells with a specific region number. The region number array can be FLUXNUM, MULTNUM or OPERNUM and these arrays must be defined and be available before the EQUALREG keyword is read by the simulator. The constant can be real or integer depending on the property array type; however, the property arrays that can be operated on is dependent on which section the EQUALREG keyword is being used.

See *EQUALREG – Sets an Array to a Constant by Region Number* in the GRID section for a full description.

10.3.35 EQUALS – SETS A SPECIFIED ARRAY TO A CONSTANT

The EQUALS keyword sets a specified array or part of an array to a constant. The constant can be real or integer depending on the array type; however, the arrays that can be operated on is dependent on which section the EQUALS keyword is being used.

See *EQUALS – Sets a Specified Array to a Constant* in the GRID section for a full description.

10.3.36 EQUIL – DEFINE THE EQUILIBRATION INITIALIZATION DATA

Description

This keyword defines the parameters used to initialize the model for when equilibration is calculated by OPM Flow. This is the standard methodology to initialize a model, the non-standard formulation of entering the pressures and saturations for each grid cell is seldom employed in the industry. The keyword can be used with all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	DATUM	DATUM is a single positive value that defines the reference datum depth for PRESS.			
		feet	m	cm	0.0
2	PRESS	PRESS is a single positive value that defines the pressure at DATUM. If the DATUM depth lies above the GOC then PRESS is the pressure with respect to the gas phase. If the DATUM depth is below OWC then PRESS refers to the water phase pressure. Otherwise, PRESS refers to the oil phase pressure.			
		psia	barsa	atma	0.0
3	WATCONT	1) For three phase runs containing oil, gas and water WATCONT is the depth of the oil-water contact (OWC). 2) For two phase runs containing oil and water WATCONT is the depth of the oil-water contact (OWC). 3) For two phase runs containing gas and water WATCONT is the depth of the gas-water contact (GWC).			
		feet	m	cm	0.0
4	WATCAP	1) For three phase runs containing oil, gas and water WATCAP is the oil-water capillary pressure at the OWC. 2) For two phase runs containing oil and water WATCAP is the oil-water capillary pressure at the OWC. 3) For two phase runs containing gas and water WATCAP is the gas-water capillary pressure at the GWC			
		psia	barsa	atma	0.0

No.	Name	Description			Default
		Field	Metric	Laboratory	
5	GASCONT	1) For three phase runs containing oil, gas and water GASCONT is the depth of the gas-oil contact (GOC). 2) Note in cases where there is no gas cap (or free gas) then GASCONT should be set to a value shallower than the top of the reservoir. 3) In cases where there is initially no oil zone, as for a gas condensate field for example, the GASCONT should be set to the same depth as WATCONT. 4) For two phase runs containing oil and water, or gas and water, GASCONT is ignored.			
		feet	m	cm	0.0
6	GASCAP	1) For three phase runs containing oil, gas and water GASCAP is the gas-oil capillary pressure at the GWC. 2) For two phase runs containing oil and water, or gas and water, GASCAP is ignored.			
		psia	barsa	atma	0.0
7	EQLOPT1	EQLOPT1 is an integer value that sets the initialization option for when dissolved gas is present in the run, as activated by the DISGAS keyword in the RUNSPEC section. 1) A positive value of EQLOPT1 results in the gas-oil ratio being calculated from data entered on the PBVD (saturation pressure or bubble-point pressure versus depth table) or the RSVD keyword (gas-oil ratio versus depth table). If this option is selected, then either the PBVD or RSVD keywords must be present in the input deck. 2) Note that the allocation of multiple PBVD and RSVD tables to each grid cell is through the EQLNUM keyword and not the PVTNUM keyword. 3) A zero value of EQLOPT1 results in the gas-oil ratio being set to the saturated gas-oil ratio at the GOC. In this case DATUM must be equal GASCONT and the PBVD and RSVD keywords may be omitted. 4) A negative value of EQLOPT1 results in the same option for when EQLOPT1 is zero. EQLOPT1 is ignored if there is no dissolved gas in the run.			
		dimensionless	dimensionless	dimensionless	0

No.	Name	Description			Default
		Field	Metric	Laboratory	
8	EQLOPT2	EQLOPT2 is an integer value that sets the initialization option for when vaporized oil (condensate) is present in the run, as activated by the VAPOIL keyword in the RUNSPEC section. 1) A positive value of EQLOPT2 results in the condensate-gas ratio being calculated from data entered on the PDVD (saturation pressure or dew point pressure versus depth table) or the RVVD keyword (condensate-gas ratio versus depth table). If this option is selected, then either the PDVD or RVVD keywords must be present in the input deck 2) Note that the allocation of multiple PDVD and RVVD tables to each grid cell is through the EQLNUM keyword and not the PVTNUM keyword. 3) A zero value of EQLOPT2 results in the condensate-gas ratio being set to the saturated condensate-gas ratio at the GOC. In this case DATUM must be equal GASCONT and the PDVD and RVVD keywords may be omitted. 4) A negative value of EQLOPT2 results in the same option for when EQLOPT2 is zero. EQLOPT2 is ignored if there is no vaporized oil in the run.			
		dimensionless	dimensionless	dimensionless	0
9	EQLOPT3	EQLOPT3 is an integer value that sets the initialization accuracy options for the equilibration calculation. 1) A zero value of EQLOPT3 results in OPM Flow using the fluid saturations at the center of the grid block in the equilibration calculation. This results in a stable initialization at the expense of a potentially less accurate fluid in-place calculation, especially for large thick grid blocks with a fluid contact in the block. 2) A negative value of EQLOPT3 results in the simulator dividing each grid cell into $2\,\lvert N\rvert + 1$ horizontal sub-blocks for the equilibration calculation. This results in an accurate fluid in-place calculation at the expense of initialization stability, that is there may be some movement of fluids when there is no production at the start of the run. Increasing the value of N increases the accuracy of the calculation, with the maximum value of N being set to 20 by OPM Flow. 3) A positive value of EQLOPT3 results in the same option for when EQLOPT3 is negative, except that tilted fault blocks are used in the calculation. Again, increasing the value of N increases the accuracy of the calculation, with the maximum value of N being set to 20 by OPM Flow. Note this option should be used with Irregular Corner-Point Grids. EQLOPT3 is ignored for Radial Grids. Only EQLOPT3 options (1) and (2), that is values less than and equal to zero, are supported by OPM Flow.			
		dimensionless	dimensionless	dimensionless	0

No.	Name	Description			Default
		Field	Metric	Laboratory	
10	EQLOPT4	A positive integer value greater than or equal one and less than or equal to three, that sets the initialization option in the commercial compositional simulator. EQLOPT4 should be defaulted with 1*, as it is not used by OPM Flow.			None
		dimensionless	dimensionless	dimensionless	
11	EQLOPT5	A positive integer value that if set to one forces PRESS to be used for the datum pressure in the commercial compositional simulator. EQLOPT5 should be defaulted with either 1*, as it is not used by OPM Flow.			None
		dimensionless	dimensionless	dimensionless	
12	EQLOPT6	EQLOPT6 is an integer value that sets the initialization option for when vaporized water is present in the run, as activated by the VAPWAT keyword in the RUNSPEC section. Note this is an OPM Flow specific parameter for use with simulator's Vaporized Water Model. 1) A positive value of EQLOPT6 results in the vaporized water-gas ratio being calculated from data entered on the RVWVD keyword (vaporized water-gas ratio versus depth table). If this option is selected, then the RVWVD keyword must be present in the input deck Note that the allocation of multiple RVWVD tables to each grid cell is through the EQLNUM keyword and not the PVTNUM keyword. 2) A zero value of EQLOPT2 results in the vaporized water-gas ratio being set to the saturated vaporized water-gas ratio at the GOC. In this case DATUM must be equal GASCONT and the RVWVD keyword may be omitted. 3) A negative value of EQLOPT6 results in the same option for when EQLOPT6 is zero. EQLOPT6 is ignored if there is no vaporized water in the run.			
		dimensionless	dimensionless	dimensionless	0

Notes:

1) The keyword is followed by NTEQUL records as declared on the EQLDIMS keyword in the RUNSPEC section.
2) Each record is terminated by a "/" and there is no "/" terminator for the keyword.

Table 10.14: EQUIL Keyword Description

Note

A common method to initialize a model is by using the SWATINIT property array to set the initial water saturation for each cell in the model. This property is normally exported from a static model, where Saturation Height Functions ("SHF") have been used to describe the water saturation profile with depth. In the dynamic model capillary pressure functions are used to described the water profile versus depth.

Note that if the SWATINIT array has been used to initialize the model then the fine grid block initialization via the EQLOPT3 variable, should not normally be used, and should be defaulted or set equal to zero; otherwise, the resulting water saturation will not strictly honor the SWATINIT array.

See also the PRESSURE, SGAS, SOIL and SWAT keywords in the SOLUTION section to initialize the model using the non-standard formulation of entering the pressures and saturations for each grid cell.

Example

```
--
--        DATUM   DATUM   OWC     PCOW   GOC     PCGO   RS   RV   N    E300  RVW
--        DEPTH   PRESS   DEPTH   ----   DEPTH   ----   OPT  OPT  OPT  OPTS  OPT
EQUIL
          3650.0  1560.0  3712.0  0.00   1000.0  0.00   1    0    -5   2*    1*   /
          3650.0  1560.0  3741.0  0.00   1000.0  0.00   1    0    -5   2*    1*   /
          3650.0  1560.0  3741.0  0.00   1000.0  0.00   1    0    -5   2*    1*   /
```

The above example defines three equilibration records for when NTEQUL equals three on the EQLDIMS keyword in the RUNSPEC section. Here there is no gas cap and the GOC has been set to a value above the reservoirs (1000.0), and the value of EQLOPT3 (-5) has been explicitly stated.

10.3.37 EXTRAPMS – ACTIVATE EXTRAPOLATION WARNING MESSAGES

The EXTRAPMS keyword activates extrapolation warning messages for when OPM Flow extrapolates the PVT or VFP tables. Frequent extrapolation warning messages should be investigated and resolved as this would indicate possible incorrect data and may result in the simulator extrapolating to unrealistic values.

See *EXTRAPMS – Activate Extrapolation Warning Messages* in the GLOBAL section for a full description.

10.3.38 FILEUNIT – ACTIVATE UNIT CONSISTENCY VERIFICATION

The FILEUNIT keyword defines the units of the of the data set, and is used to verify that the units in the input deck and any associated include files are consistent grid data. The keyword does not provide for the conversion between different sets of units.

See *FILEUNIT – Activate Unit Consistency Checking* in the GRID section for a full description.

10.3.39 FORMFEED – DEFINED THE PRINT FILE FORM-FEED CHARACTER

The FORMFEED keyword defines the form-feed character, or carriage control character, for the output print (*.PRT) run summary (*.RSM) files. The keyword should be place at the very top of the input file.

See *FORMFEED – Defined the Print File Form-Feed Character* in the GLOBAL section for a dull description.

10.3.40 GASCONC – DEFINE THE INITIAL EQUILIBRATION COAL GAS CONCENTRATION FOR ALL GRID BLOCKS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The GASCONC keyword defines the initial equilibration coal gas concentration values for all matrix grid cells in the model and should be used in conjunction with the GCVD keyword in the SOLUTION section, to fully describe the initial state of the model. The keyword should only be used if the coal phase has been activated in the model via the COAL keyword in the RUNSPEC section. Note both GASCONC and GCVD are optional as the simulator will calculate the coal gas concentration based on the equilibrium concentration and the block pressure.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

This is the non-standard method to initialize the model via enumeration and is seldom employed in the industry. The standard methodology is for OPM Flow to initialize a model using the parameters on the EQUIL keyword combined with other keywords to fully describe the initial state of the model. The keyword can be used with all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	GASCONC	GASCONC is an array of real positive numbers that define the initial equilibration coal gas concentration values to each matrix cell in the model. Repeat counts may be used, for example 20*75.0.			
		Mscf/ft3	sm^3/m^3	scc/cc	None

Notes:

1) The number of entries should correspond to number of matrix cells in the model, that is (NX x NY x NZ) divided by two, as per the parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.

2) The keyword is terminated by a "/".

Table 10.15: GASCONC Keyword Description

See also the GCVD keyword in the SOLUTION section to fully define the initial state of the model.

Example

```
--
--       DEFINE INITIAL EQUILIBRATION COAL GAS CONCENTRATION FOR ALL CELLS
--       BASED ON NX = 100, NY = 100 AND NZ = 6
--
GASCONC
         1000*75.500    1000*65.500    1000*60.000                          /
```

The above example defines the initial equilibration coal gas concentration values to be 75.500 for all the matrix cells in the first layer, 65.500 for all the cells in the second layer, and finally 60.000 for all the cells in the third layer.

10.3.41 GASSATC – DEFINE THE INITIAL EQUILIBRATION SATURATED COAL GAS CONCENTRATION FOR ALL GRID BLOCKS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The GASSATC keyword defines the initial equilibration saturated coal gas concentration values for all grid cells in the model. The keyword should only be used if the coal phase has been activated in the model via the COAL keyword in the RUNSPEC section. The keyword is used to re-scale the Langmuir isotherms entered via the LANGMUIR keyword in the PROPS section, in conjunction with a matrix grid blocks initial reservoir pressure. The keyword is optional, and if absent from the input file, the matrix grid block Langmuir isotherm is left unscaled.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

This is the non-standard method to initialize the model via enumeration and is seldom employed in the industry. The standard methodology is for OPM Flow to initialize a model using the parameters on the EQUIL keyword combined with other keywords to fully describe the initial state of the model. The keyword can be used with all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	GASSATC	GASSATC is an array of real positive numbers that define the initial equilibration saturated coal gas concentration values to each cell in the model. Repeat counts may be used, for example 20*75.0.			
		Mscf/ft3	sm^3/m^3	scc/cc	None

Notes:

1) The number of entries should correspond to number of matrix cells in the model, that is (NX x NY x NZ) divided by two, as per the parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.

2) The keyword is terminated by a "/".

Table 10.16: GASSATC Keyword Description

See also the GASCONC and the GCVD keywords in the SOLUTION section to fully define the initial state of the model.

Example

```
--
--       DEFINE INITIAL EQUILIBRATION SAT COAL GAS CONCENTRATION ALL CELLS MODEL
--       BASED ON NX = 100, NY = 100 AND NZ = 6
--
GASSATC
         1000*75.500    1000*65.500    1000*60.000                              /
```

The above example defines the initial equilibration saturated coal gas concentration values to be 75.500 for all the matrix cells in the first layer, 65.500 for all the cells in the second layer, and finally 60.000 for all the cells in the third layer.

10.3.42 GCVD – DEFINE EQUILIBRATION COAL GAS CONCENTRATION VERSUS DEPTH TABLES

Description

The GCVD keyword defines the initial coal gas concentration versus depth tables for each equilibration region for when the coal phase has been activated in the run via the COAL keyword in the RUNSPEC section. The keyword may be used in conjunction with the GASCONC keyword in the SOLUTION section, to fully describe the initial state of the model. Note both GASCONC and GCVD are optional as the simulator will calculate the coal gas concentration based on the equilibrium concentration and the block pressure.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	DEPTH	A columnar vector of real monotonically increasing down the column values that defines the depth values for the corresponding coal gas concentration, GCVALS.			
		feet	m	cm	None
2	GCVALS	A columnar vector of real values that defines the coal gas concentration values at the corresponding DEPTH.			
		Mscf/ft3	sm^3/m^3	scc/cc	None

Notes:

1) The keyword is followed by NTEQUL records as declared on the EQLDIMS keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NDRXVD rows as declared on the EQLDIMS keyword in the RUNSPEC section.
3) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 10.17: GCVD Keyword Description

See also the GASCONC and GASSATC keywords in the SOLUTION section.

Example

Given NTEQUL equals three and NDRXVD is greater than or equal to two on the EQLDIMS keyword in the RUNSPEC section, then the following example defines the coal gas concentration versus depth functions.

```
--
--        DEPTH     GC
--                  MSCF/FT
--        ------    --------
GCVD
           100.0    75.5000
          1000.0    75.5000                          / GC VS DEPTH EQUIL REGN 01
--        ------    --------
           100.0    65.5000
          1000.0    65.5000                          / GC VS DEPTH EQUIL REGN 02
--        ------    --------
           100.0    60.0000
          1000.0    60.0000                          / GC VS DEPTH EQUIL REGN 03
```

Here three tables are entered with a constant coal gas concentration versus depth relationship for each equilibration region.

10.3.43 GETDATA – LOAD AND ASSIGN DATA ARRAY FROM INIT OR RESTART FILE

The GETDATA keyword loads a data array from a previously generated INIT or RESTART file and assigns the loaded array to either same array in the run or another array name.

See *GETDATA – Load and Assign Data Array from INIT or RESTART Files* in the GRID section for a full description

10.3.44 GETGLOB – ACTIVATE LOADING OF GLOBAL GRID RESTART DATA OPTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, GETGLOB, switches on the global grid read option for when the run is restarting from a RESTART file. Only the global grid will be loaded in the subsequent RESTART keyword and any Local Grid Refinements ("LGR") on the RESTART file will be ignored.

There is no data required for this keyword.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

Example

```
--
--       ACTIVATE LOADING OF GLOBAL GRID RESTART DATA OPTION
--
GETGLOB
```

The above example switches on the option to only load the global grid from the RESTART file.

10.3.45 GI - DEFINE THE INITIAL EQUILIBRATION GI VALUES FOR ALL GRID BLOCKS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The GI keyword defines the initial equilibration GI values for all grid cells in the model and should be used in conjunction with the other enumeration equilibration keywords; PBUB, PDEW, PRESSURE, RS, RV, SOIL and SWAT keywords etc., to fully describe the initial state of the model. The keyword should only be used if the GI Pseudo Compositional option has been activated in the model via the GIMODEL keyword in the RUNSPEC section.

This is the non-standard method to initialize the model via enumeration and is seldom employed in the industry. The standard methodology is for OPM Flow to initialize a model using the parameters on the EQUIL keyword combined with other keywords to fully describe the initial state of the model. The keyword can be used with all grid types.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

See also the GIALL keyword in the PROPS section that sets the GI values as a function of pressure, as well as setting the corresponding RVGI, RSGI, BGGI and BOGI values at the same time.

10.3.46 HMAQUCT – HISTORY MATCH CARTER-TRACY AQUIFER GRADIENT PARAMETERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The HMAQUCT keyword defines the history match analytical Carter-Tracy aquifer gradient parameters for when the History Match Gradient option has been activated by the HMDIMS keyword in the RUNSPEC section, and analytical Carter-Tracy aquifers have been specified in the model via the AQUCT and connected to the grid using the AQUANCON or AQANCONL keywords. All keywords are in the SOLUTION section.

See also the HMDIMS keyword in the RUNSPEC section that specifies the dimensions for the gradient option, including the maximum number of aquifers that can be used with the History Match Gradient option.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

10.3.47 HMAQUFET – HISTORY MATCH FETKOVICH AQUIFER GRADIENT PARAMETERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The HMAQUFET keyword defines the history match analytical Fetkovich aquifer gradient parameters for when the History Match Gradient option has been activated by the HMDIMS keyword in the RUNSPEC section, and analytical Fetkovich aquifers have been specified in the model via the AQUFET and/or the AQUFETP keywords and connected to the grid using AQUANCON or AQANCONL keywords. All keywords are in the SOLUTION section.

See also the HMDIMS keyword in the RUNSPEC section that specifies the dimensions for the gradient option, including the maximum number of aquifers that can be used with the History Match Gradient option.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

10.3.48 HMMLCTAQ – HISTORY MATCH CARTER-TRACY AQUIFER GRADIENT MULTIPLIERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The HMMLCTAQ keyword defines the history match analytical Carter-Tracy aquifer gradient multipliers for when the History Match Gradient option has been activated by the HMDIMS keyword in the RUNSPEC section, and analytical Carter-Tracy aquifers have been specified in the model via the AQUCT and connected to the grid using the AQUANCON or AQANCONL keywords. All keywords are in the SOLUTION section.

Multipliers can be declared for the Carter-Tracy aquifer permeability, aquifer angle of influence and the aquifer depth.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

10.3.49 HMMLFTAQ – HISTORY MATCH FETKOVICH AQUIFER GRADIENT MULTIPLIERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The HMAQUFET keyword defines the history match analytical Fetkovich aquifer gradient multipliers for when the History Match Gradient option has been activated by the HMDIMS keyword in the RUNSPEC section, and analytical Fetkovich aquifers have been specified in the model via the AQUFET and/or the AQUFETP keywords and connected to the grid using AQUANCON or AQANCONL keywords. All keywords are in the SOLUTION section.

Multipliers can be declared for the Fetkovich aquifer water volume, aquifer permeability, and the aquifer depth.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

10.3.50 HMMLTWCN – HISTORY MATCH WELL CONNECTION AND SKIN MULTIPLIERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, HMMLTWCN, defines the history match gradient multipliers for well connection factors and connection skins, for when the History Match Gradient option has been activated by the HMDIMS keyword in the RUNSPEC section. Wells must be specified using the WELPSECS keyword in the SCHEDULE section and their connections defined by the COMPDAT and/or COMPDATL keywords, also in the SCHEDULE section

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

10.3.51 HMWELCON – HISTORY MATCH WELL CONNECTION AND SKIN PARAMETERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, HMWELCON, defines the history match gradient parameters for well connection factors and connection skins, for when the History Match Gradient option has been activated by the HMDIMS keyword in the RUNSPEC section. Wells must be specified using the WELPSECS keyword in the SCHEDULE section and their connections defined by the COMPDAT and/or COMPDATL keywords, also in the SCHEDULE section

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

10.3.52 IMPORT – IMPORT GRID FILE DATA AT THE CURRENT POSITION

The IMPORT keyword informs the simulator to continue reading input data from the specified IMPORT file. When the end of the IMPORT file is reached, input data is read from the next keyword in the current file. Normally IMPORT files are generated by grid pre-processing software and the keyword allows for both formatted and unformatted (binary) files to be loaded.

See *IMPORT – Import Grid File Data at the Current Position* in the GRID section for a full description.

10.3.53 INCLUDE – LOAD ANOTHER DATA FILE AT THE CURRENT POSITION

The INCLUDE keyword informs OPM Flow to continue reading input data from the specified INCLUDE file. When the end of the INCLUDE file is reached, or the ENDINC keyword is encountered in the included file, input data is read from the next keyword in the current file.

See *INCLUDE – Load Another Data File at the Current Position* in the GLOBAL section for a full description.

10.3.54 MESSAGE – OUTPUT USER MESSAGE

The MESSAGE keyword outputs a user message to the terminal, as well as to the print (*.PRT) and debug (*.DBG) files. Note this is different to the MESSAGES keyword, that defines OPM Flows message print limits and stop limits generated by the simulator.

See *MESSAGE – Output User Message* in the GLOBAL section for a full description.

10.3.55 MESSAGES – DEFINE MESSAGE PRINT LIMITS AND STOP LIMITS

The MESSAGES keyword defines the print and stops levels for various messages. The "print limits" set the maximum number of messages that will be printed, after which no more messages will be printed and the "stop limits" terminate the run when these limits are exceeded. There are six levels of message that increase in severity from informative all the way to programming errors, as outlined in Error: Reference source not found.

See *MESSAGES – Define Message Print Limits and Stop Limits* in the GLOBAL section for a full description.

10.3.56 MULTIPLY – MULTIPLY A SPECIFIED ARRAY BY A CONSTANT

The MULTIPLY keyword multiplies a specified array or part of an array by a constant. The constant can be real or integer depending on the array type; however, the arrays that can be operated on is dependent on which section the keyword is being used.

See *MULTIPLY – Multiply a Specified Array by a Constant* in the GRID section for a full description.

10.3.57 MULTIREG – MULTIPLY AN ARRAY BY A CONSTANT BASED ON A REGION NUMBER

The MULTIREG keyword multiplies an array or part of an array by a constant for cells with a specific region number. The region number array can be FLUXNUM, MULTNUM or OPERNUM and these arrays must be defined and be available before the MULTIREG keyword is read by the simulator. The constant can be real or integer depending on the property array type; however, the property arrays that can be operated on is dependent on which section the MULTIREG keyword is being used.

See *MULTIREG – Multiply an Array by a Constant based on a Region Number* in the GRID section for a full description.

10.3.58 NOECHO – DEACTIVATE ECHOING OF USER INPUT FILES TO THE PRINT FILE

Turns off echoing of all the input files to the print file. Note by default echoing of the inputs files is active. but can subsequently be switched off by the NOECHO activation keyword.

See *NOECHO – Deactivate Echoing of User Input Files to the Print File* in the GLOBAL section for a full description.

10.3.59 NOHMD – DEACTIVATE HISTORY MATCH GRADIENT DERIVATIVE CALCULATIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The NOHMD deactivates various history match gradient derivative calculations for when the History Match Gradient option has been activated by the HMDIMS keyword in the RUNSPEC section. The keyword consists of a series of character strings that define which derivative should be switch off based on the keyword that requested the derivatives to be calculated, for example HMFAULTS keyword in the GRID section. If an empty list is entered then all the gradient derivative calculations previously requested are switch off. The keyword is useful for changing from history matching runs to predication cases, as the prediction cases will be more computationally efficient without the burden of the gradient derivative calculations.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

10.3.60 NOHMO – DEACTIVATE HISTORY MATCH GRADIENT DERIVATIVE CALCULATIONS (ALIAS)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The NOHMO deactivates various history match gradient derivative calculations for when the History Match Gradient option has been activated by the HMDIMS keyword in the RUNSPEC section. The keyword consists of a series of character strings that define which derivative should be switch off based on the keyword that requested the derivatives to be calculated, for example HMFAULTS keyword in the GRID section. If an empty list is entered then all the gradient derivative calculations previously requested are switch off. The keyword is useful for changing from history matching runs to predication cases, as the prediction cases will be more computationally efficient without the burden of the gradient derivative calculations.

The keyword is an alias for the NOHMD keyword in the SOLUTION section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

10.3.61 NOWARN – DEACTIVATE WARNING MESSAGES

Turns off warning messages to be printed to the print file; note that this keyword is deactivated by default and can subsequently be switched off by the WARN activation keyword. The warning messages may be turned on and off using keywords WARN and NOWARN.

See *NOWARN – Deactivate Warning Messages* in the GLOBAL section for a full description.

10.3.62 OILAPI – DEFINE THE INITIAL EQUILIBRATION OIL API FOR ALL GRID BLOCKS

Description

The OILAPI keyword defines the initial equilibration oil API gravity pressures for all grid cells in the model, for when the Oil API Tracking option as been invoked by the API keyword in the RUNSPEC section. The keyword should be used in conjunction with the PBUB, PDEW, PRESSURE, RS, RV, SGAS, SOIL and SWAT keywords etc., to fully describe the initial state of the model.

The keyword is used by the Enumeration Initialization method to initialize the model, as opposed to the Equilibration Initialization method that utilizes the EQUIL keyword in the SOLUTION section. This is the non-standard formulation to initialize the model and is seldom employed in the industry. The standard methodology is for OPM Flow to initialize a model using the parameters on the EQUIL keyword combined with other keywords to fully describe the initial state of the model. The keyword can be used with all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	OILAPI	OILAPI is an array of real positive numbers assigning the initial equilibration oil API gravity to each cell in the model. The American Petroleum Institute ("API") classifies oils based on an API gravity (γ_{API}), or degrees API (°API), the relationship between relative density (γ_o) of oil and API gravity (γ_{API}) is given by: $$\gamma_{API} = \frac{141.5}{\gamma_o} - 131.5$$ Repeat counts may be used, for example 20*38.5			
		°API	°API	°API	None

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) The keyword is terminated by a "/".

Table 10.18: OILAPI Keyword Description

See also the PBUB, PDEW, PRESSURE, RS, RV, SGAS, SOIL and SWAT keywords to fully define the initial state of the model.

Example

```
--
--       DEFINE INITIAL EQUILIBRATION OIL API FOR ALL CELLS IN THE MODEL
--       BASED ON NX = 100, NY = 100 AND NZ = 3
--
OILAPI
        1000*40.2     1000*39.5     1000*38.2                                  /
```

The above example defines the initial equilibration oil API gravity to be 40.2 for all the cells in the first layer, 39.5 for all the cells in the second layer, and finally 38.2 for all the cells in the third layer.

10.3.63 OPERATE – DEFINE MATHEMATICAL OPERATIONS ON ARRAYS

This keyword, OPERATE, defines mathematical operations on property arrays (NTG, PORO etc.) and optionally using another property array as input to the function. The keyword allows for various mathematical functions and their associated variables to be defined and applied to the selected array data. Input constants can be real or integer depending on the array type; however, the arrays that can be operated on is dependent on which section the keyword is being used.

See *OPERATE – Define Mathematical Operations on Arrays* in the GRID section for a full description.

10.3.64 OPERATER – DEFINE MATHEMATICAL OPERATIONS ON ARRAYS BY REGION

This keyword defines the mathematical operations on arrays for specific regions in the commercial simulator and is currently not supported by OPM Flow. However, similar functionality is provided by the ADD and MULTIPLY keywords.

See *OPERATER – Define Mathematical Operations on Arrays by Region* in the GRID section for a full description.

10.3.65 OUTSOL – DEFINE DATA TO BE WRITTEN TO THE RESTART FILE (RETIRED)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword defines the data and frequency of the data to be written to the RESTART file at each requested restart point. The keyword has been replaced by the RPTRST keyword in the SOLUTION and SCHEDULE sections and is therefore considered retired.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

10.3.66 PBUB – DEFINE THE INITIAL EQUILIBRATION BUBBLE-POINT PRESSURE FOR ALL GRID BLOCKS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The PBUB keyword defines the initial equilibration buble-point saturation pressures values for all grid cells in the model and should be used in conjunction with the PDEW, PRESSURE, RS, RV, SGAS, SOIL and SWAT keywords etc., to fully describe the initial state of the model. The keyword should only be used if dissolved gas has been activated in the model via the DISGAS keyword in the RUNSPEC section.

This is the non-standard method to initialize the model via enumeration and is seldom employed in the industry. The standard methodology is for OPM Flow to initialize a model using the parameters on the EQUIL keyword combined with other keywords to fully describe the initial state of the model. The keyword can be used with all grid types.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	PBUB	PBUB is an array of real positive numbers assigning the initial equilibration bubble-point saturation pressure values to each cell in the model. Repeat counts may be used, for example 20*3500.0			
		psia	barsa	atma	None

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.

2) The keyword is terminated by a "/".

Table 10.19: PBUB Keyword Description

See also the PBVD, PDEW, PRESSURE, RV, SGAS, SOIL and SWAT keywords to fully define the initial state of the model.

Example

```
--
--       DEFINE INITIAL EQUILIBRATION PSAT VALUES FOR ALL CELLS IN THE MODEL
--       BASED ON NX = 100, NY = 100 AND NZ = 3
--
PBUB
         1000*3500.0    1000*3525.0    1000*0.3535.0                          /
```

The above example defines the initial equilibration bubble-point saturation pressure values to be 3500.0 for all the cells in the first layer, 3525.0 for all the cells in the second layer, and finally 3535.0 for all the cells in the third layer.

10.3.67 PBVD – EQUILIBRATION BUBBLE-POINT VERSUS DEPTH TABLES

Description

The PBVD keyword defines the bubble-point pressure versus depth tables for each equilibration region that should be used when there is dissolved gas in the model (DISGAS has been activated in the RUNSPEC section) and the EQLOPT1 variable has been set to a positive integer on the EQUIL keyword in the SOLUTION section.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	DEPTH	A columnar vector of real monotonically increasing down the column values that defines the depth values for the corresponding bubble-point values, PBVALS.			
		feet	m	cm	None
2	PBVALS	A columnar vector of real values that defines the oil bubble-point values at the corresponding DEPTH.			
		psia	barsa	atma	None

Notes:

1) The keyword is followed by NTEQUL records as declared on the EQLDIMS keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NDRXVD rows as declared on the EQLDIMS keyword in the RUNSPEC section.
3) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 10.20: PBVD Keyword Description

Alternatively, the dissolved gas-oil ratio versus depth tables may be entered using the RSVD keyword in the SOLUTION section instead of this keyword. See also the RSVD and EQUIL keywords in the SOLUTION section.

Example

Given NTEQUL equals three and NDRXVD is greater than or equal to two on the EQLDIMS keyword in the RUNSPEC section, then the following example defines the bubble-point versus depth functions.

```
--
--        DEPTH     PSAT
--                  PRESS
--        ------    ------
PBVD
          3000.0    3000.0
          8000.0    3025.0                          / PSAT VS DEPTH EQUIL REGN 01
--        ------    ------
          3000.0    3100.0
          8000.0    3125.0                          / PSAT VS DEPTH EQUIL REGN 02
--        ------    ------
          3000.0    3200.0
          8000.0    3225.0                          / PSAT VS DEPTH EQUIL REGN 03
```

Here three tables are entered and each table is terminated by a "/" and there is no keyword terminating "/".

10.3.68 PDEW – DEFINE THE INITIAL EQUILIBRATION DEW-POINT PRESSURE FOR ALL GRID BLOCKS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The PDEW keyword defines the initial equilibration dew-point pressure values for all grid cells in the model and should be used in conjunction with the PBUB, PRESSURE, RS, RV, SGAS, SOIL and SWAT keywords etc., to fully describe the initial state of the model. The keyword should only be used if vaporized oil been activated in the model via the VAPOIL keyword in the RUNSPEC section.

This is the non-standard method to initialize the model via enumeration and is seldom employed in the industry. The standard methodology is for OPM Flow to initialize a model using the parameters on the EQUIL keyword combined with other keywords to fully describe the initial state of the model. The keyword can be used with all grid types.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	PDEW	PDEW is an array of real positive numbers assigning the initial equilibration dew-point pressure values to each cell in the model. Repeat counts may be used, for example 20*3525.0			
		psia	barsa	atma	None

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.

2) The keyword is terminated by a "/".

Table 10.21: PDEW Keyword Description

See also the PBUB, PRESSURE, RS, RV, SGAS, SOIL and SWAT keywords to fully define the initial state of the model.

Example

```
--
--       DEFINE INITIAL EQUILIBRATION PSAT VALUES FOR ALL CELLS IN THE MODEL
--       BASED ON NX = 100, NY = 100 AND NZ = 3
--
PDEW
         1000*3500.0    1000*3525.0    1000*0.3535.0                          /
```

The above example defines the initial equilibration dew-point saturation pressure values to be 3500.0 for all the cells in the first layer, 3525.0 for all the cells in the second layer, and finally 3535.0 for all the cells in the third layer.

10.3.69 PDVD – DEFINE EQUILIBRATION DEW-POINT VERSUS DEPTH TABLES

Description

The PDVD keyword defines the dew-point pressure versus depth tables for each equilibration region that should be used when there is vaporized oil in the model (VAPOIL has been activated in the RUNSPEC section) and the EQLOPT2 variable has been set to a positive integer on the EQUIL keyword in the SOLUTION section.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	DEPTH	A columnar vector of real monotonically increasing down the column values that defines the depth values for the corresponding dew-point values, PDVALS.			
		feet	m	cm	None
2	PDVALS	A columnar vector of real values that defines the gas dew-point values at the corresponding DEPTH.			
		psia	barsa	atma	None

Notes:

1) The keyword is followed by NTEQUL records as declared on the EQLDIMS keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NDRXVD rows as declared on the EQLDIMS keyword in the RUNSPEC section.
3) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 10.22: PDVD Keyword Description

Alternatively, the vaporized oil-gas ratio (condensate-gas ratio) versus depth tables may be entered using the RVVD keyword in the SOLUTION section instead of this keyword.

See also the RVVD and EQUIL keywords in the SOLUTION section.

Example

Given NTEQUL equals three and NDRXVD is greater than or equal to two on the EQLDIMS keyword in the RUNSPEC section, then the following example defines the bubble-point versus depth functions.

```
--
--        DEPTH     PSAT
--                  PRESS
--        ------    ------
PDVD
          3000.0    2000.0
          8000.0    2025.0                          / PSAT VS DEPTH EQUIL REGN 01
--        ------    ------
          3000.0    2100.0
          8000.0    2125.0                          / PSAT VS DEPTH EQUIL REGN 02
--        ------    ------
          3000.0    2200.0
          8000.0    2225.0                          / PSAT VS DEPTH EQUIL REGN 03
```

Here three tables are entered and each table is terminated by a "/" and there is no keyword terminating "/".

10.3.70 PRESSURE – DEFINE THE INITIAL EQUILIBRATION PRESSURES FOR ALL GRID BLOCKS

Description

The PRESSURE keyword defines the initial equilibration pressures for all grid cells in the model and should be used in conjunction with the PBUB, PDEW, RS, RV, SGAS, SOIL and SWAT keywords etc., to fully describe the initial state of the model.

The keyword is used by the Enumeration Initialization method to initialize the model, as opposed to the Equilibration Initialization method that utilizes the EQUIL keyword in the SOLUTION section. This is the non-standard formulation to initialize the model and is seldom employed in the industry. The standard methodology is for OPM Flow to initialize a model using the parameters on the EQUIL keyword combined with other keywords to fully describe the initial state of the model. The keyword can be used with all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	PRESSURE	PRESSURE is an array of real positive numbers assigning the initial equilibration pressures to each cell in the model. Repeat counts may be used, for example 20*4200.0.			
		psia	barsa	atma	None

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.

2) The keyword is terminated by a "/".

Table 10.23: PRESSURE Keyword Description

See also the PBUB, PDEW, RS, RV, SGAS, SOIL and SWAT keywords to fully define the initial state of the model.

Example

```
--
--       DEFINE INITIAL EQUILIBRATION PRESSURES FOR ALL CELLS IN THE MODEL
--       BASED ON NX = 100, NY = 100 AND NZ = 3
--
PRESSURE
         1000*4500.0   1000*4510.0   1000*4520.0                              /
```

The above example defines the initial equilibration pressures to be 4500.0 for all the cells in the first layer, 4510.0 for all the cells in the second layer, and finally 4520.0 for all the cells in the third layer.

10.3.71 PRVD – DEFINE THE INITIAL EQUILIBRATION PRESSURES VERSUS DEPTH

Description

The PRVD keyword defines the initial reservoir pressure versus depth and should be used in conjunction with the PBUB, PDEW, RS, RV, SGAS, SOIL and SWAT keywords etc., to fully describe the initial state of the model. PRVD is an alternative to the PRESSURE keyword in the SOLUTION section, that defines the initial equilibration pressures for all grid cells in the model

The keyword is used by the Enumeration Initialization method to initialize the model, as opposed to the Equilibration Initialization method that utilizes the EQUIL keyword in the SOLUTION section. This is the non-standard formulation to initialize the model and is seldom employed in the industry. The standard methodology is for OPM Flow to initialize a model using the parameters on the EQUIL keyword combined with other keywords to fully describe the initial state of the model. The keyword can be used with all grid types.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	DEPTH	A columnar vector of real monotonically increasing down the column values that defines the depth values for the corresponding reservoir oil pressures values, PRESSURE.			
		feet	m	cm	None
2	PRESSURE	A columnar vector of real values that defines the initial equilibration oil pressure values at the corresponding DEPTH.			
		psia	barsa	atma	None

Notes:

1) The keyword is followed by NTEQUL records as declared on the EQLDIMS keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NDRXVD rows as declared on the EQLDIMS keyword in the RUNSPEC section.
3) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 10.24: PRVD Keyword Description

See also the PBUB, PDEW, RS, RV, SGAS, SOIL and SWAT keywords to fully define the initial state of the model.

Example

Given NTEQUL equals three and NDRXVD is greater than or equal to five on the EQLDIMS keyword in the RUNSPEC section, then the following example defines the initial oil reservoir pressure versus depth

```
--
--        DEPTH     INIT
--                  PRESS
--        ------    ------
PRVD
          3000.0    3000.0
          4000.0    3345.0
          5000.0    3690.0
          7000.0    4700.0
          7200.0    4769.0                          / POIL VS DEPTH EQUIL REGN 01
--        ------    ------
          3000.0    3100.0
          4000.0    3445.0
          5000.0    3790.0
          7000.0    4700.0
          7200.0    4769.0                          / POIL VS DEPTH EQUIL REGN 02
--        ------    ------
          3000.0    3150.0
          4000.0    3495.0
          5000.0    3840.0
          7000.0    4700.0
          7200.0    4769.0                          / POIL VS DEPTH EQUIL REGN 03
```

Here three tables are entered and each table is terminated by a "/" and there is no keyword terminating "/".

10.3.72 PYEND – END THE DEFINITION OF A PYINPUT SECTION

The PYINPUT and PYEND keywords are a part of OPM Flow's Python scripting facility that processes standard Python commands that can be used to manipulate and define the simulators input parameters during processing of the input deck. The main purpose of the facility is to script the construction of the various keywords. A PYINPUT Definition Section is terminated by a PYEND keyword on a separate single line.

See *PYEND – End the Definition of a PYINPUT Section* in the GRID section for a full description.

10.3.73 PYINPUT – DEFINE THE START OF A PYINPUT SECTION

The PYINPUT and PYEND keywords are a part of OPM Flow's Python scripting facility that processes standard Python commands that can be used to manipulate and define the simulators input parameters during processing of the input deck. The main purpose of the facility is to script the construction of the various keywords used by the simulator. PYINPUT declares the start of a PYINPUT Definition Section on a single separate line, which is then followed by various standard Python commands, with one command per line.

See *PYINPUT – Define the Start of a PYINPUT Section* in the GRID section for a full description.

10.3.74 RAINFALL – CONSTANT FLUX AQUIFER RAINFALL FLUX BY MONTH

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This RAINFALL keyword defines the month by month rainfall flux for constant flux aquifers.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

10.3.75 RBEDCONT – DEFINE RIVER GRID BLOCK CONTACT AREA VERSUS DEPTH

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The RBEDCONT keyword defines the river grid block contact area versus depth tables, for when the River option has been activated via the RIVRDIMS keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

10.3.76 REFINE – START THE DEFINITION OF A LOCAL GRID REFINEMENT

The REFINE keyword defines the start of a Cartesian or radial local grid refinement ("LGR") definition that sets the properties of the selected LGR. The keyword is then followed by the property keywords associated with the section where the keyword is being invoked. For example, if the REFINE keyword is used in the GRID section then most of the keywords in that section can be used to set the grid properties for the LGR.

See *REFINE – Start the Definition of a Local Grid Refinement* in the GRID section for a full description.

10.3.77 RESTART – RESTART RUN FROM AN EXISTING RESTART FILE

Description

The RESTART keyword defines the parameters to restart the simulation from a previous run that has written a RESTART file out to disk. Only restarting from RESTART files is permitted by OPM Flow; restarting from SAVE files is not implemented.

No.	Name	Description	Default
1	RSNAME	The RSNAME variable is a character string that defines the root name of the RESTART file to be read into the current input deck.	None
2	RSNUM	A positive integer that defines the restart point on the RESTART file to be read and to be used to initialize the model. When OPM Flow writes a restart point a message is printed to the *.PRT file indicating the time step the restart was written out.	None
3	RSTYPE	A defined character sting set to SAVE to read the restart data from the SAVE file, otherwise defaulted to 1* to read the data from the RESTART file. The SAVE file option is not supported by OPM Flow and should be defaulted with 1*.	1*
4	RSFORMAT	A defined character string that defines the format of the SAVE file to be read if RSTYPE has been set to SAVE, and should be set to one of the following: 1) FORMATTED: If the file is formatted as ASCII i.e. a text file, as oppose to a binary file. The option can be abbreviated to just the letter F. 2) UNFORMATTED: If the file is in binary format, note this option can be abbreviated to just the letter U. This type of file is operating system dependent If the variable RSFORMAT omitted then the default is for binary file input. This option is not supported by OPM Flow and should be defaulted with 1*.	U

Notes:

1) OPM Flow can only restart runs from a RESTART file, the commercial simulator's SAVE file format is not supported.
2) Note that due to the complexities of the RESTART file, OPM Flow may not always be able to restart from the commercial simulators RESTART file.
3) The keyword is terminated by a "/".

Table 10.25: RESTART Keyword Description

The most direct way to start a restart run is to:

1) Copy the existing data file that created the RESTART file and give it a new name. For example if the RESTART file is from a case named NOR-OPM-A01.DATA, then the copied data file could be named NOR-OPM-A01-R1.DATA.
2) Edit the copied data file (NOR-OPM-A01-R1.DATA) and delete all equilibration keywords (EQUIL, RSVD, etc.) or the enumeration equilibration keywords (PRESSURE, SGAS, SOIL. SWAT, etc.) in the SOLUTION section used to initialize the model.

3) In the SOLUTION section of NOR-OPM-A01-R1.DATA file insert the RESTART keyword, using NOR-OPM-A01 as RSNAME and the required RSNUM value for the time step to restart from.

4) In the SCHEDULE section of NOR-OPM-A01-R1.DATA file insert the SKIPREST keyword at the very beginning of the SCHEDULE section. The SKIPREST keyword causes the simulator to only read in data it requires for restarting the run up to the RESTART point (RSNUM). Note that certain keywords always need to be present in a restart run in the SCHEDULE section as the data is not stored on the RESTART file, for example the VFP tables (VFPPROD and VFPINJ keywords). The SKIPREST keyword automatically processes the input deck and reads the required data.

5) In the SCHEDULE section of NOR-OPM-A01-R1.DATA file after the RESTART point make any required changes, save the file and run the NOR-OPM-A01-R1.DATA with OPM Flow.

See also RPTRST, RPTSCHED and SKIPREST keywords.

Example

The example below defines a restart from the previously run NOR-OPM-A01 case at time step number 40.

```
-- ================================================================================
--
-- SOLUTION SECTION
--
-- ================================================================================
SOLUTION
--
--       FLEXIBLE RESTART FROM PREVIOUS SIMULATION RUN
--
--       FILE                   RESTART   RESTART   FILE
--       NAME                   NUMBER    TYPE      FORMAT
RESTART
         'NOR-OPM-A01'          40        1*        1*     /
```

In addition in the SCHEDULE section the SKIPREST keyword should be used to correctly read in the schedule data up to the RESTART point.

```
-- ================================================================================
--
-- SCHEDULE SECTION
--
-- ================================================================================
SCHEDULE
--
--       ACTIVATE SKIPREST OPTION TO AVOID MODIFYING SCHEDULE SECTION
--
SKIPREST
```

Note is advisable to place the SKIPREST keyword at the very beginning of the SCHEDULE section.

10.3.78 RIVERSYS - DEFINE RIVER SYSTEM (BRANCH STRUCTURE AND BOUNDARY CONDITIONS)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

RIVERSYS defines a river system by specifying the branch structure of the river together with the branch's associated boundary conditions, for when the River option has been activated via the RIVRDIMS keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

10.3.79 RPTRST – DEFINE DATA TO BE WRITTEN TO THE RESTART FILE

Description

This keyword defines the data and frequency of the data to be written to the RESTART file at each requested restart point. In addition to the solution data arrays required to restart a run and the frequency of the data to be written, the user may request additional data to be written to the restart file for visualization in OPM ResInsight.

The format consists of the keyword followed by a series of character strings that indicate the data to be printed. In most cases the character string is the keyword used to load the data in the OPM Flow input deck, for example KRG for the gas relative permeability of each grid block at the requested times. Its is anticipated that OPM Flow will support additional functionality as development progresses.

No.	Name	Description	Array Name
1	ALLPROPS	An alias for DEN, KRG, KRO, KRW, and VISC restart variable names combined that writes all the properties associated with these keywords.	
2	BASIC	BASIC defines the frequency at which the restart data for restarting a run and the additional requested data is written to the RESTART file. The parameter is assigned a value, OPTION, using the form BASIC = OPTION, where OPTION is an integer variable set to: 1) OPTION = 0 then output to the restart file is stopped. 2) OPTION = 1 then the restart files are written at every report time, but only the last one in the run is kept. This minimizes the restart file size but only the final results are stored, limiting the visualization in OPM ResInsight. 3) OPTION = 2 then the restart files are written at every report time step until this switch is reset and all the restarts are kept. 4) OPTION = 3 then the restart files are written every nth report time step with the frequency determined by the mnemonic "FREQ=n". This feature is not currently supported by OPM Flow. 5) OPTION = 4 then the restart files are written at the first report step of each year. 6) OPTION = 5 then the restart files are written at the first report step of each month. 7) OPTION = 6 then the restart files are written at every time step. In addition for OPTION equal to 3, 4, and 5, the data may be written every n^{th} report time with the frequency determined by the mnemonic "FREQ=n". However, this feature is currently not currently supported in OPM Flow. Note also that OPTION equal to 6 is not supported and will cause the simulator to stop.	
3	DEN	Oil, gas and water fluid phases in-situ densities.	OIL_DEN GAS_DEN WAT_DEN

No.	Name	Description	Array Name
4	FLOWS	Inter-block volumetric flows at surface conditions for each phase (OIL, GAS, WAT). The Array Name column shows the arrays for just the GAS phase, the other phase will have similar arrays.	FLOGASI+ FLOGASJ+ FLOGAS+
5	FLORES	Inter-block volumetric flows at reservoir conditions for each phase (OIL, GAS, WAT). Again, the Array Name column shows the arrays for just the GAS phase, the other phase will have similar arrays.	FLRGASI+ FLRGASJ+ FLRGAS+ FLRGASIJK
6	KRG	Gas relative permeability at the grid blocks gas saturation.	GASKR
7	KRO	Oil relative permeability at the grid blocks oil saturation.	OILKR
8	KRW	Water relative permeability at the grid blocks water saturation.	WATKR
9	RSSAT	Saturated dissolved gas-oil ratio for each grid block to enable restarts.	RSSAT
10	RVSAT	Saturated vaporized oil-gas ratio for each grid block to enable restarts.	RVSAT
11	VISC	Oil, gas and water fluid phases in-situ grid block viscosity data.	OIL_VISC GAS_VISC WAT_VISC

Notes:

1) The keyword is terminated by a "/".

Table 10.26: RPTRST Keyword Description

Note, older versions of the commercial simulator used integer values to control the output to the RESTART file, this type of format is not supported by OPM Flow and thus the they should be converted to the mnemonic format as outlined in Table 10.26.

Note that OPM Flow automatically writes out all the data required to make a restart run as outlined in the table below:

No.	Restart Variable Name	Variable Description	Variable Array Name
1	KRG	Gas relative permeability at the grid blocks gas saturation.	GASKR
2	KRNSW_GO	Gas-oil relative permeability scaling factor array used when the hysteresis option has been activated by the HYSTER variable on the SATOPTS keyword in the RUNSPEC section, to enable restarts.	KRNSW-_GO
3	KRNSW_OW	Oil-water relative permeability scaling factor array used when the hysteresis option has been activated by the HYSTER variable on the SATOPTS keyword in the RUNSPEC section, to enable restarts.	KRNSW_OW
4	KRO	Oil relative permeability at the grid blocks oil saturation.	OILKR

No.	Restart Variable Name	Variable Description	Variable Array Name
5	PCSWM_GO	Gas-oil capillary pressure scaling factor array used when the hysteresis option has been activated by the HYSTER variable on the SATOPTS keyword in the RUNSPEC section, to enable restarts.	PCSWM_GO
6	PCSWM_OW	Oil-Water capillary pressure scaling factor array used when the hysteresis option has been activated by the HYSTER variable on the SATOPTS keyword in the RUNSPEC section, to enable restarts.	PCSWM_OW
7	POLY	Polymer concentration for each grid block to enable restarts for when the POLYMER option has been activated in the RUNSPEC section	CPOLYMER
8	PRESS	Pressure data for each grid block to enable restarts.	PRESSURE
9	RS	Dissolved gas-oil ratio for each grid block to enable restarts.	RS
10	RV	Vaporized oil-gas ratio for each grid block to enable restarts.	RVS
11	SGAS	Gas saturation for each grid block to enable restarts.	SGAS
12	SOIL	Oil saturation each grid block to enable restarts.	SOIL
12	SOMAX	Maximum oil saturation used in determining the rate of oil vaporization in the presence of undersaturated gas and the rate at which the remaining oil gets "heavier" via the reduction in the solution gas-oil ratio ("Rs").	SOMAX
14	SSOL	Solvent saturation for each grid block to enable restarts for when the SOLVENT option has been activated in the RUNSPEC section.	SSOL
15	SWAT	Water ratio for each grid block to enable restarts.	SWAT
16	TEMP	Temperature of each grid block, to enable restarts for when OPM Flow's THERMAL option has been activated in the RUNSPEC section.	TEMP
17	TERNARY	Three phase ternary saturation data is data array is calculated by OPM ResInsight when the RESTART file is loaded into OPM ResInsight.	TERNARY (Calculated)
18	KRW	Water relative permeability at the grid blocks water saturation.	WATKR

Notes:

1) Only items (1) to (14) that are necessary to restart a run are written to the restart file, for example if the neither the POLYMER and SOLVENT options have not been invoked in the RUNSPEC section then the CPOLYMER ans SSOL arrays will not be written to the restart file.

Table 10.27: Data Sets Automatically Written to the RESTART File

Note

Currently, OPM Flow distinguishes between those arrays which are required for restarting a run and those which are "merely" for visualization and analysis. However, this classification is imperfect and leads to potentially exporting arrays that are incompatible with the commercial simulator, such as the TEMP array in non-thermal simulation runs.

If one is willing to forego the ability to restart an OPM Flow simulation run, using the commercial simulator, e.g., simulating the historic period using OPM Flow and the prediction period using the commercial simulator, then additional arrays are available, including the PBUB and PDEW vectors generated by the PBPD option, by using the following command line option:

--enable-opm-rst-file=true

Examples

The first example request that the standard restart data be written out every month.

```
--
--        RESTART CONTROL BASIC = 4 (YEARLY) 5 (MONTHLY)
--
RPTRST
          BASIC=5                                                              /
```

The next example requests that the standard restart data be written at every report time step until this switch is reset and all the restarts are kept. In addition to the standard the data the gas, oil and water relative permeability data will also be written out at each report time step.

```
--
--        RESTART CONTROL BASIC = 4 (YEARLY) 5 (MONTHLY)
--
RPTRST
          BASIC=2  KRG   KRO   KRW                                             /
```

10.3.80 RPTSOL – DEFINE SOLUTION SECTION REPORTING

Description

This keyword defines the data in the SOLUTION section that is to be printed to the output print file in human readable format. The keyword has two distinct forms, the first of which consists of the keyword followed by a series of integers on the next line indicating the data to be printed (see the first example). This is the original formal in the commercial simulator and was subsequently superseded by the second format. The second format consists of the keyword followed by a series of character strings that indicate the data to be printed. In most cases the character string is the keyword used to load the data in the OPM Flow input deck, for example PVDG for the dry gas PVT tables. Its is anticipated that OPM Flow will eventually support the functionality of the second format only, the first format although recognized will be completely ignored.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

No.	Name	Description	Default
1	DENO	Print the oil reservoir density array	N/A
2	EQUIL	Print the equilibration report.	N/A
3	FIP	Print the fluid in-place report.	N/A
4	FIPRESV	Print the reservoir volumes in-place report.	N/A
….		….	N/A

Notes:

1) The keyword is terminated by a "/".

Table 10.28: RPTSOL Keyword Description

Note

Except for non-array like data, FIP etc., this keyword has the potential to produce very large print files that some text editors may have difficulty loading. A more efficient solution for array type data is to load the *.INIT and *.RESTART files into OPM ResInsight to view the data graphically, this also has the benefit of being able to filter the grid based on I, J, K ranges and grid properties.

Examples

The first example shows the original format of this keyword; although the keyword and format are recognized by OPM Flow, the format is ignored and is unlikely to be implemented in in the simulator.

```
--
--       DEFINE SOLUTION SECTION REPORT OPTION (ORIGINAL FORMAT)
-
RPTSOL
         1        2*0      1        3*1                                          /
```

The next example shows the second format of the keyword which may be supported in a future release of OPM Flow.

```
--
--       DEFINE SOLUTION SECTION REPORT OPTIONS
--
RPTSOL
         FIP=2    FIPRESV  RESTART=3                                            /
```

10.3.81 RS – DEFINE THE INITIAL EQUILIBRATION GOR (RS) FOR ALL GRID BLOCKS

Description

The RS keyword defines the initial equilibration gas-oil ratio values for all grid cells in the model and should be used in conjunction with the PBUB, PDEW, PRESSURE, RV, SGAS, SOIL and SWAT keywords etc., to fully describe the initial state of the model. The keyword should only be used if dissolved gas has been activated in the model via the DISGAS keyword in the RUNSPEC section.

This is the non-standard method to initialize the model via enumeration and is seldom employed in the industry. The standard methodology is for OPM Flow to initialize a model using the parameters on the EQUIL keyword combined with other keywords to fully describe the initial state of the model. The keyword can be used with all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	RS	RS is an array of real positive numbers assigning the initial equilibration gas-oil ratio values to each cell in the model. Repeat counts may be used, for example 20*1.30.			
		Mscf/stb	sm^3/sm^3	scc/scc	None

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.

2) The keyword is terminated by a "/".

Table 10.29: RS Keyword Description

See also the PBUB, PDEW, PRESSURE, RV, SGAS, SOIL and SWAT keywords to fully define the initial state of the model.

Example

```
--
--       DEFINE INITIAL EQUILIBRATION GOR VALUES FOR ALL CELLS IN THE MODEL
--       BASED ON NX = 100, NY = 100 AND NZ = 3
--
RS
         1000*1.3500   1000*1.3010   1000*1.3000                              /
```

The above example defines the initial equilibration GOR values to be 1.3500 for all the cells in the first layer, 1.3010 for all the cells in the second layer, and finally 1.3000 for all the cells in the third layer.

10.3.82 RSVD – EQUILIBRATION DISSOLVED GAS-OIL RATIO (RS) VERSUS DEPTH TABLES

Description

The RSVD keyword defines the dissolved gas-oil ratio (Rs) versus depth tables for each equilibration region that should be used when there is dissolved gas in the model (DISGAS has been activated in the RUNSPEC section) and the EQLOPT1 variable has been set to a positive integer on the EQUIL keyword in the SOLUTION section.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	DEPTH	A columnar vector of real monotonically increasing down the column values that defines the depth values for the corresponding dissolve gas-oil ratio values, RS.			
		feet	m	cm	None
2	RS	A columnar vector of real values that defines the dissolved gas-oil ratio values at the corresponding DEPTH.			
		Mscf/stb	sm^3/sm^3	scc/scc	None

Notes:

1) The keyword is followed by NTEQUL records as declared on the EQLDIMS keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NDRXVD rows as declared on the EQLDIMS keyword in the RUNSPEC section.
3) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 10.30: RSVD Keyword Description

Alternatively, the oil bubble-point pressure versus depth tables may be entered using the PBVD keyword in the SOLUTION section instead of this keyword.

See also the PBVD and EQUIL keywords in the SOLUTION section.

Example

Given NTEQUL equals three and NDRXVD is greater than or equal to two on the EQLDIMS keyword in the RUNSPEC section, then the following example defines the bubble-point versus depth functions.

```
--
--        DEPTH     RS
--                  MSCF/STB
--        ------    --------
RSVD
          3000.0     1.400
          8000.0     1.400                              / RS VS DEPTH EQUIL REGN 01
--        ------    --------
          3000.0     1.400
          8000.0     1.400                              / RS VS DEPTH EQUIL REGN 02
--        ------    --------
          3000.0     1.400
          8000.0     1.400                              / RS VS DEPTH EQUIL REGN 03
```

Here three tables are entered with a constant GOR versus depth relationship.

10.3.83 RTEMP - DEFINE THE INITIAL RESERVOIR TEMPERATURE FOR THE MODEL

This keyword defines the initial reservoir temperature for the model. Note that the RTEMP keyword is an alias for RTEMPA, and that both keywords are supported by OPM Flow, in both the PROPS and SOLUTION sections, but are treated as being mutually exclusive.

See *RTEMP - Define the Initial Reservoir Temperature for the Model* in the PROPS section for a full description.

10.3.84 RTEMPA - DEFINE THE INITIAL RESERVOIR TEMPERATURE FOR THE MODEL

This keyword defines the initial reservoir temperature for the model. Note that the RTEMPA keyword is an alias for RTEMP, and that both keywords are supported by OPM Flow, in both the PROPS and SOLUTION sections, but are treated as being mutually exclusive.

See *RTEMPA - Define the Initial Reservoir Temperature for the Model* in the PROPS section for a full description.

10.3.85 RTEMPVD - DEFINE THE INITIAL RESERVOIR TEMPERATURE VERSUS DEPTH TABLES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword defines the initial reservoir temperature versus depth tables for each equilibration region. Note that the RTEMPVD keyword is an alias for TEMPVD, and that both keywords are supported by OPM Flow, in both the PROPS and SOLUTION sections, but are treated as being mutually exclusive.

The initial reservoir temperature must be defined when OPM Flow's thermal option has been activated by the THERMAL keyword in the RUNSPEC section. Note this is different to the commercial simulator that uses the TEMP keyword in the RUNSPEC section to activate the black-oil temperature model, and the THERMAL keyword to activate the compositional thermal model.

The initial reservoir temperature should be defined when OPM Flow's CO_2 or H_2 storage option has been activated by the CO2STORE or H2STORE keyword in the RUNSPEC section.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	DEPTH	A columnar vector of real monotonically increasing down the column values that defines the depth for corresponding reservoir temperature parameter TEMP.			
		feet	m	cm	None
2	TEMP	A columnar vector of real monotonically increasing down the column values that defines the corresponding reservoir temperature for the given depth.			
		°F	°C	°C	None

Notes:

1) The keyword is followed by NTEQUIL tables as declared on the EQLDIMS keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NDRXVD rows as declared on the EQIDIMS keyword in the RUNSPEC section.
3) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 10.31: RTEMPVD Keyword Description

See also the RTEMP keyword in the PROPS section for an alternative way to define a uniform initial reservoir temperature.

Note

The keyword is documented here in the SOLUTION section, the same as the commercial simulator, but it can also be used in the PROPS section by OPM Flow.

Example

```
--
--       INITIAL RESERVOIR TEMPERATURE VERSUS DEPTH TABLE
--
RTEMPVD
--       DEPTH    TEMPERATURE
--       FEET     DEG F
--       ------   ----------
         1000.0     90.000
         2000.0    100.000
         3000.0    130.000
         4000.0    160.000                                  / TABLE N0. 01
--       ------   ----------
         1000.0     90.000
         2000.0    100.000
         3000.0    130.000
         4000.0    160.000                                  / TABLE N0. 02
--       ------   ----------
         1000.0     90.000
         2000.0    100.000
         3000.0    130.000
         4000.0    160.000                                  / TABLE N0. 03
```

The above example defines three identical reservoir depth versus temperature tables for the three NTEQUIL regions defined on the EQLDIMS keyword in the RUNSPEC section.

10.3.86 RV – DEFINE THE INITIAL EQUILIBRATION CGR (RV) FOR ALL GRID BLOCKS

Description

The RV keyword defines the initial equilibration vaporized oil-gas ratio values for all grid cells in the model and should be used in conjunction with the PBUB, PDEW, PRESSURE, RS, SGAS, SOIL and SWAT keywords etc., to fully describe the initial state of the model. The keyword should only be used if vaporized oil been activated in the model via the VAPOIL keyword in the RUNSPEC section.

This is the non-standard method to initialize the model via enumeration and is seldom employed in the industry. The standard methodology is for OPM Flow to initialize a model using the parameters on the EQUIL keyword combined with other keywords to fully describe the initial state of the model. The keyword can be used with all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	RV	RV is an array of real positive numbers assigning the initial equilibration vaporized oil-gas ratio values to each cell in the model. Repeat counts may be used, for example 20*0.00720			
		stb/Mscf	sm^3/sm^3	scc/scc	None

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.

2) The keyword is terminated by a "/".

Table 10.32: RV Keyword Description

See also the PBUB, PDEW, PRESSURE, RS, SGAS, SOIL and SWAT keywords to fully define the initial state of the model.

Example

```
--
--       DEFINE INITIAL EQUILIBRATION CGR VALUES FOR ALL CELLS IN THE MODEL
--       BASED ON NX = 100, NY = 100 AND NZ = 3
--
RV
         1000*0.00720   1000*0.00725   1000*0.00730                          /
```

The above example defines the initial equilibration GOR values to be 0.00720 for all the cells in the first layer, 0.00725 for all the cells in the second layer, and finally 0.00730 for all the cells in the third layer.

10.3.87 RVVD – EQUILIBRATION VAPORIZED OIL-GAS RATIO (RV) VERSUS DEPTH TABLES

Description

The RVVD keyword defines the vaporized oil-gas ratio (Rv) versus depth tables for each equilibration region that should be used when there is vaporized oil in the model (VAPOIL has been activated in the RUNSPEC section) and the EQLOPT2 variable has been set to a positive integer on the EQUIL keyword in the SOLUTION section.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	DEPTH	A columnar vector of real monotonically increasing down the column values that defines the depth values for the corresponding vaporized oil-gas ratio values, RV.			
		feet	m	cm	None
2	RV	A columnar vector of real values that defines the vaporized oil-gas ratio values at the corresponding DEPTH.			
		stb/Mscf	sm^3/sm^3	scc/scc	None

Notes:

1) The keyword is followed by NTEQUL records as declared on the EQLDIMS keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NDRXVD rows as declared on the EQLDIMS keyword in the RUNSPEC section.
3) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 10.33: RVVD Keyword Description

Alternatively, the gas dew-point pressure versus depth tables may be entered using the PDVD keyword in the SOLUTION section instead of this keyword.

See also the PDVD and EQUIL keywords in the SOLUTION section.

Example

Given NTEQUL equals three and NDRXVD is greater than or equal to two on the EQLDIMS keyword in the RUNSPEC section, then the following example defines the dew-point versus depth functions.

```
--
--        DEPTH     RV
--                  STB/MSCF
--        ------    --------
RVVD
          3000.0    0.00725
          8000.0    0.00725                          / RV VS DEPTH EQUIL REGN 01
--        ------    --------
          3000.0    0.00730
          8000.0    0.00730                          / RV VS DEPTH EQUIL REGN 02
--        ------    --------
          3000.0    0.00750
          8000.0    0.00750                          / RV VS DEPTH EQUIL REGN 03
```

Here three tables are entered with a constant CGR versus depth relationship for each equilibration region.

10.3.88 RVW – DEFINE THE INITIAL EQUILIBRATION VAPORIZED WATER IN GAS RATIO FOR ALL GRID BLOCKS

Description

The RVW keyword defines the initial equilibration vaporized water in gas ratio values for all grid cells in the model and should be used in conjunction with the PBUB, PDEW, PRESSURE, RV, SGAS, SOIL and SWAT keywords etc., to fully describe the initial state of the model. The keyword should only be used if both gas and water phases haves been activated in the model via the GAS and WATER keywords, and the VAPWAT is also present activating OPM Flow's Vaporized Water Model. All the aforementioned keywords are in the RUNSPEC section.

This is the non-standard method to initialize the model via enumeration and is seldom employed in the industry. The standard methodology is for OPM Flow to initialize a model using the parameters on the EQUIL keyword combined with other keywords to fully describe the initial state of the model. The keyword can be used with all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	RVW	RVW is an array of real positive numbers assigning the initial equilibration gas-vaporized water ratio values to each cell in the model. Repeat counts may be used, for example 20*1.30.			
		stb/Mscf	sm^3/sm^3	rcc/scc	None

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.

2) The keyword is terminated by a "/".

Table 10.34: RVW Keyword Description

See also the PBUB, PDEW, PRESSURE, RV, SGAS, SOIL and SWAT keywords to fully define the initial state of the model.

Note

This is an OPM Flow specific keyword for the simulator's Water Vaporization Model that is activated by declaring that vaporized water is present in the run using the VAPWAT keyword in the RUNSPEC section.

Use the command line option --enable-opm-rst-file=true to output the RVW data to the RESTART file.

Example

```
--
--       INITIAL EQUILIBRATION WATER VAPOR IN GAS RATIO VALUES FOR ALL CELLS
--       BASED ON NX = 100, NY = 100 AND NZ = 3
--
RVW
         1000*0.0000   1000* 0.0000   1000*1.3000                                /
```

The above example defines the initial equilibration gas-vaporized water values to be 0.000 for all the cells in the first and second layers and 1.3000 for all the cells in the third layer.

10.3.89 RVWVD – EQUILIBRATION VAPORIZED WATER-GAS RATIO (RVW) VERSUS DEPTH TABLES

Description

The RVWVD keyword defines the vaporized water-gas ratio (Rvw) versus depth tables for each equilibration region that should be used when there is vaporized water in the model and the EQLOPT6 variable has been set to a positive integer on the EQUIL keyword in the SOLUTION section.

The keyword should only be used if both gas and water phases haves been activated in the model via the GAS and WATER keywords, and the VAPWAT is also present activating OPM Flow's Vaporized Water Model. All the aforementioned keywords are in the RUNSPEC section.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	DEPTH	A columnar vector of real monotonically increasing down the column values that defines the depth values for the corresponding vaporized oil-gas ratio values, RVW			
		feet	m	cm	None
2	RVW	A columnar vector of real values that defines the vaporized water-gas ratio values, values at the corresponding DEPTH.			
		stb/Mscf	sm^3/sm^3	scc/scc	None

Notes:

1) The keyword is followed by NTEQUL records as declared on the EQLDIMS keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NDRXVD rows as declared on the EQLDIMS keyword in the RUNSPEC section.
3) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 10.35: RVWVD Keyword Description

Alternatively, the vaporized water-gas ratio for each cell may be set via the RVW keyword in the SOLUTION section, if the non-standard method to initialize the model via enumeration is being employed.

See also the EQUIL keywords in the SOLUTION section.

Note

This is an OPM Flow specific keyword for the simulator's Water Vaporization Model that is activated by declaring that vaporized water is present in the run using the VAPWAT keyword in the RUNSPEC section.

Use the command line option --enable-opm-rst-file=true to output the RVW data to the RESTART file.

Example

Given NTEQUL equals three and NDRXVD is greater than or equal to two on the EQLDIMS keyword in the RUNSPEC section, then the following example defines the vaporized water-gas ratio versus depth functions.

```
--
--        DEPTH     RVW
--                  STB/MSCF
--        ------    --------
RVWVD
          3000.0    0.00000
          8000.0    0.00000                          / RVW VS DEPTH EQUIL REGN 01
--        ------    --------
          3000.0    0.00000
          8000.0    0.00000                          / RVW VS DEPTH EQUIL REGN 02
--        ------    --------
          3000.0    0.00100
          8000.0    0.00100                          / RVW VS DEPTH EQUIL REGN 03
```

The example shows three tables for three regions with constant RVW versus depth relationships for each equilibration region, with the first two tables having a zero vaporized water-gas ratio and the last region having a constant 0.001 stb/Mscf versus depth relationship.

10.3.90 SALT – DEFINE THE INITIAL EQUILIBRATION SALT CONCENTRATION FOR ALL GRID BLOCKS

Description

The SALT keyword defines the initial equilibration salt concentration values for all grid cells in the model and should be used in conjunction with the PBUB, PDEW, PRESSURE, RS, RV, SGAS, SGAS and SWAT keywords etc., to fully describe the initial state of the model. The keyword should only be used if the salt (brine) phase has been activated in the model via the BRINE keyword in the RUNSPEC section.

This is the non-standard method to initialize the model via enumeration and is seldom employed in the industry. The standard methodology is for OPM Flow to initialize a model using the parameters on the EQUIL keyword combined with other keywords to fully describe the initial state of the model. The keyword can be used with all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SALT	SALT is an array of real positive numbers that are greater than or equal to zero assigning the initial equilibration salt concentration values to each cell in the model. Repeat counts may be used, for example 20*15.0.			
		lb/stb	kg/sm^3	gm/scc	None

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) The keyword is terminated by a "/".

Table 10.36: SALT Keyword Description

See also the PBUB, PDEW, PRESSURE, RS, RV, SGAS, SOIL and SWAT keywords to fully define the initial state of the model.

Example

```
--
--       DEFINE INITIAL EQUILIBRATION SALT CONCENTRATION FOR ALL CELLS
--       BASED ON NX = 100, NY = 100 AND NZ = 3
--
SALT
         10000*0.0000    10000*0.0000    10000*15.000
/
```

The above example defines the initial equilibration salt concentration values to be 0.0000 for all the cells in the first and second layers and finally 15.000 for all the cells in the third layer.

10.3.91 SALTP – DEFINE THE INITIAL PRECIPITATED SALT VOLUME FRACTION FOR ALL GRID BLOCKS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The SALTP keyword defines the initial equilibration precipitated salt volume fraction values for all grid cells in the model and should be used in conjunction with the PBUB, PDEW, PRESSURE, RS, RV, SGAS, SOIL and SWAT keywords etc., to fully describe the initial state of the model. The keyword should only be used if the salt (brine) phase has been activated in the model via the BRINE keyword, and the PRECSALT keyword to activate OPM Flow's Salt Precipitation Model. Both keywords are in the RUNSPEC section.

This is the non-standard method to initialize the model via enumeration and is seldom employed in the industry. The standard methodology is for OPM Flow to initialize a model using the parameters on the EQUIL keyword combined with other keywords to fully describe the initial state of the model. The keyword can be used with all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SALTP	SALTP is an array of real positive numbers that are greater than or equal to zero and less than or equal to one, that define the initial equilibration salt volume fraction values to each cell in the model. Repeat counts may be used, for example 20*0.15.			
		dimensionless	dimensionless	dimensionless	None

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) The keyword is terminated by a "/".

Table 10.37: SALTP Keyword Description

See also the PBUB, PDEW, PRESSURE, RS, RV, SGAS, SOIL and SWAT keywords to fully define the initial state of the model.

Note

This is an OPM Flow specific keyword for the simulator's Salt Precipitation Model that is activated by the PRECSALT keyword and declaring that vaporized water is present in the run via the VAPWAT in the RUNSPEC section.

This keyword defines the initial precipitated salt volume fraction contained within the pore space. See SALT in the SOLUTION section that defines the initial salt concentration within the water phase.

Example

The example activates the standard Brine Tracking model using the BRINE keyword, OPM Flow's Salt Precipitation model using the PRECSALT keyword, and OPM Flow's vaporized water phase with the VAPWAT keyword; all three keywords are in the RUNSPEC section.

```
-- ==================================================================================
--
-- RUNSPEC SECTION
--
-- ==================================================================================
RUNSPEC
--
--       MAX     MAX     RSVD    TVDP    TVDP
--       EQLNUM  DEPTH   NODES   TABLE   NODES
EQLDIMS
         3       1*      20      1*      1*                                      /
--
--       ACTIVATE STANDARD BRINE MODEL
--
BRINE
--
--       ACTIVATE THE OPM FLOW SALT PRECIPITATION MODEL (OPM FLOW KEYWORD)
--
PRECSALT
--
--       VAPORIZED WATER IN WET GAS IS PRESENT IN THE RUN (OPM FLOW KEYWORD)
--
VAPWAT
```

Then in the SOLUTION section the SALTP keyword would be of the form:

```
-- ==================================================================================
--
-- SOLUTION SECTION
--
-- ==================================================================================
SOLUTION
--
--       DEFINE INITIAL PRECIPITATED SALT VOLUME FRACTION FOR ALL CELLS
--       BASED ON NX = 100, NY = 100 AND NZ = 3
--
SALTP
         1000*0.0000    1000*0.0000    1000*0.100                                 /
```

Here the initial equilibration precipitated salt volume fraction values are set to 0.0000 for all the cells in the first and second layers and finally 0.1000 for all the cells in the third layer.

10.3.92 SALTPVD – INITIAL PRECIPITATED SALT VOLUME FRACTION VERSUS DEPTH TABLES

Description

The SALTPVD keyword defines the initial precipitated salt volume fraction versus depth tables for each equilibration region for when OPM Flow's Salt Precipitation Model has been activated in the input deck via the PRECSALT keyword in the RUNSPEC section. The keyword defines the initial deposited salt as a volume fraction (Ss), that is solid salt saturation.

Note

This is an OPM Flow specific keyword for the simulator's Salt Precipitation Model that is activated by the PRECSALT keyword and declaring that vaporized water is present in the run via the VAPWAT in the RUNSPEC section. This is the initial precipitated salt volume fraction contained within the pore space, see SALTVD in the SOLUTION section that defines the initial salt concentration within the water phase.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	DEPTH	A columnar vector of real monotonically increasing down the column values that defines the depth for corresponding salt volume fraction SALTPSAT.			
		feet	m	cm	None
2	SALTPSAT	A columnar vector of real values that defines the corresponding volume fraction of precipitated salt for the given depth. Note only the standard Brine Model is supported and therefore there should be only one columnar vector of SALTPSAT.			
		dimensionless	dimensionless	dimensionless	None

Notes:

1) The keyword is followed by NTEQUL tables as declared on the EQLDIMS keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NDRXVD rows as declared on the EQIDIMS keyword in the RUNSPEC section.
3) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 10.38: SALTPVD Keyword Description

Precipitated salt can be viewed in OPM ResInsight via the SALTP dynamic grid variable that represents the precipitated salt saturation.

Example

The example activates the standard Brine Tracking model using the BRINE keyword, OPM Flow's Salt Precipitation model using the PRECSALT keyword, and OPM Flow's vaporized water phase with the VAPWAT keyword; all three keywords are in the RUNSPEC section. The example also sets the number of equilibrium regions to three (NTEQUL set to three on the EQLDIMS keyword also in the RUNSPEC), that is:

```
-- ======================================================================
--
-- RUNSPEC SECTION
--
-- ======================================================================
RUNSPEC
--
--        MAX     MAX     RSVD    TVDP    TVDP
--        EQLNUM  DEPTH   NODES   TABLE   NODES
EQLDIMS
          3       1*      20      1*      1*                              /
--
--        ACTIVATE STANDARD BRINE MODEL
--
BRINE
--
--        ACTIVATE THE OPM FLOW SALT PRECIPITATION MODEL (OPM FLOW KEYWORD)
--
PRECSALT
--
--        VAPORIZED WATER IN WET GAS IS PRESENT IN THE RUN (OPM FLOW KEYWORD)
--
VAPWAT
```

Then in the SOLUTION section the SALTPVD keyword would be of the form:

```
-- ======================================================================
--
-- SOLUTION SECTION
--
-- ======================================================================
SOLUTION
--
--        PRECIPITATED SALT VOLUME FRACTION VERSUS DEPTH (OPM FLOW KEYWORD)
--
--        DEPTH    SALTPSAT
--        ------   --------
SALTPVD
          3000.0    0.000
          8000.0    0.000                                  / EQUIL REGN 01
--        ------   --------
          3000.0    0.000
          8000.0    0.000                                  / EQUIL REGN 02
--        ------   --------
          3000.0    0.000
          8000.0    0.000                                  / EQUIL REGN 03
```

Here the initial precipitated salt volume fraction has been set to zero for all three equilibration regions.

10.3.93 SALTREST – DEFINE THE RESTART SALT CONCENTRATION FOR ALL GRID BLOCKS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The SALTREST keyword defines restart salt concentration values for all grid cells in the model and should be used in runs that are using the RESTART facility, where the initial run has not used the Low Salt or Brine options. This allows for initial runs that have used the standard water PVT properties via the PVTW keyword in the PROPS section, to be restarted with salt dependent water properties. The keyword should only be used if the salt (brine) phase has been activated in the current restart run (not the initial run) via the BRINE keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SALTREST	SALTREST is an array of real positive numbers that are greater than or equal to zero assigning the restart salt concentration values to each cell in the model. Repeat counts may be used, for example 20*15.0.			
		lb/stb	kg/sm^3	gm/scc	None

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.

2) The keyword is terminated by a "/".

Table 10.39: SALTREST Keyword Description

See also the PVTWSALT keyword in the PROPS section and the RESTART keyword in the SOLUTION section.

Example

```
--
--       DEFINE RESTART SALTREST VALUES FOR ALL CELLS IN THE MODEL
--       BASED ON NX = 100, NY = 100 AND NZ = 3
--
SALTREST
         1000*0.0000    1000*0.0000    1000*15.000                    /
```

The above example defines the restart salt concentration values to be 0.0000 for all the cells in the first and second layers and finally 15.000 for all the cells in the third layer.

10.3.94 SALTVD – EQUILIBRATION SALT CONCENTRATION VERSUS DEPTH TABLES

Description

The SALTVD keyword defines the initial salt concentration versus depth tables for each equilibration region for when the salt (brine) phase has been activated in the model via the BRINE keyword in the RUNSPEC section, and the EQLOPT1 variable has been set to a positive integer on the EQUIL keyword in the SOLUTION section. Secondly, the keyword should also be used to set the initial salt concentration versus depth if OPM Flow's PRECSALT keyword in the RUNSPEC section has been used to activate the simulators Salt Precipitation model.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	DEPTH	A columnar vector of real monotonically increasing down the column values that defines the depth for corresponding salt concentrations SALTCON.			
		feet	m	cm	None
2	SALTCON	A columnar vector of real monotonically increasing down the column values that defines the corresponding salt concentration within the water phase for the given depth. There should be one columnar vector for each type of salt. For the standard Brine Model there is only one salt type and therefore there should be only one columnar vector of SALTCON. However, if the BRINE keyword has been invoked with the ECLMC keyword in the RUNSPEC section, then there should one columnar SALTCON vector for each declared salt type. It is recommended to provide initial salt concentrations less then or equal to values provided by SALTSOL keyword in the PROPS section.			
		lb/stb	kg/sm^3	gm/scc	None

Notes:

1) The keyword is followed by NTEQUIL tables as declared on the EQLDIMS keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NDRXVD rows as declared on the EQIDIMS keyword in the RUNSPEC section.
3) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 10.40: SALTVD Keyword Description

Note

This is the initial salt concentration contained within the water phase, see the SALTPVD keyword in the SOLUTION section that defines the initial salt volume fraction that has been precipitated into the pore space.

Examples

The first example activates the standard Brine Tracking model using the BRINE keyword in the RUNSPEC section and sets the number of equilibrium regions to three (NTEQUIL set to 3 on the EQLDIMS keyword also in the RUNSPEC), that is:

```
-- ======================================================================
--
-- RUNSPEC SECTION
--
-- ======================================================================
RUNSPEC
--
--       MAX     MAX     RSVD    TVDP    TVDP
--       EQLNUM  DEPTH   NODES   TABLE   NODES
EQLDIMS
         3       1*      20      1*      1*                                /
--
--       ACTIVATE STANDARD BRINE MODEL
--
BRINE
```

Then in the SOLUTION section the SALTVD keyword would be of the form:

```
-- ======================================================================
--
-- SOLUTION SECTION
--
-- ======================================================================
SOLUTION
--
--       DEPTH    SALT-1    SALT-2    SALT-3    SALT-4
--                SALTCON   SALTCON   SALTCON   SALTCON
--       ------   -------   -------   -------   -------
SALTVD
         3000.0    1.200
         8000.0    1.200                              / EQUIL REGN 01
--       ------   --------
         3000.0    1.300
         8000.0    1.300                              / EQUIL REGN 02
--       ------   --------
         3000.0    1.400
         8000.0    1.400                              / EQUIL REGN 03
```

The next example shows how the SALTVD keyword is entered when both the ECLMC and BRINE keywords have activated the Multi-Component Brine model in the RUNSPEC section, that is:

```
-- ================================================================================
--
-- RUNSPEC SECTION
--
-- ================================================================================
RUNSPEC
--
--       MAX     MAX     RSVD    TVDP    TVDP
--       EQLNUM  DEPTH   NODES   TABLE   NODES
EQLDIMS
         3       1*      20      1*      1*                                    /
--
--
--       ACTIVATE MULTI-COMPONENT BRINE MODEL
--
ECLMC
--
--       DEFINE WATER PHASE MULTI-COMPONENT BRINE COMPONENTS
--
--       SALT1   SALT2   SALT3   SALT4   SALT5
BRINE
         NACL    CACL    MGC03                                                 /
```

The above example activates the Multi-Component Brine model with three different water salinities for three equilibrium regions. In this case the resulting SALTVD keyword would be of the form:

```
-- ================================================================================
--
-- SOLUTION SECTION
--
-- ================================================================================
SOLUTION
--
--       DEPTH    SALT-01   SALT-02   SALT-03   SALT-04
--                CONCENTR  CONCENTR  CONCENTR  CONCENTR
--       ------   --------  --------  --------  --------
SALTVD
         3000.0    1.200    0.540     0.020
         7000.0    1.200    0.640     0.040             / EQUIL REGN 01
--       ------   --------  --------  --------
         3000.0    1.300    0.440     0.020
         8000.0    1.300    0.540     0.040             / EQUIL REGN 02
--       ------   --------  --------  --------
         5000.0    1.400    0.640     0.002
         8000.0    1.400    0.640     0.002             / EQUIL REGN 03
```

In this case there are three data sets, on one for each equilibrium region and three SALTCON columnar vectors, one for each salt type (NACL, CACL and MGC03) declared via the BRINE keyword in the RUNSPEC section.

Note that the Multi-Component Brine model is not available in OPM Flow.

10.3.95 SBIOF - DEFINE THE INITIAL EQUILIBRATION BIOFILM VOLUME FRACTION FOR ALL GRID BLOCKS

Description

The SBIOF keyword defines the initial equilibration biofilm volume fraction for all grid cells in the model and should be used in conjunction with the PRESSURE, SCALC, SMICR, SOXYG, and SUREA keywords to fully describe the initial state of the model. The keyword should only be used if the MICP model has been activated via the MICP keyword in the RUNSPEC section.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SBIOF	SBIOF is an array of real positive numbers that are greater than or equal to zero and less than or equal to one assigning the initial equilibration biofilm volume fraction values to each cell in the model. Repeat counts may be used, for example 20*0..0010.			None
		dimensionless	dimensionless	dimensionless	

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) The keyword is terminated by a "/".

Table 10.41: SBIOF Keyword Description

See also the PRESSURE, SCALC, SMICR, SOXYG, and SUREA keywords to fully define the initial state of the model.

Example

```
--
--       DEFINE INITIAL EQUILIBRATION BIOFILM VOLUME FRACTION FOR ALL CELLS
--       BASED ON NX = 100, NY = 100 AND NZ = 3
--
SBIOF
         10000*0.0000 10000*0.0000 10000*0.0010
/
```

The above example defines the initial equilibration biofilm volume fraction values to be 0.0000 for all the cells in the first and second layers and finally 0.0010 for all the cells in the third layer.

10.3.96 SCALC – DEFINE THE INITIAL EQUILIBRATION CALCITE VOLUME FRACTION FOR ALL GRID BLOCKS

Description

The SCALC keyword defines the initial equilibration calcite volume fraction for all grid cells in the model and should be used in conjunction with the PRESSURE, SBIOF, SMICR, SOXYG and SUREA keywords to fully describe the initial state of the model. The keyword should only be used if the MICP model has been activated via the MICP keyword in the RUNSPEC section.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SCALC	SCALC is an array of real positive numbers that are greater than or equal to zero and less than or equal to one assigning the initial equilibration calcite volume fraction values to each cell in the model. Repeat counts may be used, for example 20*0.0010.			None
		dimensionless	dimensionless	dimensionless	

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.

2) The keyword is terminated by a "/".

Table 10.42: SCALC Keyword Description

See also the PRESSURE, SBIOF, SMICR, SOXYG and SUREA keywords to fully define the initial state of the model.

Example

```
--
--       DEFINE INITIAL EQUILIBRATION CALCITE VOLUME FRACTION FOR ALL CELLS
--       BASED ON NX = 100, NY = 100 AND NZ = 3
--
SCALC
         1000*0.0000 1000*0.0000 1000*0.0010                                  /
```

The above example defines the initial equilibration calcite volume fraction values to be 0.0000 for all the cells in the first and second layers and finally 0.0010 for all the cells in the third layer.

10.3.97 SCVD – DEFINE EQUILIBRATION COAL SOLVENT CONCENTRATION VERSUS DEPTH TABLES

Description

The SCVD keyword defines the initial coal solvent concentration versus depth tables for each equilibration region for when the coal phase has been activated in the run via the COAL keyword in the RUNSPEC section. The keyword may be used in conjunction with the SOLVCONC keyword in the SOLUTION section, to fully describe the initial state of the model. Note both SOLVCONC and SCVD are optional as the simulator will calculate the coal gas concentration based on the equilibrium concentration and the block pressure.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	DEPTH	A columnar vector of real monotonically increasing down the column values that defines the depth values for the corresponding coal solvent concentration, SCVALS.			
		feet	m	cm	None
2	SCVALS	A columnar vector of real values that defines the coal solvent concentration values at the corresponding DEPTH.			
		Mscf/ft3	sm^3/m^3	scc/cc	None

Notes:

1) The keyword is followed by NTEQUL records as declared on the EQLDIMS keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NDRXVD rows as declared on the EQLDIMS keyword in the RUNSPEC section.
3) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 10.43: SCVD Keyword Description

See also the SOLVCONC, GCVD, GASCONC and GASSATC keywords in the SOLUTION section.

Example

Given NTEQUL equals three and NDRXVD is greater than or equal to two on the EQLDIMS keyword in the RUNSPEC section, then the following example defines the coal solvent concentration versus depth functions.

```
--
--        DEPTH     SOLVC
--                  MSCF/FT
--        ------    --------
SCVD
           100.0    75.5000
          1000.0    75.5000                          / SC VS DEPTH EQUIL REGN 01
--        ------    --------
           100.0    65.5000
          1000.0    65.5000                          / SC VS DEPTH EQUIL REGN 02
--        ------    --------
           100.0    60.0000
          1000.0    60.0000                          / SC VS DEPTH EQUIL REGN 03
```

Here three tables are entered with a constant coal solvent concentration versus depth relationship for each equilibration region

10.3.98 SFOAM – DEFINE THE INITIAL EQUILIBRATION FOAM CONCENTRATION FOR ALL GRID BLOCKS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The SFOAM keyword defines the initial equilibration foam concentration values for all grid cells in the model and should be used in conjunction with the PBUB, PDEW, PRESSURE, RS, RV, SGAS, SGAS and SWAT keywords etc., to fully describe the initial state of the model. The keyword should only be used if the foam phase has been activated in the model via the FOAM keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

This is the non-standard method to initialize the model via enumeration and is seldom employed in the industry. The standard methodology is for OPM Flow to initialize a model using the parameters on the EQUIL keyword combined with other keywords to fully describe the initial state of the model. The keyword can be used with all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SFOAM	SFOAM is an array of real positive numbers that are greater than or equal to zero assigning the initial equilibration foam concentration values to each cell in the model. Units are dependent on the transport phase specified via the FOAMOPT1 variable on the FOAMOPTS keyword in the PROPS section. FOAMOPT1 should be set to either GAS or WATER. Repeat counts may be used, for example 20*0.5			
		Gas: lb/Mscf Water: lb/stb	Gas: kg/sm^3 Water: kg/sm^3	Gas: gm/scc Water: gm/scc	None

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.

2) The keyword is terminated by a "/".

Table 10.44: SFOAM Keyword Description

See also the PBUB, PDEW, PRESSURE, RS, RV, SGAS, SOIL and SWAT keywords to fully define the initial state of the model.

Example

```
--
--       DEFINE INITIAL EQUILIBRATION FOAM VALUES FOR ALL CELLS IN THE MODEL
--       BASED ON NX = 100, NY = 100 AND NZ = 3
--
SFOAM
         1000*0.0000    1000*0.0000    1000*0.500                          /
```

The above example defines the initial equilibration foam concentration values to be 0.0000 for all the cells in the first and second layers and finally 0.500 for all the cells in the third layer.

10.3.99 SGAS – DEFINE THE INITIAL EQUILIBRATION GAS SATURATION FOR ALL GRID BLOCKS

Description

The SGAS keyword defines the initial equilibration gas saturation values for all grid cells in the model and should be used in conjunction with the PBUB, PDEW, PRESSURE, RS, RV, SOIL and SWAT keywords etc., to fully describe the initial state of the model. The keyword should only be used if the gas phase has been activated in the model via the GAS keyword in the RUNSPEC section.

This is the non-standard method to initialize the model via enumeration and is seldom employed in the industry. The standard methodology is for OPM Flow to initialize a model using the parameters on the EQUIL keyword combined with other keywords to fully describe the initial state of the model. The keyword can be used with all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SGAS	SGAS is an array of real positive numbers that are greater than or equal to zero and less than or equal to one assigning the initial equilibration gas saturation values to each cell in the model. Repeat counts may be used, for example 20*0.600.			
		dimensionless	dimensionless	dimensionless	None

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.

2) The keyword is terminated by a "/".

Table 10.45: SGAS Keyword Description

Note for two phase runs it is only necessary to enter one saturation array of one of the phases present in the run (SGAS, SOIL, or SWAT), as the simulator will calculate the other phases by difference. Similarly for three phase runs it is only necessary to enter the array data for two of the phases, as the third saturation will again be calculated by the simulator.

See also the PBUB, PDEW, PRESSURE, RS, RV, SOIL and SWAT keywords to fully define the initial state of the model.

Example

```
--
--       DEFINE INITIAL EQUILIBRATION GAS SAT VALUES FOR ALL CELLS IN THE MODEL
--       BASED ON NX = 100, NY = 100 AND NZ = 3
--
SGAS
         1000*0.7000    1000*0.6500    1000*0.6000                              /
```

The above example defines the initial equilibration gas saturation values to be 0.7000 for all the cells in the first layer, 0.6500 for all the cells in the second layer, and finally 0.6000 for all the cells in the third layer.

10.3.100 SKIP – ACTIVATE SKIPPING OF ALL KEYWORDS AND INPUT DATA

The SKIP keyword turns on skipping of keywords until the ENDSKIP activation keyword is encountered. All keywords and between the SKIP and ENDSKIP keywords are ignored by OPM Flow.

See *SKIP – Activate Skipping of All Keywords and Input Data* in the GLOBAL section for a full description.

10.3.101 SKIP100 – ACTIVATE SKIPPING OF BLACK-OIL KEYWORDS AND INPUT DATA

This keyword turns on skipping of black-oil keywords until the ENDSKIP activation keyword is encountered. All black-oil keywords between the SKIP100 and ENDSKIP keywords are ignored by OPM Flow.

See *SKIP100 – Activate Skipping of Black-Oil Keywords and Input Data* in the GLOBAL section for a full description.

10.3.102 SKIP300 – ACTIVATE SKIPPING OF "COMPOSITIONAL" KEYWORDS AND INPUT DATA

Turns on skipping of "compositional" keywords until the ENDSKIP activation keyword is encountered. All "compositional" keywords between the SKIP300 and ENDSKIP keywords are ignored by OPM Flow.

See *Error: Reference source not found* in the GLOBAL section for a full description.

10.3.103 SMICR – DEFINE THE INITIAL EQUILIBRATION MICROBIAL CONCENTRATION FOR ALL GRID BLOCKS

Description

The SMICR keyword defines the initial equilibration microbial concentration values for all grid cells in the model and should be used in conjunction with the PRESSURE, SBIOF, SCALC, SOXYG, and SUREA keywords to fully describe the initial state of the model. The keyword should only be used if the MICP model has been activated via the MICP keyword in the RUNSPEC section.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SMICR	SMICR is an array of real positive numbers that are greater than or equal to zero assigning the initial equilibration microbial concentration values to each cell in the model. Repeat counts may be used, for example 20*0.1500.			None
		lb/stb	kg/sm^3	gm/scc	

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.

2) The keyword is terminated by a "/".

Table 10.46: SMICR Keyword Description

See also the PRESSURE, SBIOF, SCALC, SOXYG, and SUREA keywords to fully define the initial state of the model.

Example

```
--
--       DEFINE INITIAL EQUILIBRATION MICROBIAL CONCENTRATION FOR ALL CELLS
--       BASED ON NX = 100, NY = 100 AND NZ = 3
--
SMICR
         1000*0.0000 1000*0.0000 1000*0.1500                                  /
```

The above example defines the initial equilibration microbial concentration values to be 0.0000 for all the cells in the first and second layers and finally 0.1500 for all the cells in the third layer.

10.3.104 SOIL – DEFINE THE INITIAL EQUILIBRATION OIL SATURATION FOR ALL GRID BLOCKS

Description

The SOIL keyword defines the initial equilibration oil saturation values for all grid cells in the model and should be used in conjunction with the PBUB, PDEW, PRESSURE, RS, RV, SGAS and SWAT keywords etc., to fully describe the initial state of the model. The keyword should only be used if the oil phase has been activated in the model via the OIL keyword in the RUNSPEC section.

This is the non-standard method to initialize the model via enumeration and is seldom employed in the industry. The standard methodology is for OPM Flow to initialize a model using the parameters on the EQUIL keyword combined with other keywords to fully describe the initial state of the model. The keyword can be used with all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SOIL	SOIL is an array of real positive numbers that are greater than or equal to zero and less than or equal to one assigning the initial equilibration oil saturation values to each cell in the model. Repeat counts may be used, for example 20*0.600.			None
		dimensionless	dimensionless	dimensionless	

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.

2) The keyword is terminated by a "/".

Table 10.47: SOIL Keyword Description

Note for two phase runs it is only necessary to enter one saturation array of one of the phases present in the run (SGAS, SOIL, or SWAT), as the simulator will calculate the other phases by difference. Similarly for three phase runs it is only necessary to enter the array data for two of the phases, as the third saturation will again be calculated by the simulator.

See also the PBUB, PDEW, PRESSURE, RS, RV, SGAS and SWAT keywords to fully define the initial state of the model.

Example

```
--
--       DEFINE INITIAL EQUILIBRATION OIL SAT VALUES FOR ALL CELLS IN THE MODEL
--       BASED ON NX = 100, NY = 100 AND NZ = 3
--
SOIL
         1000*0.7000    1000*0.6500    1000*0.6000                              /
```

The above example defines the initial equilibration oil saturation values to be 0.7000 for all the cells in the first layer, 0.6500 for all the cells in the second layer, and finally 0.6000 for all the cells in the third layer.

10.3.105 SOLUTION - DEFINE THE START OF THE SOLUTION SECTION OF KEYWORDS

Description

The SOLUTION activation keyword marks the end of the REGIONS section and the start of the SOLUTION section that defines the parameters used to initialize the model, by:

1) defining fluid contacts and pressures, or
2) defining pressures and fluid saturations for all cells in the model, or
3) by restarting from a previously OPM Flow completed run.

There is no data required for this keyword.

Example

```
-- ================================================================================
--
-- SOLUTION SECTION
--
-- ================================================================================
SOLUTION
```

The above example marks the end of the REGIONS section and the start of the SOLUTION section in the OPM Flow data input file.

10.3.106 SOLVCONC – DEFINE THE INITIAL COAL SOLVENT CONCENTRATION FOR ALL GRID BLOCKS

Description

The SOLVCONC keyword defines the initial coal solvent concentration values for all matrix grid cells in the model and should be used in conjunction with the SCVD keyword in the SOLUTION section, to fully describe the initial state of the model. The keyword should only be used if the coal phase has been activated in the model via the COAL keyword in the RUNSPEC section. Note both SOLVCONC and SCVD are optional as the simulator will calculate the coal solvent concentration based on the equilibrium concentration and the block pressure.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

This is the non-standard method to initialize the model via enumeration and is seldom employed in the industry. The standard methodology is for OPM Flow to initialize a model using the parameters on the EQUIL keyword combined with other keywords to fully describe the initial state of the model. The keyword can be used with all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SOLVCONC	SOLVCONC is an array of real positive numbers that define the initial equilibration coal solvent concentration values to each matrix cell in the model. Repeat counts may be used, for example 20*75.0.			
		Mscf/ft3	sm^3/m^3	scc/cc	None
Notes: 1) The number of entries should correspond to number of matrix cells in the model, that is (NX x NY x NZ) divided by two, as per the parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement. 2) The keyword is terminated by a "/".					

Table 10.48: SOLVCONC Keyword Description

See also the SCVD keyword in the SOLUTION section to fully define the initial state of the model.

Example

```
--
--       DEFINE INITIAL EQUILIBRATION COAL SOLVENT CONCENTRATION FOR ALL CELLS
--       BASED ON NX = 100, NY = 100 AND NZ = 6
--
SOLVCONC
         1000*75.500    1000*65.500    1000*60.000                          /
```

The above example defines the initial coal solvent concentration values to be 75.500 for all the matrix cells in the first layer, 65.500 for all the cells in the second layer, and finally 60.000 for all the cells in the third layer.

10.3.107 SOLVFRAC – DEFINE THE INITIAL GAS SOLVENT FRACTION FOR ALL GRID BLOCKS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The SOLVFRAC keyword defines the initial solvent faction within the gas phase values for all matrix grid cells in the model. The keyword should only be used if the coal phase has been activated in the model via the COAL keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

SOLVFRAC is used with the standard equilibration method to initialize the model via the EQUIL keyword in the RUNSPEC section, as oppose to the non-standard enumeration method.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SOLVFRAC	SOLVFRAC is an array of real positive numbers that define the initial solvent fraction within the gas phase values for each matrix cell in the model. Repeat counts may be used, for example 20*0.075.			
		dimensionless	dimensionless	dimensionless	None

Notes:

1) The number of entries should correspond to number of matrix cells in the model, that is (NX x NY x NZ) divided by two, as per the parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) The keyword is terminated by a "/".

Table 10.49: SOLVFRAC Keyword Description

See also the EQUIL keyword in the SOLUTION section to fully define the initial state of the model.

Example

```
--
--       DEFINE INITIAL EQUILIBRATION GAS SOLVENT FRACTION FOR ALL CELLS
--       BASED ON NX = 100, NY = 100 AND NZ = 6
--
SOLVFRAC
         1000*0.0250    1000*0.0350    1000*0.0500                              /
```

The above example defines the initial gas solvent fraction values to be 0.250 for all the matrix cells in the first layer, 0.0350 for all the cells in the second layer, and finally 0.0500 for all the cells in the third layer.

10.3.108 SOXYG - DEFINE THE INITIAL EQUILIBRATION OXYGEN CONCENTRATION FOR ALL GRID BLOCKS

Description

The SOXYG keyword defines the initial equilibration oxygen concentration values for all grid cells in the model and should be used in conjunction with the PRESSURE, SBIOF, SCALC, SMICR, and SUREA keywords to fully describe the initial state of the model. The keyword should only be used if the MICP model has been activated via the MICP keyword in the RUNSPEC section.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SOXYG	SOXYG is an array of real positive numbers that are greater than or equal to zero assigning the initial equilibration oxygen concentration values to each cell in the model. Repeat counts may be used, for example 20*0.1500			None
		lb/stb	kg/sm^3	gm/scc	

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.

2) The keyword is terminated by a "/".

Table 10.50: SOXYG Keyword Description

See also the PRESSURE, SBIOF, SCALC, SMICR, and SUREA keywords to fully define the initial state of the model.

Example

```
--
--       DEFINE INITIAL EQUILIBRATION OXYGEN CONCENTRATION FOR ALL CELLS
--       BASED ON NX = 100, NY = 100 AND NZ = 3
--
SOXYG
         1000*0.0000 1000*0.0000 1000*0.1500                                  /
```

The above example defines the initial equilibration oxygen concentration values to be 0.0000 for all the cells in the first and second layers and finally 0.1500 for all the cells in the third layer.

10.3.109 SPOLY – DEFINE THE INITIAL EQUILIBRATION POLYMER CONCENTRATION FOR ALL GRID BLOCKS

Description

The SPOLY keyword defines the initial equilibration polymer concentration values for all grid cells in the model and should be used in conjunction with the PBUB, PDEW, PRESSURE, RS, RV, SGAS, SGAS and SWAT keywords etc., to fully describe the initial state of the model. The keyword should only be used if the polymer phase has been activated in the model via the POLYMER keyword in the RUNSPEC section.

This is the non-standard method to initialize the model via enumeration and is seldom employed in the industry. The standard methodology is for OPM Flow to initialize a model using the parameters on the EQUIL keyword combined with other keywords to fully describe the initial state of the model. The keyword can be used with all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SPOLY	SPOLY is an array of real positive numbers that are greater than or equal to zero assigning the initial equilibration polymer concentration values to each cell in the model. Repeat counts may be used, for example 20*25.0.			
		lb/stb	kg/sm^3	gm/scc	None

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) The keyword is terminated by a "/".

Table 10.51: SPOLY Keyword Description

See also the PBUB, PDEW, PRESSURE, RS, RV, SGAS, SOIL and SWAT keywords to fully define the initial state of the model.

Example

```
--
--       DEFINE INITIAL EQUILIBRATION POLYMER VALUES FOR ALL CELLS IN THE MODEL
--       BASED ON NX = 100, NY = 100 AND NZ = 3
--
SPOLY
         1000*0.0000    1000*0.0000    1000*15.000                          /
```

The above example defines the initial equilibration polymer concentration values to be 0.0000 for all the cells in the first and second layers and finally 15.000 for all the cells in the third layer.

10.3.110 SPOLYMW – DEFINE THE INITIAL EQUILIBRATION POLYMER MOLECULAR WEIGHTS FOR ALL GRID BLOCKS

Description

The SPOLYMW keyword defines the initial equilibration polymer molecular weights for all grid cells in the model and should only be be used with OPM Flow's Polymer Molecular Weight Transport option, together with the other standard equilibration keywords, in order to fully describe the initial state of the model.

This keyword should only be used if the POLYMER and POLYMW keywords in the RUNSPEC section are also activated.

Note

This is an OPM Flow specific keyword that employs an alternative polymer flood model based on a Polymer Molecular Weight Transport equation, that is not available in the commercial simulator.

The model has been tested using metric units; however, using either field or laboratory units with the option should be considered experimental.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SPOLYMW	SPOLYMW is an array of real positive numbers that are greater than or equal to zero assigning the initial equilibration polymer molecular weights to each cell in the model. Repeat counts may be used, for example 20*5.0			0.0
		lb/lb-M	kg/kg-M	gm/gm-M	

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.

2) The keyword is terminated by a "/".

Table 10.52: SPOLYMW Keyword Description

See also the PBUB, PDEW, PRESSURE, RS, RV, SGAS, SOIL and SWAT keywords to fully define the initial state of the model.

Example

```
--
--        INITIAL EQUILIBRATION POLYMER MOLECULAR WEIGHTS FOR ALL CELLS
--
--        ARRAY       CONSTANT     ---------- BOX ---------
--                                 I1  I2   J1  J2   K1  K2
EQUALS
          SPOLYMW     0.0000       1*  1*   1*  1*    1   5 / LAYERS 1 TO 5
          SPOLYMW     5.0000       1*  1*   1*  1*    6   7 / LAYERS 6 TO 7
          SPOLYMW     0.0000       1*  1*   1*  1*    8  20 / LAYERS 8 TO 20
/
```

The above example defines the initial equilibration polymer molecular weights to be 0.0000 for all the cells, except for layers six to seven, where the polymer molecular weight is set to five for these cells.

10.3.111 SSOL – DEFINE THE INITIAL EQUILIBRATION SOLVENT SATURATION FOR ALL GRID BLOCKS

Description

The SSOL keyword defines the initial equilibration solvent saturation values for all grid cells in the model and should be used in conjunction with the PBUB, PDEW, PRESSURE, RS, RV, SGAS, SOIL and SWAT keywords etc., to fully describe the initial state of the model. The keyword should only be used if the solvent phase has been activated in the model via the SOLVENT keyword in the RUNSPEC section.

This is the non-standard method to initialize the model via enumeration and is seldom employed in the industry. The standard methodology is for OPM Flow to initialize a model using the parameters on the EQUIL keyword combined with other keywords to fully describe the initial state of the model. The keyword can be used with all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SSOL	SSOL is an array of real positive numbers that are greater than or equal to zero and less than or equal to one assigning the initial equilibration solvent saturation values to each cell in the model. Repeat counts may be used, for example 20*0.000.			
		dimensionless	dimensionless	dimensionless	None

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) The keyword is terminated by a "/".

Table 10.53: SSOL Keyword Description

See also the PBUB, PDEW, PRESSURE, RS, RV, SGAS, SOIL, and SWAT keywords to fully define the initial state of the model.

Example

```
--
--       DEFINE INITIAL EQUILIBRATION GAS SAT VALUES FOR ALL CELLS IN THE MODEL
--       BASED ON NX = 100, NY = 100 AND NZ = 3
--
SSOL
         1000*0.0000    1000*0.0000    1000*0.0000                              /
```

The above example defines the initial equilibration solvent saturation values to be 0.0 for all the cells in the in the model.

10.3.112 SUREA - DEFINE THE INITIAL EQUILIBRATION UREA CONCENTRATION FOR ALL GRID BLOCKS

Description

The SUREA keyword defines the initial equilibration urea concentration values for all grid cells in the model and should be used in conjunction with the PRESSURE, SBIOF, SCALC, SMICR, and SOXYG keywords to fully describe the initial state of the model. The keyword should only be used if the MICP model has been activated via the MICP keyword in the RUNSPEC section.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SUREA	SUREA is an array of real positive numbers that are greater than or equal to zero assigning the initial equilibration urea concentration values to each cell in the model. Repeat counts may be used, for example 20*30.0.			None
		lb/stb	kg/sm^3	gm/scc	

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.

2) The keyword is terminated by a "/".

Table 10.54: SUREA Keyword Description

See also the PRESSURE, SBIOF, SCALC, SMICR, and SOXYG keywords to fully define the initial state of the model.

Example

```
--
--       DEFINE INITIAL EQUILIBRATION UREA CONCENTRATION FOR ALL CELLS
--       BASED ON NX = 100, NY = 100 AND NZ = 3
--
SUREA
         1000*0.0000 1000*0.0000 1000*20.0                                  /
```

The above example defines the initial equilibration urea concentration values to be 0.0000 for all the cells in the first and second layers and finally 20.0 for all the cells in the third layer.

10.3.113 SURF – DEFINE THE INITIAL EQUILIBRATION POLYMER CONCENTRATION FOR ALL GRID BLOCKS

Description

The SURF keyword defines the initial equilibration surfactant concentration values for all grid cells in the model and should be used in conjunction with the PBUB, PDEW, PRESSURE, RS, RV, SGAS, SGAS and SWAT keywords etc., to fully describe the initial state of the model. The keyword should only be used if the surfactant phase has been activated in the model via the SURFACT keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

This is the non-standard method to initialize the model via enumeration and is seldom employed in the industry. The standard methodology is for OPM Flow to initialize a model using the parameters on the EQUIL keyword combined with other keywords to fully describe the initial state of the model. The keyword can be used with all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SURF	SURF is an array of real positive numbers that are greater than or equal to zero assigning the initial equilibration surfactant concentration values to each cell in the model. Repeat counts may be used, for example 20*25.0.			
		lb/stb	kg/sm^3	gm/scc	None
Notes: 1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement. 2) The keyword is terminated by a "/".					

Table 10.55: SURF Keyword Description

See also the PBUB, PDEW, PRESSURE, RS, RV, SGAS, SOIL and SWAT keywords to fully define the initial state of the model.

Example

```
--
--       DEFINE INITIAL EQUILIBRATION SURFACTANT VALUES FOR ALL CELLS
--       BASED ON NX = 100, NY = 100 AND NZ = 3
--
SURF
         1000*0.0000    1000*0.0000    1000*0.2500                          /
```

The above example defines the initial equilibration surfactant concentration values to be 0.0000 for all the cells in the first and second layers and finally 0.2500 for all the cells in the third layer.

10.3.114 SWAT – DEFINE THE INITIAL EQUILIBRATION WATER SATURATION FOR ALL GRID BLOCKS

Description

The SWAT keyword defines the initial equilibration water saturation values for all grid cells in the model and should be used in conjunction with the PBUB, PDEW, PRESSURE, RS, RV, SGAS and SOIL keywords etc., to fully describe the initial state of the model. The keyword should only be used if the water phase has been activated in the model via the WATER keyword in the RUNSPEC section.

This is the non-standard method to initialize the model via enumeration and is seldom employed in the industry. The standard methodology is for OPM Flow to initialize a model using the parameters on the EQUIL keyword combined with other keywords to fully describe the initial state of the model. The keyword can be used with all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SWAT	SWAT is an array of real positive numbers that are greater than or equal to zero and less than or equal to one assigning the initial equilibration water saturation values to each cell in the model. Repeat counts may be used, for example 20*0.300.			
		dimensionless	dimensionless	dimensionless	None
Notes: 1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement. 2) The keyword is terminated by a "/".					

Table 10.56: SWAT Keyword Description

Note for two phase runs it is only necessary to enter one saturation array of one of the phases present in the run (SGAS, SOIL, or SWAT), as the simulator will calculate the other phases by difference. Similarly for three phase runs it is only necessary to enter the array data for two of the phases, as the third saturation will again be calculated by the simulator.

See also the PBUB, PDEW, PRESSURE, RS, RV, SGAS and SOIL keywords to fully define the initial state of the model.

Example

```
--
--       DEFINE INITIAL EQUILIBRATION WAT SAT VALUES FOR ALL CELLS IN THE MODEL
--       BASED ON NX = 100, NY = 100 AND NZ = 3
--
SWAT
         1000*0.2000    1000*0.2500    1000*0.4500                              /
```

The above example defines the initial equilibration water saturation values to be 0.2000 for all the cells in the first layer, 0.2500 for all the cells in the second layer, and finally 0.4500 for all the cells in the third layer.

10.3.115 TBLK – DEFINE TRACER INITIAL GRID BLOCK CONCENTRATIONS

Description

TBLK keyword defines the initial tracer concentration for all or selected cells in the model, for when the TRACERS keyword in the RUNSPEC section has declared the maximum number of tracers for each phase, and the TRACER keyword in the PROPS section has defined the tracer. This keyword is not in the standard keyword format due to the tracer name being concatenated to the keyword TBLK to fully define the tracer being initialized.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	NAME	A character string of up to eight characters, consisting of TBLK as the first four characters followed by a four letter character string defining the tracer's name. The fifth character should either be the letter F or the letter S, that indicates the state of the tracer either to be free (F) or in solution (S). For example, TBLKFIGS (free) or TBLKSIGS (solution). The last three characters of NAME (the effective tracer name) must also match an entry on the TRACER keyword's NAME parameter, in the PROPS section. Note it is best to void names beginning with the letters F, S, and T as these names may create naming issues in post-processing software.			None
2	TBLK	TBLK is an array of real numbers greater than or equal to zero, that are assigned the tracer concentration values for each cell in the model or the current input BOX. Repeat counts may be used, for example 200*0.0. The units for the tracer, if required, are set on the TRACER keyword in the PROPS section. This should be the same as the PHASE in the model.			
		Liquid: TBLK/stb Gas: TBLK/Mscf	Liquid: TBLK/sm^3 Gas: TBLK/sm^3	Liquid: TBLK/scc Gas: TBLK/scc	None

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.

2) The keyword is terminated by a "/".

Table 10.57: TBLK Keyword Description

See also the TRACERS keyword in the RUNSPEC section to declared the maximum number of tracers for each phase, the TRACER keyword in the PROPS section to define the tracer, and the WTRACER keyword in the SCHEDULE section that defines the wells injecting the tracer.

Note

Currently, one cannot initialize tracers using the EQUALS keyword. Instead use the array format, that is the keyword followed by the required number of values, or the TVDP keyword in the SOLUTION section to set the initial tracer concentrations as a function of depth.

Example

The following TRACERS keyword in the RUNSPEC section declares the number of tracers in the model.

```
--
--       NUMBER AND TYPE OF TRACERS
--       NO OIL  NO WAT  NO GAS  NO ENV  DIFF    MAX    MIN    TRACER
--       TRACERS TRACERS TRACERS TRACERS CONTL   NONLIN NONLIN NONLIN
TRACERS
         0       0       1       0      'NODIFF' 1*     1*     1*             /
```

And the TRACER keyword in the PROPS section declares the tracer name and the phase for the tracer.

```
--
--       DEFINE TRACER NAMES
--
--       TRACER   TRACER
--       NAME     PHASE
--       ------   ------
TRACER
        'IGS'     'GAS'                                    / INJECTED GAS
/
```

Finally, the TBLK keyword in the SOLUTION section sets the initial tracer grid block concentrations in both the free and solution states.

```
--
--       INITIAL TRACER CONCENTRATIONS
--
TBLKFIGS
         1000*0.0                      / TRACER FIGS CONCENTRATIONS

TBLKSIGS
         1000*0.0                      / TRACER SIGS CONCENTRATIONS
```

Here the initial concentrations are set to zero.

Then in the SCHEDULE section one can us the WTRACER keyword to define the well injecting the tracer and the tracer concentration being injected,.

```
--
--       DEFINE CONCENTRATION OF TRACERS IN THE INJECTION STREAMS,
--       INJECTION TRACER CONCENTRATIONS NOT DEFINED USING THE WTRACER
--       KEYWORD ARE ASSUMED TO BE ZERO.
--
-- WELL  NAME    TRACER  TRACER  TRACER
-- NAME  TRACER  VALUE   CUM     GROUP
WTRACER
'GI01'   'IGS'   1.0                                                  /
/
```

In this case, well GI01 is a gas injection well injecting gas with a tracer concentration of 1.0. The example shows how to track dry gas injection in a gas condensate reservoir, although, the example can be used for any type of gas injection.

10.3.116 TEMPI – DEFINE THE INITIAL TEMPERATURE VALUES FOR ALL CELLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword defines the initial reservoir temperature for each cell in the model. The keyword is used to explicitly define the initial reservoir temperature via the Enumeration Initialization method rather than defining a uniform initial temperature or defining temperature versus depth tables.

The initial reservoir temperature must be defined when OPM Flow's thermal option has been activated by the THERMAL keyword in the RUNSPEC section. Note this is different to the commercial simulator that uses the TEMP keyword in the RUNSPEC section to activate the black-oil temperature model, and the THERMAL keyword to activate the compositional thermal model.

The initial reservoir temperature should be defined when OPM Flow's CO_2 or H_2 storage option has been activated by the CO2STORE or H2STORE keyword in the RUNSPEC section.

The keyword can be used with all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	TEMPI	TEMPI is an array of real positive numbers assigning the initial temperature to each cell in the model. Repeat counts may be used, for example 20*100.0.			
		°F	°C	°C	None

Notes:

1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
2) The keyword is terminated by a "/".

Table 10.58: TEMPI Keyword Description

See also the RTEMP keyword in the PROPS section and the RTEMPVD keyword in the SOLUTION section for alternative ways to initialize the model's initial temperature.

Example

```
--
--       DEFINE GRID BLOCK TEMPERATURE FOR ALL CELLS
--       (BASED ON NX x NY x NZ = 300)
--
TEMPI
         100*212.0   100*215.0   100*220.0                                  /
```

The above example defines the initial temperature to be 212.0, 215.0, and 220.0 oF for the first, second and third layers in the model for all 300 cells, as defined by the DIMENS keyword in the RUNSPEC section.

10.3.117 TEMPVD - DEFINE THE INITIAL RESERVOIR TEMPERATURE VERSUS DEPTH TABLES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword defines the initial reservoir temperature versus depth tables for each equilibration region. Note that the TEMPVD keyword is an alias for RTEMPVD, and that both keywords are supported by OPM Flow, in both the PROPS and SOLUTION sections, but are treated as being mutually exclusive.

The initial reservoir temperature must be defined when OPM Flow's thermal option has been activated by the THERMAL keyword in the RUNSPEC section. Note this is different to the commercial simulator that uses the TEMP keyword in the RUNSPEC section to activate the black-oil temperature model, and the THERMAL keyword to activate the compositional thermal model.

The initial reservoir temperature should be defined when OPM Flow's CO2 or H2 storage option has been activated by the CO2STORE or H2STORE keyword in the RUNSPEC section.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	DEPTH	A columnar vector of real monotonically increasing down the column values that defines the depth for corresponding reservoir temperature parameter RTEMP.			
		feet	m	cm	None
2	RTEMP	A columnar vector of real monotonically increasing down the column values that defines the corresponding reservoir temperature for the given depth.			
		°F	°C	°C	None

Notes:

1) The keyword is followed by NTEQUL tables as declared on the EQLDIMS keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NDRXVD rows as declared on the EQLDIMS keyword in the RUNSPEC section.
3) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 10.59: TEMPVD Keyword Description

See also the RTEMP keyword in the PROPS section for an alternative way to define a uniform initial reservoir temperature.

Example

```
--
--       INITIAL RESERVOIR TEMPERATURE VERSUS DEPTH TABLE
--
TEMPVD
--       DEPTH    TEMPERATURE
--       FEET     DEG F
--       ------   ----------
         1000.0     90.000
         2000.0    100.000
         3000.0    130.000
         4000.0    160.000                                  / TABLE N0. 01
--       ------   ----------
         1000.0     90.000
         2000.0    100.000
         3000.0    130.000
         4000.0    160.000                                  / TABLE N0. 02
--       ------   ----------
         1000.0     90.000
         2000.0    100.000
         3000.0    130.000
         4000.0    160.000                                  / TABLE N0. 03
```

The above example defines three identical reservoir depth versus temperature tables for the three NTEQUL regions defined on the EQLDIMS keyword in the RUNSPEC section.

10.3.118 THPRES - DEFINE EQUILIBRATION REGION THRESHOLD PRESSURES

Description

The THPRES defines the threshold pressure between various equilibration regions that have been defined by the EQLNUM keyword in the REGIONS section. The threshold pressure defines the potential difference between two regions which must be exceeded before flow can occur between the two regions. Once flow occurs the potential between the two regions is reduced by the threshold pressure.

This option must be activated by THPRES variable on EQLOPTS keyword in the RUNSPEC section in order to utilize this feature. Note that the irreversible option, as defined by IRREVER variable on EQLOPTS keyword in the RUNSPEC section, is not supported.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	EQLNUM1	EQLNUM1 is an a positive integer that is greater or equal to one and less than or equal to NTEQUL on the EQLDIMS keyword in the RUNSPEC section, that defines the "from" equilibration region number.			
		dimensionless	dimensionless	dimensionless	None
2	EQLNUM2	EQLNUM1 is an a positive integer that is greater or equal to one and less than or equal to NTEQUL on the EQLDIMS keyword in the RUNSPEC section, that defines the "to" equilibration region number.			
		dimensionless	dimensionless	dimensionless	None
3	THPRES	THPRES defines the threshold pressure from EQLNUM1 to EQLNUM2 and from EQLNUM2 to EQLNUM1. The default value of 1* sets the threshold pressure to a value that initially prevents flow between the two equilibration regions. Any subsequent production or injection in either of the two equilibration regions will therefore result in flow between the two regions. Thus, this default initially isolates the two equilibration regions. If a equilibration region number pair has not been explicitly defined by this keyword the THPRES is set to zero, for no threshold pressure.			
		psia	barsa	atma	1*

Notes:

1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword is terminated by a "/".

Table 10.60: THPRES Keyword Description

See also the MULTREGT keyword in the GRID section that uses the transmissibility between the MULTNUM, FLUXNUM or OPERNUM region arrays to control the flow between various regions within the model.

Note

Care should be taken that cells in different EQLNUM regions are not in communication, as this will result in an unstable initial equilibration.

Examples

Given NTEQUL is equal to six on the EQLDIMS keyword in the RUNSPEC section,

```
--
--        EQLNUM  EQLNUM  THPRES
--        FROM    TO      VALUE
THPRES
          1       2       0.588031                    / REGN 1 TO REGN 2
          2       1       0.588031                    / REGN 2 TO REGN 1
          1       3       0.787619                    / REGN 1 TO REGN 3
          3       1       0.787619                    / REGN 3 TO REGN 1
          1       4       7.000830                    / REGN 1 TO REGN 4
          4       1       7.000830                    / REGN 4 TO REGN 1
/
```

The above example defines the threshold pressures between equilibration regions one and two, one and three and one and four. As the threshold pressures between regions one and five and one and six (as well as other combinations), have not been explicitly set in the example, the threshold pressures for these combinations are set to zero.

However, as the irreversible option, as defined by IRREVER variable on EQLOPTS keyword in the RUNSPEC section, is not supported, then example can be simplified to:

```
--
--        EQLNUM  EQLNUM  THPRES
--        FROM    TO      VALUE
THPRES
          1       2       0.588031                    / REGN 1 AND REGN 2
          1       3       0.787619                    / REGN 1 AND REGN 3
          1       4       7.000830                    / REGN 1 AND REGN 4
/
```

Again, as the threshold pressures between regions one and five and one and six (as well as other combinations), have not been explicitly set in the example, the threshold pressures for these combinations are set to zero.

10.3.119 TVDP – DEFINE THE INITIAL EQUILIBRATION TRACER SATURATION VERSUS DEPTH FUNCTIONS

Description

This keyword defines the initial equilibration tracer concentration versus depth functions for each grid cell in the model, for when the Tracer option has been enabled by the TRACERS keyword in the RUNSPEC section. The maximum number of tracers for each phase are declared on the TRACERS keyword in the RUNSPEC section. Unlike other keywords, the TVDP keyword must be concatenated with the name of the tracer declared by the TRACER keyword in the PROPS section as outlined in Table 10.61.

No.	Name	Description	Default
1	TVDP	A four letter character string equal to TVDP that is the root keyword name for this data set function.	None
2	STATE	A one letter character string defining the tracer's state, which is concatenated to TVDP to give the partial name of the keyword. STATE should be set to F ("free") or S ("solution"), with the latter related to a tracer that is dissolved in an associated phase. If the tracer defined by NAME is associated with the OIL, GAS or WATER phases then STATE should be set to F. If the tracer is associated with the DISGAS or VAPOIL phases then STATE should be set to S.	None
3	NAME	A three letter character string defining the tracer's name as defined by the TRACER keyword, which is concatenated to TVDP and STATE to given the full name of the keyword. For example, if the TRACER keyword has been used to define a tracer named SEA in the WAT phase, then the full keyword name would be TVDPFSEA. Whereas, if a tracer was defined as IGS for the GAS phase then it could be TVDPFIGS and/or TVDPSIGS depending if one was wishing to track the "free" or "solution" gas phase, or both. Note it is best to avoid names beginning with the letters F, S, and T as these names may create naming issues in post-processing software.	None

Table 10.61: TVDP Keyword Name Format

Following the declaration of the full keyword name, TVDPSTATENAME, the keyword is followed by the data as outlined below.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	DEPTH	A columnar vector of real monotonically increasing down the column values that defines the depth values for the corresponding initial tracer saturations, TVDP			
		feet	m	cm	None
2	TVDPVAL	A columnar vector of real values, greater than or equal to zero, that defines the initial tracer concentration values at the corresponding DEPTH. If tracer units have been defined by the UNITS parameter on the TRACER keyword in the PROPS section, then the units of TVDPVAL are the ratio of UNITS divided by the TVDPVAL units given below. For example, if UNITS was defined as kg, then for metric units TVDPVAL units would be kg/sm^3.			None

No.	Name	Description			Default
		Field	Metric	Laboratory	
		Liquid: stb Gas: Mscf	Liquid: sm^3 Gas: sm^3	Liquid: scc Gas: scc	

Notes:

1) The keyword is followed by NTTRVD records as declared on the EQLDIMS keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NSTRVD rows as declared on the EQLDIMS keyword in the RUNSPEC section.
3) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 10.62: TVDP Keyword Description

See also the TBLK keyword in the SOLUTION section that also sets the initial tracer concentration for each grid block.

Example

This example is taken from Norne model, in which there are seven tracers related to the water phase.

```
-- ==================================================================================
--
-- RUNSPEC SECTION
--
-- ==================================================================================
RUNSPEC
--
--        MAX     MAX     RSVD    TVDP    TVDP
--        EQLNUM  DEPTH   NODES   TABLE   NODES
EQLDIMS
          9       1*      20      7       1*                                      /
-- ==================================================================================
--
-- PROPS SECTION
--
-- ==================================================================================
PROPS
--
--        DEFINE TRACER NAMES
--
--        TRACER   TRACER
--        NAME     PHASE
--        ------   ------
TRACER
          SEA      WAT                                        /
          HTO      WAT                                        /
          S36      WAT                                        /
          2FB      WAT                                        /
          4FB      WAT                                        /
          DFB      WAT                                        /
          TFB      WAT                                        /
/
```

```
-- ================================================================================
--
-- SOLUTION SECTION
--
-- ================================================================================
SOLUTION
--
--       INITIAL EQUILIBRATION TRACER SATURATION VERSUS DEPTH
--
--       DEPTH    TRACER
--                CONCENT
--       ------   --------
TVDPFSEA
         1000.0   0.0
         5000.0   0.0                        / TRACER FSEA CONCENTRATION VS DEPTH

TVDPFHTO
         1000.0   0.0
         5000.0   0.0                        / TRACER FHTO CONCENTRATION VS DEPTH

TVDPFS36
         1000.0   0.0
         5000.0   0.0                        / TRACER FS36 CONCENTRATION VS DEPTH

TVDPF2FB
         1000.0   0.0
         5000.0   0.0                        / TRACER F2FB CONCENTRATION VS DEPTH

TVDPF4FB
         1000.0   0.0
         5000.0   0.0                        / TRACER F4FB CONCENTRATION VS DEPTH

TVDPFDFB
         1000.0   0.0
         5000.0   0.0                        / TRACER FDFB CONCENTRATION VS DEPTH

TVDPFTFB
         1000.0   0.0
         5000.0   0.0                        / TRACER FTFB CONCENTRATION VS DEPTH
```

Here we first define the number of tracers in the model via the EQLDIMS keyword in the RUNSPEC section, then the actual tracers themselves in the PROPS section using the TRACER keyword, and finally the initial tracer concentrations are all set to zero via the TVDP keyword in the PROPS section.

10.3.120 VAPPARS – OIL VAPORIZATION PARAMETERS

Description

VAPPARS defines the rate of oil vaporization in the presence of undersaturated gas and the rate at which the remaining oil gets "heavier" via the reduction in the solution gas-oil ratio ("Rs"). This keyword should only be used if the OIL, GAS, DISGAS and VAPOIL keywords in the RUNSPEC section have been invoked to allow oil, gas, dissolved gas and vaporized oil to be present in the model.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	VAPPAR1	VAPPAR1 is a real positive dimensionless number that defines the rate at which oil vaporizes into the available undersaturated gas in a grid block. The default value of zero invokes the standard black-oil formulation in which all oil vaporizes into the available undersaturated phase in a grid cell. Increasing this parameter decreases the rate of vaporization. Typical values for VAPPAR1 range from zero to five.			
		dimensionless	dimensionless	dimensionless	0
2	VAPPAR2	VAPPAR2 is a real positive dimensionless number that defines the rate at which the Rs of the remaining oil in a grid cell decreases The default value of zero invokes the standard black-oil formulation in which the remaining oil's Rs does not change as the oil vaporizes into the available undersaturated gas in a grid cell. Increasing this parameter increases the difference between the remaining oil and the vaporized oil Rs values. Typical values for VAPPAR2 are less than one.			
		dimensionless	dimensionless	dimensionless	0
Notes: 1) The keyword is terminated by a "/".					

Table 10.63: VAPPARS Keyword Description

Note this keyword is normally used in history matching field performance to control the availability of the vaporized oil phase.

See also the DRSDT and DRVDT keywords in the SCHEDULE section that control the rate at which the solution gas-oil ratio and the vaporized oil-gas ratio increase within a grid block, respectively.

Examples

The first example sets the black-oil default parameters

```
--
--        OIL VAPORIZATION PARAMETERS
--
--        OIL-VAP    RS-INCS
--        VAPPAR1    VAPPAR2
VAPPARS
          0          0                                        /
```

And the second example decreases the rate at which the oil vaporizes into the available undersaturated gas and increases the difference between the grid block oil saturation Rs and the vaporized oil Rs within a grid cell.

```
--
--        OIL VAPORIZATION PARAMETERS
--
--        OIL-VAP    RS-INCS
--        VAPPAR1    VAPPAR2
VAPPARS
          1.5        0.150                                    /
```

Again, the keyword is normally used in history matching field performance to control the availability of the vaporized oil phase.

10.3.121 VISDATES – DEFINE EXTERNAL RESERVOIR GEO-MECHANICS VISAGE STRESS DATES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The VISDATES keyword defines External Reservoir Geo-Mechanics VISAGE option stress dates. The keyword should not be used in input decks as the associated data is generated by an external program.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

10.3.122 VISOPTS – DEFINE EXTERNAL RESERVOIR GEO-MECHANICS VISAGE OPTIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The VISDATES keyword defines External Reservoir Geo-Mechanics VISAGE option modeling options. The keyword should not be used in input decks as the associated data is generated by an external program.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

10.3.123 WARN – ACTIVATE WARNING MESSAGES

Turns on warning messages to be printed to the print file (*.PRT); note that this keyword is activated by default and can subsequently be switched off by the NOWARN activation keyword. The warning messages may be turned on and off using keywords WARN and NOWARN. OPM Flow always prints error messages.

See *WARN – Activate Warning Messages* in the GLOBAL section for a full description.

CHAPTER 11: SUMMARY SECTION

11.1 INTRODUCTION

The SUMMARY section defines the variables to be written to the summary files that are used to generate line graphs of properties such as oil flow rate versus time, grid block pressure versus time, etc. Unlike the other sections, the SUMMARY section has basically two types of keywords, keywords that request a variable to be written out to the summary file as described in section *11.2 Data Requirements*, and the conventional type of keyword described in section *11.3 Keyword Definitions* that perform an action. There are literally hundreds of keywords associated with the former indicating the variety of data that can be written to the summary file. Whereas, for the latter there are less than twenty keywords.

Summary file output consists of three files as outlined Table 11.1, with the raw requested data stored in SUMMARY index and SUMMARY data files, the SUMMARY RSM file is an ASCII file that is generated from the other two files by the simulator at the end of the run.

File Type	Data Type	Description	OPM Flow Status
ESMRY	Dynamic Vector Data	This is an OPM FLOW specific Enhanced SUMMARY (*.ESMRY) output file format that is optimized for fast loading of selected vectors by post-processing applications. The *.ESMRY file is re-written for every time step and does not have a performance impact on the simulation. Load times are in the order of 30 times faster compared with fetching selected vectors using the *.UNSMRY file format in post processing software. The file type is compatible with OPM ResInsight and the option is activated via the enable-esmry=true command line option. Note that the FMTOUT, MULTOUT, and UNIFOUT keywords have no impact on this file type.	OPM Flow Fully Supported.
RSM	Output File	The RSM file contains the variables requested to be written to the SUMMARY file via the keywords described in the *11.2 Data Requirements*, in a tabulated ASCII output format. The format of the file enables the data to be easily loaded into the LibreOffice Calc program for further processing, as each vector represents one column. The report is written at the end of the simulation run by parsing the SUMMARY Index and SUMMARY Data files.	Fully Supported
SUMMARY Index	Dynamic Vector Index	The SUMMARY index file type specifies and defines the format and data type written to the SUMMARY Data file.	Fully Supported
SUMMARY Data	Dynamic Vector Data	The SUMMARY data file contains the variables requested to be written to the file via the keywords described in the *11.2 Data Requirements*. The data consists of vectors that are used to generate line graphs of properties such as oil flow rate versus time, grid block pressure versus time, etc. The properties to be stored on the SUMMARY file are written to the summary file at the end of each successful time step. The data can be used to compare actual production data with the simulation derived results in post processing software like OPM ResInsight.	Fully Supported

Notes:

1) If no summary data is requested then the files will not be created.
2) The SUMMARY files can be loaded or written out in either unified or non-unified formats as well as in binary or ASCII formats.

Table 11.1: OPM Flow Summary Output File Summary

As mentioned in Table 11.1 all files can be written out in either ASCII or binary formats and in addition the SUMMARY file can be loaded or written out in either unified or non-unified formats. The file type (ASCII or binary) and file structure format (unified or non-unified formats) is set via a set of keywords in the RUNSPEC section, as described in Table 11.2 for easy of reference.

Process	RUNSPEC Keyword	Description	Files
Input	FMTIN	The keyword defines the input files to be formatted as ASCII i.e. text files, as oppose to binary files.	*.FSMSPEC *.FUNSMRY
	MULTIN	The keyword defines the input files to be non-unified multiple files, as opposed to unified files.	*.SMSPEC *.S0001
	UNIFIN	UNIFIN defines the input files to be unified files, as opposed to non-unified multiple files.	*.SMSPEC *.UNSMRY
Output	FMTOUT	The keyword sets all output files to be formatted as ASCII i.e. text files, as opposed to binary files.	*.FSMSPEC *.FUNSMRY
	MULTOUT	MULTOUT defines the output files to be non-unified multiple files, as opposed to unified files.	*.SMSPEC *.S0001
	UNIFOUT	UNIFOUT defines the output files to be unified files, as opposed to non-unified multiple files.	*.SMSPEC *.UNSMRY
Notes: 1) A binary file is computer-readable but not human-readable. 2) For unified files if the run terminates unexpectedly, or there is insufficient disk space, then the last reported output is not stored. The main advantage of unified files is that if a number of simulation cases reside in one directory, the number of output files per case is minimum compared to using non-unified multiple files. There is no limit on the number of reporting steps that a unified file can store.			

Table 11.2: RUNSPEC Input and Output File Format Keywords

OPM Flow automatically generates the SUMMARY file names based on the input file name and the output options selected via the keywords in the RUNSPEC section as summarized in in Table 11.2. For example, starting OPM Flow using the following command from the terminal:

```
flow CASENAME
```

will start the simulator and run the case specified by CASENAME.DATA. Here CASENAME is the "root" of the filename and DATA is the extension of the filename. OPM Flow will generate the SUMMARY output files based on the CASENAME with the extension based on the type (ASCII or binary) and file structure format (unified or non-unified formats), as outlined in Table 11.3.

File Type	Unformatted Extension	Description	Formatted Extension
RSM	*.RSM	Output RSM file.	*.RSM

File Type	Unformatted Extension	Description	Formatted Extension
SUMMARY Index	*.SMSPEC	The SUMMARY index file for both unified and non-unified formats.	*.FSMSPEC
SUMMARY Data Non-Unified	*.Snnnn	The SUMMARY data files containing the variables requested to be written to the SUMMARY files. For example, the SUMMARY non-unified binary files would be: CASENAME.S0001 CASENAME.S0002 CASENAME.S0003 etc. CASENAME.SMSPEC	*.Annnn
SUMMARY Data Unified	*.UNSMRY	The SUMMARY data file containing the variables requested to be written to the SUMMARY file. For example, the SUMMARY data unified binary files: CASENAME.UNSMRY CASENAME.SMSPEC	*.FUNSMRY
Notes: 1) The above file naming convention is for Linux type operating systems, as OPM Flow is currently only officially supported for Linux distributions.			

Table 11.3: OPM Flow File Naming Conventions

The default behavior is write out the requested variables at each time step. As this can lead to large files, especially for full field simulation models, the *RPTONLY - Activate the Report Time Steps Only Option for the SUMMARY File* keyword allows one to write out the data only at a report time step instead.

The SUMMARY section is terminated by the SCHEDULE keyword.

11.2 DATA REQUIREMENTS

To generate a summary vector, the appropriate keyword must be entered in the SUMMARY section of the input data file; only variables explicitly requested will be written to the SUMMARY files. Normally the data is written to the summary file at the end of each successful time step, but as mentioned previously, this can be changed by the *RPTONLY - Activate the Report Time Steps Only Option for the SUMMARY File* keyword to write out the data only at report time steps, thus reducing the overall size of the SUMMARY files.

The following sections describe the summary variable mnemonic syntax which defines the type of summary variable object (Aquifer, Field, Group, Well, etc.), the variable format for a given variable object, and the variable names for the various variable objects.

11.2.1 SUMMARY VARIABLE MNEMONIC SYNTAX

As mentioned earlier, there are literally hundreds of variables that can be written to the SUMMARY file, which can make the process of requesting the data rather complex. Fortunately in most cases, but not always, the variable names follow a four or five letter syntax that defines the variable mnemonic used to describe the data to be written out. Table 11.4 outlines the general syntax employed in deriving the variable mnemonic.

Summary Variable Mnemonic Syntax									
First Character Object		Second Character Object Type		Third Character Flow Type		Four Character Value Type		Fifth Character Special	
A	**Aquifers**	A	Analytical Aquifer	Q	Influx	R			
		LI	Analytical Aquifer List	L	Liquid influx	T			
		N	Numerical Aquifer						
B	**Block (Grid Cell)**	O	Oil	P	Production	R	Rate	L	Liquid
F	**Field**	W	Water	I	Injection	T	Total (cumulative)	H	History
G	**Group**	G	Gas	F	Flow			G	Gas
R	**Region**	V	Volume at reservoir conditions						
W	**Well**	L	Liquid						
C	**Well Connection**	P2	Pressure						
S	**Well Segment**	T	Tracer						

Summary Variable Mnemonic Syntax				
First Character Object	Second Character Object Type	Third Character Flow Type	Four Character Value Type	Fifth Character Special

Notes:

1) Analytical Aquifer Lists are currently not supported by OPM Flow; however the standard analytical and numerical aquifer vectors are supported to a varying degree.
2) For Field and Region pressures the mnemonic is PR, that is FPR for the field reservoir pressure and RPR for region average pressure. Secondly, a well's bottom-hole pressure is WBHP and the tubing head pressure is WTHP.

Table 11.4: Summary Variable Mnemonic Syntax

So for example, FOPR would mean Field Oil Production Rate. Also if there is historical data associated with a variable, then the same variable mnemonic is used with the suffix H, that is for the historical field oil production rate the mnemonic would be FOPRH. Table 11.5 shows some examples of variable mnemonics that follow the general syntax.

Summary Variable Mnemonics Examples	
Name	Description
BOFT	Block Oil Flow Total
COFR	Connection Oil Flow Rate
FOPR	Field Oil Production Rate
GVPR	Group Volume Production Rate
RTIT	Region Tracer Injection Total
WWPT	Well Water Production Total

Table 11.5: Summary Variable Mnemonics Examples

And Table 11.6 shows some examples that do not follow the general syntax. Where applicable, each of these keywords would also have a history equivalent mnemonics available in history matching runs, that is: FPRH, FWCTH, WBHPH, WTHPH, WWCTH, etc.

Summary Variable Mnemonics Examples Not Following the General	
Name	Description
BAPI	Block API
BOSAT	Block Oil Saturation
BPR	Block Pressure
FAQR	Field Aquifer Influx Rate
FPR	Field Average Pressure
FWCT	Field Water Cut

Summary Variable Mnemonics Examples Not Following the General	
Name	**Description**
WBHP	Well Bottom-Hole Pressure
WTHP	Well Tubing Head Pressure
WWCT	Well Water Cut

Table 11.6: Summary Variable Mnemonics Examples Not Following the General Syntax

11.2.2 SUMMARY VARIABLE FORMAT

In addition to the general mnemonic syntax, each object type (Field, Group, Region, Well etc.) has additional syntax governing what specific object (group, well, etc.) should be written out to the SUMMARY file as explained in Table 11.7.

Summary Variable Format		
Summary Object	**Format**	**Example**

AQUIFER	Analytical aquifers are identified by the first two characters AA in the variable mnemonic and numerical aquifers are identified by the first two characters of AQ.
	Aquifer variables for a given aquifer can be followed by a list of aquifer numbers and therefore a terminating "/" is required to end the list of aquifers. A blank list requests output for all the aquifers.
		For instance to obtain the aquifer influx rate and cumulative influx for all analytical aquifers in the model, one would use:
		`AAQR`
		`/`
		`AAQT`
		`/`
		or just for numerical aquifers one, two and three:
		`ANQR`
		`1 2 3 /`
		`ANQR`
		`1 2 3 /`

	Analytical aquifers that have been allocated to an analytical aquifer list via the AQUALIST keyword in the SOLUTION section are identified by the first two characters AL in the variable mnemonic.
	Aquifer variables for a given aquifer list can be followed by a list of aquifer names and therefore a terminating "/" is required to end the list of aquifers. A blank list requests output for all the aquifers.
		For instance to obtain the aquifer influx rate and cumulative influx for all two analytical aquifers list, EAST and WEST in the model, one would use:
		`AAQR`
		`'EAST' 'WEST'/`
		`AAQT`
		`'EAST' 'WEST'/`
		Note OPM Flow does not currently support Analytical Aquifer Lists.

BLOCK
(Grid Cells)
	Block variables are followed by a list of cell (I, J, K) indices, with each line terminated by a "/"., and the list terminated with another terminating "/".	To request the oil saturation for a series of grid cells one would use the following:
		`BOSAT`
		`1 1 1 /`
		`10 10 3 /`
		`15 15 10 /`
		`/`

FIELD	Field variables take no additional parameters and therefore do not require a terminating "/".	For example:
		`FOPR`
		`FOPT`
		`FPR`
		Would export the field oil rate and total, plus the field average reservoir pressure to the SUMMARY file.

Summary Variable Format		
Summary Object	**Format**	**Example**

GROUP	Group variables can be followed by a list of group names enclosed in quotes and therefore a terminating "/" is required to end the list of groups. A blank list requests output for all groups.	As an example to get the oil production rates and totals for two groups representing two platforms one would use
		`GOPR`
		`'PLAT1' 'PLAT2' /`
		
		`GOPT`
		`'PLAT1' 'PLAT2' /`
		or
		`GOPR`
		`/`
		`GOPT`
		`/`
		if there are only two groups in the model.

REGION	Region variables for a given region can be followed by a list of region numbers and therefore a terminating "/" is required to end the list of regions. A blank list requests output for all the regions.	For instance to obtain the reservoir pressures and oil totals for all regions:
		`RPR`
		`/`
		`ROPT`
		`/`
		or just for regions one, two and three:
		`RPR`
		`1 2 3 /`
		`ROPT`
		`1 2 3 /`

	Region variables that are based on flows between regions must be followed by a list of region pair numbers, indicating the two regions. Each region pair is terminated by a "/". In addition the variable mnemonic should also be terminated by a terminating "/".	To obtain the oil and water flow between regions one and two, and also three and four, one would use:
		`ROFT`
		`1 2 /`
		`3 4 /`
		`/`
		`RWFT`
		`1 2 /`
		`3 4 /`
		`/`

WELL	Well variables can be followed by a list of well names enclosed in quotes and therefore a terminating "/" is required to end the list of wells. A blank list requests output for all wells.	As an example to get the oil production rates and totals for all oil wells beginning with the letters OP one would use
		`WOPR`
		`'OP*' /`
		
		`WOPT`
		`'OP*' /`
		To output the oil production rate for all wells, one may use:
		`WOPR`
		`/`
		This would also write out the oil production rate for all well types: producers and injectors, as well as wells declared as gas and water wells.

Summary Variable Format		
Summary Object	**Format**	**Example**

WELL COMPLETIONS	Well variables for well completion vectors[293], generally consist of the well mnemonic name suffixed with the letter "L", for example WOPRL, for the completion oil production rate.
	The completion mnemonic is then followed by a list of well names enclosed in quotes and the completion number, each terminated by a "/". In addition the list is terminated by a terminating "/".
	A blank list for the completions results in all completions for a well being written out. And a blank list for the well will result in all well completions being written to the SUMMARY file.
	Care should be exercised when defaulting the list of wells and completions as there is the potential to generate large volumes of data.
		For example, to get the completion oil production rates for well OP01 completions two, three, and four, and for completion two for OP02 one would use:
		`WOPRL`
		`'OP01' 2 /`
		`'OP01' 3 /`
		`'OP01' 4 /`
		`'OP02' 2 /`
		`/`
		or to get all the oil production rates for all the completions for well OP01:
		`WORPL`
		`'OP01' /`
		`/`

WELL CONNECTIONS	Connection variables are followed by a list of well names in quotes and completion I, J, K indices, each terminated by a "/". In addition the list is terminated by a terminating "/".
	A blank list for the connections results in all connections for a well being written out. And a blank list for the well will result in all well connections being written to the SUMMARY file.
	Again, care should be exercised when defaulting the list of wells and connections as there is the potential to generate large volumes of data.
		To request the gas-oil ratio and water cut for the OP01 and OP02 wells for several connections, one would use:
		`CGOR`
		`OP01 1 1 1 /`
		`OP01 1 1 2 /`
		`OP02 10 10 1 /`
		`/`
		
		`CWCT`
		`OP01 1 1 1 /`
		`OP01 1 1 2 /`
		`OP02 10 10 1 /`
		`/`

[293] *The term well connection is used to describe individual connections from the wellbore to the reservoir grid, as opposed to well completions. A well completion is used to describe a set of connections, for example, a well may consist of several completions with each completion consisting of multiple connections.*

Summary Variable Format		
Summary Object	**Format**	**Example**

WELL CONNECTIONS (COMPLETIONS)	This type of data is the same as for the Well Completion data, except the Well Connection series of mnemonics are used instead of the well mnemonics.
	Here the connection vectors consist of the connection mnemonic name suffixed with the letter "L", for example COPRL, for the completion oil production rate.
	The connection mnemonic is then followed by a list of well names enclosed in quotes and the connection number, each terminated by a "/". In addition, the list is terminated by a terminating "/". If any connection has been defined as being within a completion via the COMPLUMP keyword in the SCHEDULE section, then the completion data is written out instead of the connection data.
	A blank list for the completions results in all completions for a well being written out. However, the well name cannot be defaulted, unlike the other well and connection mnemonics.
	Care should be exercised when defaulting the list of wells and completions as there is the potential to generate large volumes of data.
		To request the gas-oil ratio and water cut for the OP01 and OP02 wells for several connections that have lumped together in a completion, one would use:
		`CGORL`
		`OP01 1 1 1 /`
		`OP01 1 1 2 /`
		`OP02 10 10 1 /`
		`/`
		
		`CWCTL`
		`OP01 1 1 1 /`
		`OP01 1 1 2 /`
		`OP02 10 10 1 /`
		`/`

WELL SEGMENTS	Well variables for multi-segment well segments can be followed by a list of well names enclosed in quotes and the segment number, each terminated by a "/". In addition, the list is terminated by a terminating "/".
	A blank list for the segments results in all segments for a well being written out. And a blank list for the well will result in all well segments being written to the SUMMARY file.
	Care should be exercised when defaulting the list of wells and segments as there is the potential to generate large volumes of data.
		For example, to get the segment oil production rates for well OP01 segments two, three, and four and for segment two for OP02 one would use:
		`SOFR`
		`'OP01' 2 /`
		`'OP01' 3 /`
		`'OP01' 4 /`
		`'OP02' 2 /`
		`/`
		or to get all the oil production rates for all the segments for well OP01:
		`SOFR`
		`'OP01' /`
		`/`

Table 11.7: Summary Variable Format

11.2.3 AQUIFER VARIABLES

Table 11.8 outlines the aquifer summary variables based on the type of aquifer: analytical, analytical list and numerical. Note that the analytical list aquifer type is just an analytical aquifer "set" defined using the AQULIST keyword in the SOLUTION section, that assigns an analytic aquifer name to a set of aquifer numbers for greater readability in the output. This type of aquifer is not supported by OPM Flow and neither are numerical aquifers.

Aquifer Summary Variables						
Variable	**Root**	**Field**	**Analytical Aquifer**	**Analytical Aquifer List**	**Numerical Aquifer**	**Comment**
Aquifer Influx Rate (Water Aquifers)	QR	FAQR	AAQR	ALQR	ANQR	
Aquifer Influx Total (Water Aquifers)	QT	FAQT	AAQT	ALQT	ANQT	
Aquifer Influx Rate (Gas Aquifers)	QRG	FAQRG	AAQRG	ALQRG		
Aquifer Influx Total (Gas Aquifers)	QTG	FAQTG	AAQTG	ALQTG		
Aquifer Pressure	QP		AAQP		ANQP	Water pore volume weighted average
Aquifer Carter-Tracy Dimensionless Pressure	QTD		AAQTD			
Aquifer Carter-Tracy Dimensionless Time	QPD		AAQPD			

Notes:

1) Cells colored in gray with a mnemonic indicate that the summary variable is not available in OPM Flow.
2) Cells colored in gray with no mnemonic indicate that the summary variable is not available for this combination.
3) Cells colored orange show combinations that are not available in OPM Flow because the underlying feature is not available, for example, numerical aquifers are currently not supported by OPM Flow.

Table 11.8: Aquifer Summary Variables

Example

The following example requests the field and analytical aquifer pressure, rate and cumulative water influxes for all analytical aquifers to be written to the SUMMARY file.

```
-- ==========================================================================
--
-- SUMMARY SECTION
--
-- ==========================================================================
SUMMARY
--
--       ANALYTICAL AQUIFERS (FIELD)
--
FAQR
FAQT
--
--       ANALYTICAL AQUIFERS
--
AAQP
/
AAQR
/
AAQT
/
```

11.2.4 FIELD, GROUP, WELL, WELL CONNECTION AND COMPLETION VARIABLES

The following table (Table 11.9) lists the field, group, well, well connection and well completion production, injection, pressure and productivity summary variables. Note that not all of these variables are available in OPM Flow; however, the simulator will issue a warning messages if this is indeed the case. It is anticipated that the number of recognized summary variables will increase in future releases of OPM Flow.

See also section *11.2.6 Field, Region and Block Variables* that outlines the field, region and block production and injection data summary variables, and section *11.2.7 Field and Region Recovery Variables* that describes the summary variables for oil recovery efficiency and the oil recovery mechanism.

Field, Group, Well, Well Connection and Completion Summary Variables

Type	Variable	Root	Field	Group	Well	Well Connection	Well Completion	Comment
Flow	Artificial Lift Quantity	ALQ			WALQ			
Flow	Artificial Lift and VFP Table Number	MVFP			WMVFP			Well VFP table number.
Flow	Efficiency Factor	EFF		GEFF	WEFF			
Flow	Efficiency Factor Product	EFFG			WEFFG			Efficiency factor of the well times the product of all the groups above the well.
Flow	Energy Production Rate	EPR	FEPR	GEPR	WEPR			
Flow	Energy Production Total	EPT	FEPT	GEPT	WEPT			
Flow	Gas Consumption Rate	GCR	FGCR	GGCR				
Flow	Gas Consumption Total	GCT	FGCT	GGCT				
Flow	Gas Flow Rate	GFR				CGFR	CGFRL	Positive for production and negative for injection
Flow	Gas Flow Rate (Free)	GFRF				CGFRF		
Flow	Gas Flow Rate (Solution)	GFRS				CGFRS		
Flow	Gas Flow Rate (Upstream)	GFRU				CGFRU		Sum of connection Gas flow rates upstream of, and including, this connection
Flow	Gas Import Rate	GIMR	FGIMR	GGIMR				
Flow	Gas Import Total	GIMT	FGIMT	GGIMT				
Flow	Gas Injection Guide Rate	GIGR		GGIGR	WGIGR			
Flow	Gas Injection Rate	GIR	FGIR	GGIR	WGIR	CGIR	CGIRL	

Field, Group, Well, Well Connection and Completion Summary Variables								
Type	Variable	Root	Field	Group	Well	Well Connection	Well Completion	Comment
Flow	Gas Injection Rate History	GIRH	FGIRH	GGIRH	WGIRH			
Flow	Gas Injection Rate Target/Limit	GIRT	FGIRT	GGIRT	WGIRT			
Flow	Gas Injection Total	GIT	FGIT	GGIT	WGIT	CGIT	CGITL	
Flow	Gas Injection Total History	GITH	FGITH	GGITH	WGITH			
Flow	Gas Lift Rate	GLIR	FGLIR	GGLIR	WGLIR			Gas lift gas with group and field total rates based on well efficiency
Flow	Gas Potential Injection Rate	GPI	FGPI	GGPI	WGPI	CGPI		Also WGIP for WGPI
Flow	Gas Potential Injection Rate	GPI2	FGPI2	GGPI2	WGPI2			Including shut and stopped wells.
Flow	Gas Potential Production Rate	GPP	FGPP	GGPP	WGPP	CGPP		
Flow	Gas Potential Production Rate	GPP2	FGPP2	GGPP2	WGPP2			Including shut and stopped wells.
Flow	Gas Potential Production Rate (Free)	GPPF	FGPPF	GGPPF	WGPPF			
Flow	Gas Potential Production Rate (Free)	GPPF2	FGPPF2	GGPPF2	WGPPF2			Including shut and stopped wells.
Flow	Gas Potential Production Rate (Soln)	GPPS	FGPPS	GGPPS	WGPPS			
Flow	Gas Potential Production Rate (Soln)	GPPS2	FGPPS2	GGPPS2	WGPPS2			Including shut and stopped wells.
Flow	Gas Production Guide Rate	GPGR		GGPGR	WGPGR			
Flow	Gas Production Rate	GPR	FGPR	GGPR	WGPR	CGPR		Produced reservoir gas only, gas lift gas excluded
Flow	Gas Production Rate (Free)	GPRF	FGPRF	GGPRF	WGPRF			
Flow	Gas Production Rate (Solution)	GPRS	FGPRS	GGPRS	WGPRS			
Flow	Gas Production Rate History	GPRH	FGPRH	GGPRH	WGPRH			
Flow	Gas Production Rate Target/Limit	GPRT	FGPRT	GGPRT	WGPRT			
Flow	Gas Production Total	GPT	FGPT	GGPT	WGPT	CGPT	CGPTL	
Flow	Gas Production Total (Free)	GPTF	FGPTF	GGPTF	WGPTF			
Flow	Gas Production Total (Solution)	GPTS	FGPTS	GGPTS	WGPTS			
Flow	Gas Production Total History	GPTH	FGPTH	GGPTH	WGPTH			

Field, Group, Well, Well Connection and Completion Summary Variables								
Type	Variable	Root	Field	Group	Well	Well Connection	Well Completion	Comment
Flow	Gas Sales Rate	GSR	FGSR	GGSR				
Flow	Gas Sales Total	GST	FGST	GGST				
Flow	Gas Voidage Injection Rate	GVIR			WGVIR			
Flow	Gas Voidage Production Rate	GVPR			WGVPR			
Flow	Liquid Flow Rate	LFR				CLFR	CLFRL	Positive for production and negative for injection
Flow	Liquid Production Rate	LPR	FLPR	GLPR	WLPR	CLPR		
Flow	Liquid Production Rate History	LPRH	FLPRH	GLPRH	WLPRH			
Flow	Liquid Production Rate Target/Limit	LPRT	FLPRT	GLPRT	WLPRT			
Flow	Liquid Production Total	LPT	FLPT	GLPT	WLPT	CLPT	CLPTL	
Flow	Liquid Production Total History	LPTH	FLPTH	GLPTH	WLPTH			
Flow	Oil Flow Rate	OFR				COFR	COFRL	Positive for production and negative for injection
Flow	Oil Flow Rate (Free)	OFRF				COFRF		
Flow	Oil Flow Rate (Solution)	OFRS				COFRS		
Flow	Oil Flow Rate (Upstream)	OFRU				COFRU		Sum of connection oil flow rates upstream of, and including, this connection
Flow	Oil Injection Guide Rate	OIGR		GOIGR	WOIGR			
Flow	Oil Injection Rate	OIR	FOIR	GOIR	WOIR	COIR	COIRL	
Flow	Oil Injection Rate History	OIRH	FOIRH	GOIRH	WOIRH			
Flow	Oil Injection Rate Target/Limit	OIRT	FOIRT	GOIRT	WOIRT			
Flow	Oil Injection Total	OIT	FOIT	GOIT	WOIT	COIT	COITL	
Flow	Oil Injection Total History	OITH	FOITH	GOITH	WOITH			
Flow	Oil Potential Injection Rate	OPI	FOPI	GOPI	WOPI			
Flow	Oil Potential Injection Rate	OPI2	FOPI2	GOPI2	WOPI2			Including shut and stopped wells.
Flow	Oil Potential Production Rate	OPP	FOPP	GOPP	WOPP	COPP		
Flow	Oil Potential Production Rate	OPP2	FOPP2	GOPP2	WOPP2			Including shut and stopped wells.
Flow	Oil Production Guide Rate	OPGR		GOPGR	WOPGR			

Field, Group, Well, Well Connection and Completion Summary Variables								
Type	Variable	Root	Field	Group	Well	Well Connection	Well Completion	Comment
Flow	Oil Production Rate	OPR	FOPR	GOPR	WOPR	COPR		
Flow	Oil Production Rate (Free)	OPRF	FOPRF	GOPRF	WOPRF			
Flow	Oil Production Rate (Solution)	OPRS	FOPRS	GOPRS	WOPRS			
Flow	Oil Production Rate History	OPRH	FOPRH	GOPRH	WOPRH			
Flow	Oil Production Rate Target/Limit	OPRT	FOPRT	GOPRT	WOPRT			
Flow	Oil Production Total	OPT	FOPT	GOPT	WOPT			
Flow	Oil Production Total (Free)	OPTF	FOPTF	GOPTF	WOPTF			
Flow	Oil Production Total (Solution)	OPTS	FOPTS	GOPTS	WOPTS			
Flow	Oil Production Total History	OPTH	FOPTH	GOPTH	WOPTH			
Flow	Res.Vol. Flow Rate	VFR				CVFR	CVFRL	Positive for production and negative for injection
Flow	Res.Vol. Flow Rate (Free)	VFRF				CVFRF		
Flow	Res.Vol. Injection Guide Rate	VIGR		GVIGR	WVIGR			
Flow	Res.Vol. Injection Rate	VIR	FVIR	GVIR	WVIR	CVIR	CVIRL	
Flow	Res.Vol. Injection Rate Target/Limit	VIRT	FVIRT	GVIRT	WVIRT			
Flow	Res.Vol. Injection Total	VIT	FVIT	GVIT	WVIT	CVIT	CVITL	
Flow	Res.Vol. Potential Injection Rate	VPI	FVPI	GVPI	WVPI			
Flow	Res.Vol. Potential Injection Rate	VPI2	FVPI2	GVPI2	WVPI2			Including shut and stopped wells.
Flow	Res.Vol. Potential Production Rate	VPP	FVPP	GVPP	WVPP	CVPP		
Flow	Res.Vol. Potential Production Rate	VPP2	FVPP2	GVPP2	WVPP2			Including shut and stopped wells.
Flow	Res.Vol. Production Guide Rate	VPGR		GVPGR	WVPGR			
Flow	Res.Vol. Production Guide Rate	VPGR		GVPGR	WVPGR			
Flow	Res.Vol. Production Rate	VPR	FVPR	GVPR	WVPR	CVPR		
Flow	Res.Vol. Production Rate Target/Limit	VPRT	FVPRT	GVPRT	WVPRT			
Flow	Res.Vol. Production Total	VPT	FVPT	GVPT	WVPT	CVPT		
Flow	Water Flow Rate	WFR				CWFR	CWFRL	Positive for production and negative for injection

Field, Group, Well, Well Connection and Completion Summary Variables								
Type	Variable	Root	Field	Group	Well	Well Connection	Well Completion	Comment
Flow	Water Flow Rate (Upstream)	WFRU				CWFRU		Sum of connection water flow rates upstream of, and including, this connection
Flow	Water Injection Guide Rate	WIGR		GWIGR	WWIGR			
Flow	Water Injection Rate	WIR	FWIR	GWIR	WWIR	CWIR	CWIRL	
Flow	Water Injection Rate History	WIRH	FWIRH	GWIRH	WWIRH			
Flow	Water Injection Rate Target/Limit	WIRT	FWIRT	GWIRT	WWIRT			
Flow	Water Injection Total	WIT	FWIT	GWIT	WWIT	CWIT	CWITL	
Flow	Water Injection Total History	WITH	FWITH	GWITH	WWITH			
Flow	Water Potential Injection Rate	WPI	FWPI	GWPI	WWPI			Also WWIP for WWPI
Flow	Water Potential Injection Rate	WPI2	FWPI2	GWPI2	WWPI2			Including shut and stopped wells.
Flow	Water Potential Production Rate	WPP	FWPP	GWPP	WWPP	CWPP		
Flow	Water Potential Production Rate	WPP2	FWPP2	GWPP2	WWPP2			Including shut and stopped wells.
Flow	Water Produce/Injected Ratio	WPIR	FWPIR	GWPIR	WWPIR			Reported as a percent.
Flow	Water Production Guide Rate	WPGR		GWPGR	WWPGR			
Flow	Water Production Rate	WPR	FWPR	GWPR	WWPR	CWPR		
Flow	Water Production Rate History	WPRH	FWPRH	GWPRH	WWPRH			
Flow	Water Production Rate Target/Limit	WPRT	FWPRT	GWPRT	WWPRT			
Flow	Water Production Total	WPT	FWPT	GWPT	WWPT	CWPT	CWPTL	
Flow	Water Production Total History	WPTH	FWPTH	GWPTH	WWPTH			
Flow	Water Voidage Injection Guide Rate	WVIR			WWVIR			
Flow	Water Voidage Injection Guide Total	WVIT			WWVIT			
No. Wells	Number of abandoned injection wells	MWIA	FMWIA	GMWIA				
No. Wells	Number of abandoned production wells	MWPA	FMWPA	GMWPA				
No. Wells	Number of drilling events in total	MWDT	FMWDT	GMWDT				
No. Wells	Number of drilling events this timestep	MWDR	FMWDR	GMWDR				

Field, Group, Well, Well Connection and Completion Summary Variables								
Type	Variable	Root	Field	Group	Well	Well Connection	Well Completion	Comment
No. Wells	Number of injection wells currently flowing	MWIN	FMWIN	GMWIN				
No. Wells	Number of injectors on group control	MWIG	FMWIG	GMWIG				
No. Wells	Number of injectors on own reservoir volume rate limit control	MWIV	FMWIV	GMWIV				
No. Wells	Number of injectors on own surface rate limit control	MWIS	FMWIS	GMWIS				
No. Wells	Number of injectors on pressure control	MWIP	FMWIP	GMWIP				
No. Wells	Number of producers controlled by own oil rate limit	MWPO	FMWPO	GMWPO				
No. Wells	Number of producers on group control	MWPG	FMWPG	GMWPG				
No. Wells	Number of producers on own reservoir volume rate limit control	MWPV	FMWPV	GMWPV				
No. Wells	Number of producers on own surface rate limit control	MWPS	FMWPS	GMWPS				
No. Wells	Number of producers on pressure control	MWPP	FMWPP	GMWPP				
No. Wells	Number of producers using artificial lift (with ALQ > 0.0)	MWPL	FMWPL	GMWPL				
No. Wells	Number of production wells currently flowing	MWPR	FMWPR	GMWPR				
No. Wells	Number of unused injection wells	MWIU	FMWIU	GMWIU				
No. Wells	Number of unused production wells	MWPU	FMWPU	GMWPU				
No. Wells	Number of workover events in total	MWWT	FMWWT	GMWWT				
No. Wells	Number of workover events this time step.	MWWO	FMWWO	GMWWO				
No. Wells	The number of connections capable of flowing in the well	MCON			WMCON			
No. Wells	Total number of injection wells	MWIT	FMWIT	GMWIT				
No. Wells	Total number of production wells	MWPT	FMWPT	GMWPT				
Pressure	Blocking Factor (GPP)	CDBF				CCDBF		
Pressure	Bottom-Hole Pressure	BHP			WBHP			
Pressure	Bottom-Hole Pressure (Five Point	BP5			WBP5			
Pressure	Bottom-Hole Pressure (Four Point)	BP4			WBP4			
Pressure	Bottom-Hole Pressure (Nine Point)	BP9			WBP9			
Pressure	Bottom-Hole Pressure (One Point)	BP			WBP			

Field, Group, Well, Well Connection and Completion Summary Variables								
Type	Variable	Root	Field	Group	Well	Well Connection	Well Completion	Comment
Pressure	Bottom-Hole Pressure History	BHPH			WBHPH			
Pressure	Bottom-Hole Pressure Target/Limit	BHPT			WBHPT			
Pressure	Connection Pressure	PR				CPR	CPRL	Completion pressure is an average pressure for the completion
Pressure	Connection Transmissibility Factor	TFAC				CTFAC		
Pressure	Productivity (Gas Phase)	PIG			WPIG			
Pressure	Productivity (Liquid Phase)	PIL			WPIL			
Pressure	Productivity (Oil Phase)	PIO			WPIO			
Pressure	Productivity (Preferred Phase – Five Point)	PI5			WPI5			
Pressure	Productivity (Preferred Phase – Four Point)	PI4			WPI4			
Pressure	Productivity (Preferred Phase – Nine Point)	PI9			WPI9			
Pressure	Productivity (Preferred Phase – One Point)	PI1			WPI1			
Pressure	Productivity (Preferred Phase)	PI			WPI	CPI		
Pressure	Productivity (Water Phase)	PIW			WPIW			
Pressure	Tubing Head Pressure	THP			WTHP			
Pressure	Tubing Head Pressure History	THPH			WTHPH			
Ratio	Gas-Liquid Ratio	GLR	FGLR	GGLR	WGLR	CGLR	CGLRL	
Ratio	Gas-Liquid Ratio (Bottom-Hole)	BGLR			WBGLR			
Ratio	Gas-Liquid Ratio History	GLRH	FGLRH	GGLRH	WGLRH			
Ratio	Gas-Oil Ratio	GOR	FGOR	GGOR	WGOR	CGOR	CGORL	
Ratio	Gas-Oil Ratio History	GORH	FGORH	GGORH	WGORH			
Ratio	Oil-Gas Ratio	OGR	FOGR	GOGR	WOGR	COGR	COGRL	
Ratio	Oil-Gas Ratio History	OGRH	FOGRH	GOGRH	WOGRH			
Ratio	Water -Gas Ratio	WGR	FWGR	GWGR	WWGR	CWGR	CWGRL	
Ratio	Water Cut	WCT	FWCT	GWCT	WWCT	CWCT	CWCTL	
Ratio	Water Cut History	WCTH	FWCTH	GWCTH	WWCTH			
Ratio	Water-Gas Ratio History	WGRH	FWGRH	GWGRH	WWGRH			

Field, Group, Well, Well Connection and Completion Summary Variables								
Type	Variable	Root	Field	Group	Well	Well Connection	Well Completion	Comment

Notes:

1) Cells colored in gray with a mnemonic indicate that the summary variable is not available in OPM Flow.
2) Cells colored in gray with no mnemonic indicate that the summary variable is not available for this combination.
3) Cells colored orange show combinations that are not available in OPM Flow because the underlying feature is not available.

Table 11.9: Field, Group, Well, Well Connection and Completion Summary Variables

11.2.5 FIELD, GROUP AND WELL CONTROL MODE VARIABLES

In addition to the production, injection, pressure and productivity summary variables, there are also summary variables that output the control mode at which the field, groups and wells are being controlled. Table 11.10 summarizes the mnemonics for the field and groups together with a description and the values written out to the SUMMARY file.

Field and Group Control Mode Reporting						
Object	**Field**	**Group**	**Field**	**Group**	**Field**	**Group**
Mnemonic	FMCTP	GMCTP	FMCTW	GMCTW	FMCTG	GMCTG
Description	Production Group.		Water Injection Group.		Gas Injection Group.	
Values	**Value**	**Mnemonic**	**Value**	**Mnemonic**	**Value**	**Mnemonic**
	0.0	NONE	0.0	NONE	0.0	NONE
	1.0	ORAT	1.0	RATE	1.0	RATE
	2.0	WRAT	2.0	RESV	2.0	RESV
	3.0	GRAT	3.0	REIN	3.0	REIN
	4.0	LRAT	4.0	VREP	4.0	VREP
	5.0	RESV	5.0		5.0	
	6.0	PRBL	6.0		6.0	
	7.0	ENERGY	7.0		7.0	
	8.0	WGRA	8.0	WGRA	8.0	WGRA
	9.0	CRAT	9.0	Availability	9.0	Availability
	10.0	PBGS	10.0		10.0	
	11.0	PBWS	11.0		11.0	
	12.0	CRAT	12.0		12.0	

Notes:

1) Cells colored in gray with a mnemonic indicate that the summary variable is not available in OPM Flow.
2) Cells colored in gray with no mnemonic indicate that the summary variable is not available for this combination.
3) Cells colored orange show combinations that are not available in OPM Flow because the underlying feature is not available.
4) For both production and injection groups the mode of control set to negative if the rate was set by a higher group.
5) Note that the output to the SUMMARY file is the numeric values, whereas for the RSM file it is the mnemonics, that is: ORAT, WRAT, etc.
6) See Table 11.11 for an explanation of the mnemonics used in Table 11.10.

Table 11.10: Field and Group Control Mode Reporting

Similarly, Table 11.11 shows the control mode reporting variables for wells.

Well Control Mode Reporting						
Object	**Well**			**Well**		
Mnemonic	WSTAT			WMCTL		
Description	Well Status indicator.			Well Mode of Control indicator.		
Values	**Value**	**Mnemonic**	**Description**	**Value**	**Mnemonic**	**Description**
	1.0	PROD	Well is a producing well.	-1.0		Well is under group control.
	2.0	INJ	Well is an injection well.	0.0	SHUT or STOP	Well is shut-in or stopped.
	3.0	SHUT	Well is shut-in except those under priority control.	1.0	ORAT	Well is on oil rate control.
	4.0	STOP	Well is stopped except those under priority control.	2.0	WRAT	Well is on water rate control.
	5.0	PSHUT	Well is shut-in under priority control.	3.0	GRAT	Well is on gas rate control.
	6.0	PSTOP	Well is stopped under priority control	4.0	LRAT	Well is on liquid rate control.
				5.0	RESV	Well is on reservoir voidage rate control.
				6.0	THP	Well is on THP control.
				7.0	BHP	Well is on BHP control.
				9.0	CRAT	Not used
				11.0	GOR Penalty	
				12.0	Drawdown	Well is draw down control
				13.0	NGL	Not used
				30.0	Availability	
				31.0	REIN	
				32.0	TMRA	
				33.0	WGRA	
				34.0	CVAL	
				41.0 - 51.0		

Well Control Mode Reporting		
Object	**Well**	**Well**
Mnemonic	WSTAT	WMCTL

Notes:

1) Cells colored in gray with a mnemonic indicate that the summary variable is not available in OPM Flow.
2) Cells colored in gray with no mnemonic indicate that the summary variable is not available for this combination.
3) Cells colored orange show combinations that are not available in OPM Flow because the underlying feature is not available.
4) Note that the output to the SUMMARY file is the numeric values, whereas for the RSM file it is the mnemonics, that is: PROD, INJ, etc.

Table 11.11: Well Control Mode Reporting

11.2.6 FIELD, REGION AND BLOCK VARIABLES

The next set of summary data is based on the block, field and region summary variables and the variables are tabulated in Table 11.9. Note when the data is for a block variable, like for example the oil saturation, this will be the actual value for the a grid cell (block), whereas, for the field and region items this will be an average value.

See also section *11.2.4 Field, Group, Well, Well Connection and Completion Variables* that outlines the field production data summary variables, and section *11.2.7 Field and Region Recovery Variables* that describes the summary variables for oil recovery efficiency and the oil recovery mechanism.

Block, Field and Region Summary Variables

Type	Variable	Root	Field	Region	Block	Comment
Flow	Gas Flow (Inter-Block in I+ Direction)	FLOGI			BFLOGI	
Flow	Gas Flow (Inter-Block in J+ Direction)	FLOGJ			BFLOGJ	
Flow	Gas Flow (Inter-Block in K+ Direction)	FLOGK			BFLOGK	
Flow-Reg	Gas Flow Rate Inter-Region	GFR		RGFR		OPM Flow Implementation
Flow-Reg	Gas Flow Rate Inter-Region (positive contribution)	GFR+		RGFR+		
Flow-Reg	Gas Flow Rate Inter-Region (negative contribution)	GFR-		RGFR-		
Flow-Reg	Gas Flow Total Inter-Region (Liquid and Gas Phases)	GFT		RGFT		
Flow-Reg	Gas Flow Total Inter-Region (+VE Liquid and Gas Phases)	GFT+		RGFT+		OPM Flow Implementation
Flow-Reg	Gas Flow Total Inter-Region (-VE Liquid and Gas Phases)	GFT-		RGFT-		
Flow-Reg	Gas Flow Total Inter-Region (Gas Phase)	GFTG		RGFTG		
Flow-Reg	Gas Flow Total Inter-Region (Liquid Phase)	GFTL		RGFTL		
Flow	Gas Injection Rate	GIR		RGIR		
Flow	Gas Injection Total	GIT		RGIT		
Flow	Gas Potential	PPG	FPPG	RPPG	BPPG	
Flow	Gas Production Rate	GPR		RGPR		
Flow	Gas Production Rate (Free)	GPRF		RGPRF		
Flow	Gas Production Rate (Solution)	GPRS		RGPRS		
Flow	Gas Production Total	GPT		RGPT		Liquid and gas phases
Flow	Gas Production Total (Free)	GPTF		RGPTF		
Flow	Gas Production Total (Net From Region)	GP		RGP		Minus injected gas
Flow	Gas Production Total (Solution)	GPTS		RGPTS		
Flow	Gas Velocity (Inter-Block in I+ Direction)	VELGI			BVELGI	
Flow	Gas Velocity (Inter-Block in J+ Direction)	VELGJ			BVELGJ	
Flow	Gas Velocity (Inter-Block in K+ Direction)	VELGK			BVELGK	

Block, Field and Region Summary Variables						
Type	**Variable**	**Root**	**Field**	**Region**	**Block**	**Comment**
Flow	Oil Flow (Inter-Block in I+ Direction)	FLOOI			BFLOOI	
Flow	Oil Flow (Inter-Block in J+ Direction)	FLOOJ			BFLOOJ	
Flow	Oil Flow (Inter-Block in K+ Direction)	FLOOK			BFLOOK	
Flow-Reg	Oil Flow Rate Inter-Region	OFR		ROFR		OPM Flow Implementation
Flow-Reg	Oil Flow Rate Inter-Region (positive contribution)	OFR+		ROFR+		
Flow-Reg	Oil Flow Rate Inter-Region (negative contribution)	OFR-		ROFR-		
Flow-Reg	Oil Flow Total Inter-Region (Liquid and Gas Phases)	OFT		ROFT		
Flow-Reg	Oil Flow Total Inter-Region (+VE Liquid and Gas Phases)	OFT+		ROFT+		OPM Flow Implementation
Flow-Reg	Oil Flow Total Inter-Region (-VE Liquid and Gas Phases)	OFT-		ROFT-		
Flow-Reg	Oil Flow Total Inter-Region (Gas Phase)	OFTG		ROFTG		
Flow-Reg	Oil Flow Total Inter-Region (Liquid Phase)	OFTL		ROFTL		
Flow	Oil Injection Rate	OIR		ROIR		
Flow	Oil Injection Total	OIT		ROIT		
Flow	Oil Potential	PPO	FPPO	RPPO	BPPO	
Flow	Oil Production Rate	OPR		ROPR		
Flow	Oil Production Total	OPT		ROPT		
Flow	Oil Production Total (Net From Region)	OP		ROP		
Flow	Oil Velocity (Inter-Block in I+ Direction)	VELOI			BVELOI	
Flow	Oil Velocity (Inter-Block in J+ Direction)	VELOJ			BVELOJ	
Flow	Oil Velocity (Inter-Block in K+ Direction)	VELOK			BVELOK	
Flow	Water Flow (Inter-Block in I+ Direction)	FLOWI			BFLOWI	
Flow	Water Flow (Inter-Block in J+ Direction)	FLOWJ			BFLOWJ	
Flow	Water Flow (Inter-Block in K+ Direction)	FLOWK			BFLOWK	
Flow-Reg	Water Flow Rate Inter-Region	WFR		RWFR		OPM Flow Implementation
Flow-Reg	Water Flow Rate Inter-Region	WFR+		RWFR+		
Flow-Reg	Water Flow Rate Inter-Region	WFR-		RWFR-		
Flow-Reg	Water Flow Total Inter-Region	WFT		RWFT		
Flow-Reg	Water Flow Total Inter-Region	WFT+		RWFT+		OPM Flow Implementation
Flow-Reg	Water Flow Total Inter-Region	WFT-		RWFT-		
Flow	Water Injection Rate	WIR		RWIR		

Block, Field and Region Summary Variables						
Type	**Variable**	**Root**	**Field**	**Region**	**Block**	**Comment**
Flow	Water Injection Total	WIT		RWIT		
Flow	Water Potential	PPW	FPPW	RPPW	BPPW	
Flow	Water Production Rate	WPR		RWPR		
Flow	Water Production Total	WPT		RWPT		
Flow	Water Production Total (Net From Region)	WP		RWP		
Flow	Water Velocity (Inter-Block in I+ Direction)	VELWI			BVELWI	
Flow	Water Velocity (Inter-Block in J+ Direction)	VELWJ			BVELWJ	
Flow	Water Velocity (Inter-Block in K+ Direction)	VELWK			BVELWK	
Pressure	Gas Phase Pressure	GPR			BGPR	
Pressure	Oil Phase Pressure	PR	FPR	RPR	BPR BPRESSUR	Field and Region HCPV weighted
Pressure	Oil Phase Pressure	PRH	FPRH	RPRH		Field and Region HCPV weighted
Pressure	Oil Phase Pressure (PV Weighted)	PRP	FPRP	RPRP		Field and Region PV weighted
Pressure	Water Phase Pressure	WPR			BWPR	
Property	Block Hydrocarbon Phase State	STATE			BSTATE	Gas (1), Gas & Oil (2), Oil (3)
Property	Bubble Point Pressure	PBUB			BPBUB	
Property	Capillary Pressure (Gas-Oil)	GPC			BGPC	
Property	Capillary Pressure (Water-Oil)	WPC			BWPC	
Property	Dew Point	PDEW			BPDEW	
Property	Gas P/Z	PRGZ	FPRGZ	RPRGZ		
Property	Gas Reservoir Density	GDEN	FGDEN	RGDEN	BGDEN	
Property	Gas Reservoir Density	DENG			BDENG	Same as BGDEN
Property	Gas Saturation	GSAT	FGSAT	RGSAT	BGSAT BGSAT	
Property	Gas Saturation (Drainage to Imbibition)	GSHY			BGSHY	Leaving gas saturation for gas capillary pressure hysteresis
Property	Gas Saturation (Dynamically Trapped)	GTPD			BGTPD	WAG Hysteresis only
Property	Gas Saturation (Trapped Critical)	GSTRP			BGSTRP	For gas capillary pressure hysteresis
Property	Gas Saturation (Trapped)	GTRP			BGTRP	WAG Hysteresis only
Property	Gas Viscosity	GVIS	FGVIS	RGVIS	BGVIS	

Block, Field and Region Summary Variables						
Type	**Variable**	**Root**	**Field**	**Region**	**Block**	**Comment**
Property	Gas-Oil Ratio (Saturated)	RSSAT			BRSSAT	
Property	Initial Contact Corrected Potential	PPC	FPPC	RPPC	BPPC	
Property	Oil Reservoir Density	ODEN	FODEN	RODEN	BODEN	
Property	Oil Reservoir Density	DENO			BDENO	Same as BODEN
Property	Oil Saturation	OSAT	FOSAT	ROSAT	BOSAT	Also BSOIL
Property	Oil Viscosity	OVIS	FOVIS	ROVIS	BOVIS	Also BVOIL
Property	Oil-Gas Ratio (Saturated)	RVSAT			BRVSAT	
Property	Relative Permeability (Gas in the R- Direction)	GKRR-			BGKRR-	
Property	Relative Permeability (Gas in the R+ Direction)	GKRR			BGKRR	
Property	Relative Permeability (Gas in the Theta- Direction)	GKRT-			BGKRT-	
Property	Relative Permeability (Gas in the Theta+ Direction)	GKRT			BGKRT	
Property	Relative Permeability (Gas in the X- Direction)	GKRX-			BGKRX-	Also BGKRI-
Property	Relative Permeability (Gas in the X+ Direction)	GKRX			BGKRX	Also BGKRI
Property	Relative Permeability (Gas in the Y- Direction)	GKRY-			BGKRY-	Also BGKRJ-
Property	Relative Permeability (Gas in the Y+ Direction)	GKRY			BGKRY	Also BGKRJ
Property	Relative Permeability (Gas in the Z- Direction)	GKRZ-			BGKRZ-	Also BGKRK-
Property	Relative Permeability (Gas in the Z+ Direction)	GKRZ			BGKRZ	Also BGKRK
Property	Relative Permeability (Gas)	KRG			BKRG	Also BGWR
Property	Relative Permeability (Krw Reduction Factor)	RK			BRK	Due to Polymer
Property	Relative Permeability (Oil Two Phase to Gas)	KROG			BKROG	
Property	Relative Permeability (Oil Two Phase to Water)	KROW			BKROW	
Property	Relative Permeability (Oil in the R- Direction)	OKRR-			BOKRR-	
Property	Relative Permeability (Oil in the R+ Direction)	OKRR			BOKRR	
Property	Relative Permeability (Oil in the Theta- Direction)	OKRT-			BOKRT-	
Property	Relative Permeability (Oil in the Theta+ Direction)	OKRT			BOKRT	
Property	Relative Permeability (Oil in the X- Direction)	OKRX-			BOKRX-	Also BOKRI-
Property	Relative Permeability (Oil in the X+ Direction)	OKRX			BOKRX	Also BOKRI
Property	Relative Permeability (Oil in the Y- Direction)	OKRY-			BOKRY-	Also BOKRJ-
Property	Relative Permeability (Oil in the Y+ Direction)	OKRY			BOKRY	Also BOKRJ
Property	Relative Permeability (Oil in the Z- Direction)	OKRZ-			BOKRZ-	Also BOKRK-
Property	Relative Permeability (Oil in the Z+ Direction)	OKRZ			BOKRZ	Also BOKRK

Block, Field and Region Summary Variables						
Type	**Variable**	**Root**	**Field**	**Region**	**Block**	**Comment**
Property	Relative Permeability (Oil)	KRO			BKRO	Also BOKR
Property	Relative Permeability (Water in the R- Direction)	WKRR-			BWKRR-	
Property	Relative Permeability (Water in the R+ Direction)	WKRR			BWKRR	
Property	Relative Permeability (Water in the Theta- Direction)	WKRT-			BWKRT-	
Property	Relative Permeability (Water in the Theta+ Direction)	WKRT			BWKRT	
Property	Relative Permeability (Water in the X- Direction)	WKRX-			BWKRX-	Also BWKRI-
Property	Relative Permeability (Water in the X+ Direction)	WKRX			BWKRX	Also BWKRI
Property	Relative Permeability (Water in the Y- Direction)	WKRY-			BWKRY-	Also BWKRJ-
Property	Relative Permeability (Water in the Y+ Direction)	WKRY			BWKRY	Also BWKRJ
Property	Relative Permeability (Water in the Z- Direction)	WKRZ-			BWKRZ-	Also BWKRK-
Property	Relative Permeability (Water in the Z+ Direction)	WKRZ			BWKRZ	Also BWKRK
Property	Relative Permeability (Water)	KRW			BKRW	Also BWKR
Property	Water Reservoir Density	WDEN	FWDEN	RWDEN	BWDEN	
Property	Water Reservoir Density	DENW			BDENW	Same as BWDEN
Property	Water Saturation	WSAT	FWSAT	RWSAT	BWSAT	Also BSWAT
Property	Water Saturation (Drainage to Imbibition)	WSHY			BWSHY	Leaving water saturation for gas capillary pressure hysteresis
Property	Water Saturation (Maximum Wetting)	WSMA			BWSMA	Capillary pressure hysteresis maximum water saturation
Property	Water Viscosity	WVIS	FWVIS	RWVIS	BWVIS	
Ratio	Gas-Oil Ratio	RS	FRS	RRS	BRS	
Ratio	Oil-Gas Ratio	RV	FRV	RRV	BRV	
Volumes	Gas In-Place	GIP	FGIP	RGIP	BGIP	Liquid and gas phases
Volumes	Gas In-Place (Gas Phase)	GIPG	FGIPG	RGIPG	BGIPG	
Volumes	Gas In-Place (Liquid Phase)	GIPL	FGIPL	RGIPL	BGIPL	Only for cases that have dissolved gas in the water phase
Volumes	Gas In-Place (Reservoir Conditions)	GIPR	FGIPR			Compositional vector implemented in OPM Flow.
Volumes	Oil In-Place	OIP	FOIP	ROIP	BOIP	Liquid and gas phases
Volumes	Oil In-Place (Gas Phase)	OIPG	FOIPG	ROIPG	BOIPG	
Volumes	Oil In-Place (Liquid Phase)	OIPL	FOIPL	ROIPL	BOIPL	

Block, Field and Region Summary Variables						
Type	**Variable**	**Root**	**Field**	**Region**	**Block**	**Comment**
Volumes	Oil In-Place (Reservoir Conditions)	OIPR	FOIPR			Compositional vector implemented in OPM Flow.
Volumes	Pore Volume (Gas)	GPV	FGPV	RGPV	BGPV	
Volumes	Pore Volume (HCPV)	HPV	FHPV	RHPV	BHPV	Also BHPV
Volumes	Pore Volume (Oil)	OPV	FOPV	ROPV	BOPV	
Volumes	Pore Volume (Reservoir Conditions)	RPV	FRPV		BRPV	
Volumes	Pore Volume (Water)	WPV	FWPV	RWPV	BWPV	
Volumes	Rock Compaction (Trans. Multiplier X Direction)	PERMMDX			BPERMMDX	Also BPERMOD
Volumes	Rock Compaction (Trans. Multiplier Y Direction)	PERMMDY			BPERMMDY	
Volumes	Rock Compaction (Trans. Multiplier Z Direction)	PERMMDZ			BPERMMDZ	
Volumes	Rock Compaction (Trans. Multiplier)	RTM	FRTM	RRTM	BRTM	
Volumes	Rock Compaction (Dual Porosity SIGMA Multiplier)	SIGMMOD			BSIGMMOD	
Volumes	Rock Compaction (PV Multiplier)	PORVMOD			BPORVMOD	
Volumes	Water In-Place	WIP	FWIP	RWIP	BWIP	
Volumes	Water In-Place (Reservoir Conditions)	WIPR	FWIPR			Compositional vector implemented in OPM Flow.

Notes:

1) Cells colored in gray with a mnemonic indicate that the summary variable is not available in OPM Flow.
2) Cells colored in gray with no mnemonic indicate that the summary variable is not available for this combination.
3) Cells colored orange show combinations that are not available in OPM Flow because the underlying feature is not available.

Table 11.12: Block, Field and Region Summary Variables

11.2.7 FIELD AND REGION RECOVERY VARIABLES

Oil recovery factors (or efficiency) may be reported at both the field and the region levels using a variety of parameters as summarized in Table 11.13 and equations (11.1) to (11.6). In addition to the actual recovery factors the simulator can also report the oil volumes produced by various drive mechanisms as described in the table.

See also section *11.2.4 Field, Group, Well, Well Connection and Completion Variables* that outlines the field production data summary variables, and section *11.2.6 Field, Region and Block Variables* that outlines the field, region and block production and injection data summary variables.

Field and Region Summary Recovery Variables					
Type	**Variable**	**Root**	**Field**	**Region**	**Comment**
Recovery	Oil Recovery Efficiency (STOIIP)	OE	FOE	ROE	Based on STOIIP
Recovery	Oil Recovery Efficiency (Wells)	OEW	FOEW	ROEW	Based on well production and STOIIP
Recovery	Oil Recovery Efficiency (STOIIP Mobile with respect to Water)	OEIW	FOEIW	ROEIW	Based on STOIIP and initial mobile oil saturate with respect to water
Recovery	Oil Recovery Efficiency Wells Mobile with respect to Water)	OEWW	FOEWW	ROEWW	Based on well production and initial mobile oil saturate with respect to water
Recovery	Oil Recovery Efficiency (STOIIP Mobile with respect to Gas)	OEIG	FOEIG	ROEIG	Based on STOIIP and initial mobile oil saturate with respect to gas.
Recovery	Oil Recovery Efficiency (Wells Mobile with respect to Gas)	OEWG	FOEWG	ROEWG	Based on well production and initial mobile oil saturate with respect to gas.
Recovery	Oil Production Total (Free Gas Gas)	OTMF	FOTMF	ROTMF	Volumes reported at stock tank conditions
Recovery	Oil Production Total (Gas Influx)	ORMG	FORMG	RORMG	
Recovery	Oil Production Total (Oil Expansion)	ORME	FORME	RORME	
Recovery	Oil Production Total (Rock Compaction)	ORMR	FORMR	RORMR	
Recovery	Oil Production Total (Solution Gas)	ORMS	FORMS	RORMS	
Recovery	Oil Production Total (Water Influx)	ORMW	FORMW	RORMW	

Notes:

1) Cells colored in gray with a mnemonic indicate that the summary variable is not available in OPM Flow.
2) Cells colored in gray with no mnemonic indicate that the summary variable is not available for this combination.
3) Cells colored orange show combinations that are not available in OPM Flow because the underlying feature is not available.

Table 11.13: Field and Region Summary Recovery Variables

Where:

$$\mathrm{OE} = \frac{\left(STOIIP_{t=0} - STOIIP_{t=i}\right)}{STOIIP_{t=0}} \tag{11.1}$$

$$\mathrm{OEW} = \frac{\left(\text{Well Oil Production}_{t=i}\right)}{STOIIP_{t=0}} \tag{11.2}$$

$$\mathrm{OEIW} = \frac{\left(STOIIP_{t=0} - STOIIP_{t=i}\right)}{\left(\text{Initial Mobile Oil Saturation with respect to Water}_{t=0}\right)} \tag{11.3}$$

$$\mathrm{OEWW} = \frac{\left(\text{Well Oil Production}_{t=i}\right)}{\left(\text{Initial Mobile Oil Saturation with respect to Water}_{t=0}\right)} \tag{11.4}$$

$$\mathrm{OEIG} = \frac{\left(STOIIP_{t=0} - STOIIP_{t=i}\right)}{\left(\text{Initial Mobile Oil Saturation with respect to Gas}_{t=0}\right)} \tag{11.5}$$

$$\mathrm{OEWG} = \frac{\left(\text{Well Oil Production}_{t=i}\right)}{\left(\text{Initial Mobile Oil Saturation with respect to Gas}_{t=0}\right)} \tag{11.6}$$

In addition to all the aforementioned summary variables, there are variables associated with specific features that may have been enabled in the input deck, for instance, the Brine Model, or the Polymer Model. The next set of sections describe the variables associated with these specific options.

11.2.8 OPTION SPECIFIC VARIABLES – ALKALINE MODEL

This feature has not been implemented in OPM Flow.

11.2.9 OPTION SPECIFIC VARIABLES – API AND TRACER TRACKING

The summary variables in this section are for use for when either the API tracking model has be activated by the API keyword in the RUNSPEC section so that the various "oil types" are tracked in the model, or for when the Tracer Model has been requested via the TRACERS keyword in the RUNSPEC section, that defines the number of tracers in the model and the various passive tracer tracking options. Only the Tracer model has been currently implement in OPM Flow.

API and Tracer Tracking Summary Variables

Type	Variable	Root	Field	Group	Well	Well Connection	Region	Block
Flow	Oil API	API	FAPI	GAPI	WAPI	CAPI	RAPI	BAPI
Flow	Tracer Flow Rate	TFR				CTFR		
Flow	Tracer Flow Total (Free Inter Region)	TFTF					RTFTF	
Flow	Tracer Flow Total (Free Inter Region)	TFTT					RTFTT	
Flow	Tracer Flow Total (Solution Inter Region)	TFTS					RTFTS	
Flow	Tracer Injection Concentration	TIC	FTIC	GTIC	WTIC	CTIC		
Flow	Tracer Injection Rate	TIR	FTIR	GTIR	WTIR	CTIR		
Flow	Tracer Injection Total	TIT	FTIT	GTIT	WTIT	CTIT		
Flow	Tracer Production Concentration	TPC	FTPC	GTPC	WTPC	CTPC		
Flow	Tracer Production Rate	TPR	FTPR	GTPR	WTPR	CTPR		
Flow	Tracer Production Total	TPT	FTPT	GTPT	WTPT	CTPT		
Volume	Tracer Concentration	TCN						BTCN
Volume	Tracer Concentration (Free)	TCNF						BTCNF
Volume	Tracer Concentration (Solution)	TCNS						BTCNS
Volume	Tracer In-Place	TIPT	FTIPT				RTIPT	BTIPT
Volume	Tracer In-Place (Free)	TIPF	FTIPF				RTIPF	BTIPF
Volume	Tracer In-Place (Solution)	TIPS	FTIPS				RTIPS	BTIPS

Notes:

1) Cells colored in gray with a mnemonic indicate that the summary variable is not available in OPM Flow.
2) Cells colored in gray with no mnemonic indicate that the summary variable is not available for this combination.
3) Cells colored orange show combinations that are not available in OPM Flow because the underlying feature is not available.

Table 11.14:API and Tracer Tracking Summary Variables

Note that for the tracer summary variables the mnemonic should be concatenated with the tracer name as defined by the TRACER keyword in the PROPS section. This keyword defines a series of passive tracers that are associated with a phase (oil, water, or gas) in the model, see the example for reference.

Example

In the PROPS section the TRACER keyword defines four passive tracers one for a gas injection well, one for tracking the dissolved gas, and two to track the injected water from two water injection wells.

```
-- ======================================================================
--
-- PROPS SECTION
--
-- ======================================================================
PROPS
--
--       DEFINE TRACER NAMES
--
--       TRACER   TRACER
--       NAME     PHASE
--       ------   ------
TRACER
        'IGS'     'GAS'                                  / GAS INJECTOR
        'DGS'     'GAS'                                  / DISOLVED GAS
        'IW1'     'WAT'                                  / WAT INJECTOR 1
        'Iw2'     'WAT'                                  / WAT INJECTOR 2
/
```

If we wish to report the field total amount of the tracers in-place we would use the following in the SUMMARY section:

```
-- ======================================================================
--
-- SUMMARY SECTION
--
-- ======================================================================
SUMMARY

FTIPTIGS
FTIPTDGS
FTIPIW1
FTIPIW2
```

Notice there are no terminating "/" for field summary variables.

11.2.10 OPTION SPECIFIC VARIABLES – ASPHALTENE MODEL

This feature has not been implemented in OPM Flow.

11.2.11 OPTION SPECIFIC VARIABLES – BRINE MODEL

The available Brine model summary keywords are dependent on which form of the Brine model has been invoked in the simulation input deck. In the RUNSPEC section the BRINE keyword activates the standard Brine Tracking model and optionally defines the water phase to have various salinities if the ECLMC keyword in the RUNSPEC section has been used to activate the Multi-Component Brine model. The Multi-Component Brine model allows for the water phase to have multiple water salinities.

Note that the Multi-Component Brine model is currently not supported by OPM Flow.

If OPM Flow's Salt Precipitation Model has been activated in the input deck via the PRECSALT keyword in the RUNSPEC section, then the Brine model summary keywords can also be employed with this option.

Brine and Multi-Component Brine Model Summary Variables

Type	Variable	Root	Field	Group	Well	Well Connection	Region	Block	Comment
Flow	Salt Flow Rate	SFR				CSFR			Brine and Salt Precipitation Models
Flow	Salt Injection Rate	SIR	FSIR	GSIR	WSIR	CSIR			
Flow	Salt Injection Total	SIT	FSIT	GSIT	WSIT	CSIT			
Flow	Salt Production Rate	SPR	FSPR	GSPR	WSPR	CSPR			
Flow	Salt Production Total	SPT	FSPT	GSPT	WSPT	CSPT			
Property	Salt Corrected Water Viscosity	EMV_SAL						BEMV_SAL	
Volume	Salt Concentration (Block)	SCN						BSCN	
Volume	Salt Concentration (Injection)	SIC	FSIC	GSIC	WSIC	CSIC			
Volume	Salt Concentration (Production)	SPC	FSPC	GSPC	WSPC	CSPC			
Volume	Salt In-Place	SIP	FSIP				RSIP	BSIP	
Flow	Anion Concentration (Block)	TCNFANI						BTCNFANI	Multi-Component Brine Model
Flow	Anion Flow Rate	TFRANI				CTFRANI			
Flow	Anion Injection Rate	TIRANI	FTIRANI	GTIRANI	WTIRANI				
Flow	Anion Injection Total	TITANI	FTITANI	GTITANI	WTITANI	CTITANI			
Flow	Anion Production Rate	TPRANI	FTPRANI	GTPRANI	WTPRANI				
Flow	Anion Production Total	TPTANI	FTPTANI	GTPTANI	WTPTANI	CTPTANI			
Flow	Cation Flow Rate	TFRCAT				CTFRCAT			
Flow	Cation Injection Rate	TIRCAT	FTIRCAT	GTIRCAT	WTIRCAT				
Flow	Cation Injection Total	TITCAT	FTITCAT	GTITCAT	WTITCAT	CTITCAT			
Flow	Cation Production Rate	TPRCAT	FTPRCAT	GTPRCAT	WTPRCAT				
Flow	Cation Production Total	TPTCAT	FTPTCAT	GTPTCAT	WTPTCAT	CTPTCAT			

Brine and Multi-Component Brine Model Summary Variables									
Type	**Variable**	**Root**	**Field**	**Group**	**Well**	**Well Connection**	**Region**	**Block**	**Comment**
Property	Effective Salinity (Polymer)	ESALPLY						BESALPLY	
Property	Effective Salinity (Surfactant)	ESALSUR						BESALSUR	
Volume	Cation Concentration (Block)	TCNFCAT						BTCNFCAT	
Volume	Cation Concentration (Rock Associated)	TRADCAT						BTRADCAT	
Volume	Cation Concentration (Surfactant Associated)	TSADCAT						BTSADCAT	

Notes:

1) Cells colored in gray with a mnemonic indicate that the summary variable is not available in OPM Flow.
2) Cells colored in gray with no mnemonic indicate that the summary variable is not available for this combination.
3) Cells colored orange show combinations that are not available in OPM Flow because the underlying feature is not available.

Table 11.15: Brine and Multi-Component Brine Model Summary Variables

Note that not all these variables are available in OPM Flow; however, the simulator will issue a warning messages if this is indeed the case. It is anticipated that the number of recognized summary variables will increase in future releases of OPM Flow.

11.2.12 OPTION SPECIFIC VARIABLES - CO2STORE MODEL

The variables in this section are for OPM Flow's black-oil CO2STORE Model, and are based on the commercial simulator's compositional vectors, as the model is only available in the commercial simulator's compositional simulator.

CO2STORE Summary Variables								
Type	**Variable**	**Root**	**Field**	**Group**	**Well**	**Well Connection**	**Region**	**Block**
Volume	CO_2 Dissolved and immobile in the gas phase	GCDI	FGCDI				RGCDI	
Volume	CO_2 Dissolved and mobile in the gas phase	GCDM	FGCDM				RGCDM	
Volume	CO_2 Dissolved in the water phase	WCD	FWCD				RWCD	
Volume	Water in the gas phase	WIPG	FWIPG				RWIPG	
Volume	Water in the water phase	WIPL	FWIPL				RWIPL	

Notes:

1) Cells colored in gray with a mnemonic indicate that the summary variable is not available in OPM Flow.
2) Cells colored in gray with no mnemonic indicate that the summary variable is not available for this combination.
3) Cells colored orange show combinations that are not available in OPM Flow because the underlying feature is not available.

Table 11.16: CO2STORE Summary Variables

Note that the WIPG and WIPL series of variables are OPM Flow specific variables.

11.2.13 OPTION SPECIFIC VARIABLES – COAL BED METHANE MODEL

The Coal Bed Methane model is not supported by OPM Flow.

11.2.14 OPTION SPECIFIC VARIABLES – CONDUCTIVE FAULTS MODEL

Conductive faults have not been implemented in OPM Flow.

11.2.15 OPTION SPECIFIC VARIABLES – ENVIRONMENTAL TRACERS

Environmental tracers are not supported by OPM Flow.

11.2.16 OPTION SPECIFIC VARIABLES – FORMATION DAMAGE MODEL

The variables in this section are for OPM Flow's Formation Damage Model. The WINJDAM keyword sets up the filter cake properties for the specific water injection well in the SCHEDULE section.

Formation Damage Model Summary Variables								
Type	**Variable**	**Root**	**Field**	**Group**	**Well**	**Well Connection**	**Region**	**Block**
Flow	Filtrate Injection Concentration	INJFC			WINFC			
Flow	Filtrate Volume Injection Rate	INJFVR			WINJFVR	CINJFVR		
Flow	Filtrate Volume Injection Total	INJFVT			WINJFVT	CINJFVT		
Property	Filter Cake Area Of Flow	FCAOF				CFCAOF		
Property	Filter Cake Permeability	FCPERM				CFCPERM		
Property	Filter Cake Porosity	FCPORO				CFCPORO		
Property	Filter Cake Radius	FCRAD				CFCRAD		
Property	Filter Cake Skin	FCSKIN				CFCSKIN		
Property	Filter Cake Width	FCWIDTH				CFCWIDTH		

Notes:

1) Cells colored in gray with a mnemonic indicate that the summary variable is not available in OPM Flow.
2) Cells colored in gray with no mnemonic indicate that the summary variable is not available for this combination.
3) Cells colored orange show combinations that are not available in OPM Flow because the underlying feature is not available.

Table 11.17: Formation Damage Model Summary Variables

Note that these are OPM Flow specific variables.

11.2.17 OPTION SPECIFIC VARIABLES – FOAM MODEL TRACERS

The Foam phase and model are activated via the FOAM keyword in the RUNSPEC section. Note in the commercial simulator the FOAM phase and model can be used in conjunction with the POLYMER and SURFACT phases; this is not the case for OPM Flow. OPM Flow's FOAM phase and model is a standalone implementation and cannot be used in conjunction with the either the POLYMER or SURFACT phases.

Foam Model Tracers Summary Variables

Type	Variable	Root	Field	Group	Well	Well Connection	Region	Block	Comment
Flow	Foam Flow Rate	TFRFOA				CTFRFOA			
Flow	Foam Injection Rate	TIRFOA	FTIRFOA	GTIRFOA	WTIRFOA				
Flow	Foam Injection Total	TITFOA	FTITFOA	GTITFOA	WTITFOA	CTITFOA			
Flow	Foam Production Rate	TPRFOA	FTPRFOA	GTPRFOA	WTPRFOA				
Flow	Foam Production Total	TPTFOA	FTPTFOA	GTPTFOA	WTPTFOA	CTPTFOA			
Property	Foam Adsorption Total	TADSFOA	FTADSFOA			CTADSFOA	RTADSFOA	BTADSFOA	Block is current value
Property	Foam Capillary Number	TCNMFOA						BTCNMFOA	
Property	Foam Decayed Half Life	THLFFOA						BTHLFFOA	
Property	Foam Decayed Tracer	TDCYFOA	FTDCYFOA				RTDCYFOA	BTDCYFOA	
Property	Foam Gas Mobility Factor	TMOBFOA	FTMOBFOA						Excluding shear
Property	Foam in Solution	TIPTFOA	FTIPTFOA			CTIPTFOA	RTIPTFOA	BTIPTFOA	
Volume	Foam Concentration	TCNFFOA						BTCNFFOA	

Notes:

1) Cells colored in gray with a mnemonic indicate that the summary variable is not available in OPM Flow.
2) Cells colored in gray with no mnemonic indicate that the summary variable is not available for this combination.
3) Cells colored orange show combinations that are not available in OPM Flow because the underlying feature is not available.

Table 11.18: Foam Model Tracers Summary Variables

Note that not all these variables are available in OPM Flow; however, the simulator will issue a warning messages if this is indeed the case. It is anticipated that the number of recognized summary variables will increase in future releases of OPM Flow.

11.2.18 OPTION SPECIFIC VARIABLES – GAS FIELD OPERATIONS MODEL

The Gas Field Operations model has not been implemented in OPM Flow.

11.2.19 OPTION SPECIFIC VARIABLES – GAS LIFT OPTIMIZATION MODEL

For the Gas Lift Optimization model only a few SUMMARY vectors are available, both in OPM Flow and the commercial simulator, as depicted in Table 11.19. Gas lift optimization is activated by the LIFTOPT keyword in the RUNSPEC section as gas lift activation can vary through time depending on the flow characteristics of the wells.

Gas Lift Optimization Model Summary Variables								
Type	**Variable**	**Root**	**Field**	**Group**	**Well**	**Well Connection**	**Region**	**Block**
Flow	Gas lift gas rate, based on ALQ.	GLIR	FGLIR	GGLIR	WGLIR			
Flow	Incremental oil rate per unit incremental gas lift gas quantity, that is the well's GRADIENT as defined by equation (11.7) below.	OGLR			WOGLR			

Notes:

1) Cells colored in gray with a mnemonic indicate that the summary variable is not available in OPM Flow.
2) Cells colored in gray with no mnemonic indicate that the summary variable is not available for this combination.
3) Cells colored orange show combinations that are not available in OPM Flow because the underlying feature is not available.

Table 11.19: Gas Lift Optimization Model Summary Variables

The incremental gas lift gas supply is allocated to a well based on the well's current incremental gradient using the following formulae:

$$Gradient = \left(\frac{(\beta_w \times \Delta Q_{Oil})}{(\Delta Q_{GasLift} + \beta_g \times \Delta Q_{Gas})} \right) \quad (11.7)$$

Where:

βw = a weighting factor for the preferential allocation of lift gas,
βg = the gas production rate weighting factor,
ΔQOil = the increment/decrement in oil production rate,
ΔQGas = the increment/decrement in gas production rate, and
ΔQGasLift = the increment/decrement in gas lift gas rate.

Normally βg is defaulted to zero and in which case equation (11.7) reduces to:

$$Gradient = \left(\frac{(\beta_w \times \Delta Q_{Oil})}{\Delta Q_{GasLift}} \right) \quad (11.8)$$

Note if gas lift optimization has been activated in the model then the gas production SUMMARY variables (FGPR, GGPR, WGPR, FGPT etc.) only contain the produced gas volumes, that is the reported values exclude the gas associated with gas lift gas.

Secondly, the well gas lift rates are adjusted by the well efficiencies in the calculating the group and field gas lift rates.

11.2.20 OPTION SPECIFIC VARIABLES – GAS CALORIFIC VALUE REPORTING

This feature has not been implemented in OPM Flow.

11.2.21 OPTION SPECIFIC VARIABLES – GI PSUEDO-COMPOSITIONAL MODEL

This feature has not been implemented in OPM Flow.

11.2.22 OPTION SPECIFIC VARIABLES – LGR AND COARSENING

Local Grid Refinement ("LGR") and cell coarsening have not been implemented in OPM Flow.

11.2.23 OPTION SPECIFIC VARIABLES – MULTI-SEGMENT WELLS

The summary variables in this section are for use for when multi-segment wells are active in the model, and the variables only apply to the well segments requested for a multi-segment well, not to the well, well connection or well completion.

Note that not all these variables are available in OPM Flow; however, the simulator will issue a warning messages if this is indeed the case. It is anticipated that the number of recognized summary variables will increase in future releases of OPM Flow.

Multi-Segment Wells Summary Variables				
Type	**Variable**	**Root**	**Well Segment**	**Comment**
Flow	Artificial Lift Quantity	ALQ	SALQ	See keyword WSEGTABL
Flow	Brine Flow Rate	FR	SFR	Surfactant and Polymer model
Flow	Gas Flow Rate	GFR	SGFR	
Flow	Gas Flow Rate (Free)	GFRF	SGFRF	
Flow	Gas Flow Rate (Solution)	GFRS	SGFRS	
Flow	Gas Flow Total (Absolute)	GFTA	SGFTA	
Flow	Gas Flow Total	GFT	SGFT	
Flow	Gas Flow Velocity	GFV	SGFV	
Flow	Gas Holdup Fraction	GHF	SGHF	
Flow	Gas Import Rate	GIMR	SGIMR	See keyword WSEGEXSS
Flow	Gas Import Total	GIMT	SGIMT	See keyword WSEGEXSS
Flow	Oil Absolute Flow Total	OFTA	SOFTA	
Flow	Oil Flow Rate	OFR	SOFR	
Flow	Oil Flow Rate (Free)	OFRF	SOFRF	
Flow	Oil Flow Rate (Solution)	OFRS	SOFRS	
Flow	Oil Flow Total	OFT	SOFT	
Flow	Oil Flow Velocity	OFV	SOFV	
Flow	Oil Holdup Fraction	OHF	SOHF	
Flow	Polymer Flow Rate	CFR	SCFR	Polymer model
Flow	Tracer Flow Rate	TFR	STFR	Tracer model
Flow	Water Flow Rate	WFR	SWFR	
Flow	Water Flow Total	WFT	SWFT	
Flow	Water Flow Total (Absolute)	WFTA	SWFTA	
Flow	Water Flow Velocity	WFV	SWFV	
Flow	Water Holdup Fraction	WHF	SWHF	

Multi-Segment Wells Summary Variables				
Type	**Variable**	**Root**	**Well Segment**	**Comment**
Flow	Water Import Rate	WIMR	SWIMR	See keyword WSEGEXSS
Flow	Water Import Total	WIMT	SWIMT	See keyword WSEGEXSS
Pressure	Pressure	PR	SPR	
Pressure	Pressure Drop	PRD	SPRD	
Pressure	Pressure Drop component due to Friction	PRDF	SPRDF	
Pressure	Pressure Drop component due to Hydrostatic head	PRDH	SPRDH	
Pressure	Pressure drop due to Acceleration head	PRDA	SPRDA	
Pressure	Pressure Drop Frictional Multiplier	PRDM	SPRDM	See keyword WSEGMULT
Property	API	API	SAPI	API model
Property	Fluid Mixture Density (Excludes Exponents)	DENM	SDENM	Excludes exponents of flowing fractions
Property	Fluid Mixture Density (Includes Exponents)	MDEN	SMDEN	Includes exponents of flowing fractions
Property	Fluid Mixture Viscosity (Includes Exponents)	MVIS	SMVIS	Includes exponents of flowing fractions
Property	Fluid Viscosity (Effective Mixture)	EMVIS	SEMVIS	Water/polymer fluid
Property	Gas Density	GDEN	SGDEN	
Property	Gas Viscosity	GVIS	SGVIS	
Property	Gas-Liquid Drift Velocity (Vd)	GLVD	SGLVD	Drift flux slip model
Property	Gas-Liquid Profile Parameter (C0)	GLPP	SGLPP	Drift flux slip model
Property	Oil Density	ODEN	SODEN	
Property	Oil Viscosity	OVI	SOVI	
Property	Oil-Water Drift Velocity (Vd)	OWVD	SOWVD	Drift flux slip model
Property	Oil-Water Profile Parameter (C0)	OWPP	SOWPP	Drift flux slip model
Property	Pump Working Power	PPOW	SPPOW	See keyword WSEGPULL
Property	Scale Deposition Diameter (Karst Conduit Calcite Dissolution)	FD	SFD	Scale Deposition model
Property	Setting of segment	FOPN	SFOPN	See keywords WSEGVALV, WSEGAICD, WSEGSICD, WSEGTABL, WSEGLABY and WSEGFLIM)
Property	Strength of ICD on segment	STR	SSTR	See keywords WSEGVALV, WSEGAICD and WSEGSICD)
Property	Water Density (Pure Water)	WDEN	SWDEN	
Property	Water Viscosity (Pure Water)	WVIS	SWVIS	
Ratio	Gas-Oil Ratio	GOR	SGOR	
Ratio	Oil-Gas Ratio	OGR	SOGR	

Multi-Segment Wells Summary Variables				
Type	**Variable**	**Root**	**Well Segment**	**Comment**
Ratio	Water Cut	WCT	SWCT	
Ratio	Water Gas Ratio	WGR	SWGR	
Volume	Brine Concentration	SCN	SSCN	Surfactant and Polymer model
Volume	Polymer Concentration	CCN	SCCN	Polymer model
Volume	Tracer Concentration	TFC	STFC	Tracer model

Notes:

1) Cells colored in gray with a mnemonic indicate that the summary variable is not available in OPM Flow.
2) Cells colored in gray with no mnemonic indicate that the summary variable is not available for this combination.
3) Cells colored orange show combinations that are not available in OPM Flow because the underlying feature is not available.

Table 11.20: Multi-Segment Wells Summary Variables

Examples

For example, to get the segment oil production rates for well OP01 segments two, three, and four and for segment two for OP02 one would use:

```
-- ==========================================================================
--
-- SUMMARY SECTION
--
-- ==========================================================================
SUMMARY

SOFR
'OP01' 2 /
'OP01' 3 /
'OP01' 4 /
'OP02' 2 /
/
```

or to get all the oil production rates for all the segments for well OP01:

```
-- ==========================================================================
--
-- SUMMARY SECTION
--
-- ==========================================================================
SUMMARY

SOFR
'OP01' /
/
```

Care should be exercised when defaulting the list of wells and segments as there is the potential to generate large volumes of data.

11.2.24 OPTION SPECIFIC VARIABLES – NETWORK MODEL

For the Network models only a few SUMMARY vectors for groups and nodes are available, both in OPM Flow and the commercial simulator, as depicted in Table 11.21. There are two types of network option in the simulator a Standard Network and an Extended Network option, the latter is activated by the NETWORK keyword in the RUNSPEC section. The summary vectors apply to both the Standard and Extended Network options.

Network Model Summary Variables								
Type	**Variable**	**Root**	**Field**	**Group**	**Well**	**Well Connection**	**Region**	**Block**
Pressure	Group/Node pressure in a production network.			GPR				
Pressure	Group /Node pressure in a gas injection network.			GPRG				
Pressure	Group/Node pressure in a water injection network.			GPRW				
Pressure	Pressure drop along the group's or node's outlet branch in a production network.			GPRB				
Pressure	Pressure drop along the group's or node's inlet branch in a gas injection network.			GPRBG				
Pressure	Pressure drop along the group's or node's inlet branch in a water injection network.			GPRBW				
Flow	Artificial Lift Quantity ("ALQ") in the group's or node's outlet branch in a production network. Note no units are stated as ALQ units are variable, for example Hz for ESP or rate for gas lift gas.			GALQ				
Flow	Oil flow rate along the group's or node's outlet branch in a production network.			GOPRNB				
Flow	Water flow rate along the group's or node's outlet branch in a production network.			GWPRNB				
Flow	Gas flow rate along the group's or node's outlet branch in a production network.			GGPRNB				
Flow	Liquid flow rate along the group's or node's outlet branch in a production network.			GLPRNB				
Flow	Water flow rate along the group's or node's inlet branch in water injection network.			GWIRNB				
Flow	Gas flow rate along the group's or node's inlet branch in gas injection network.			GGIRNB				

Network Model Summary Variables								
Type	**Variable**	**Root**	**Field**	**Group**	**Well**	**Well Connection**	**Region**	**Block**

Notes:

1) Cells colored in gray with a mnemonic indicate that the summary variable is not available in OPM Flow.
2) Cells colored in gray with no mnemonic indicate that the summary variable is not available for this combination.
3) Cells colored orange show combinations that are not available in OPM Flow because the underlying feature is not available.

Table 11.21: Network Model Summary Variables

Note that not all these variables are available in OPM Flow; however, the simulator will issue a warning messages if this is indeed the case. It is anticipated that the number of recognized summary variables will increase in future releases of OPM Flow.

11.2.25 OPTION SPECIFIC VARIABLES – OPM FLOW SIMULATION PERFORMANCE

The following table (Table 11.22) lists the OPM Flow simulation performance summary variables that can be written to the SUMMARY file. Note that not all these variables are available in OPM Flow; however, the simulator will issue a warning messages if this is indeed the case. It is anticipated that the number of recognized summary variables will increase in future releases of OPM Flow.

OPM Flow Simulation Performance		
Variable Description	**Variable**	**Comment**
Elapsed - Cumulative waiting time per process in seconds.	DTMWAIT	Unsupported
Elapsed - Elapsed time in seconds.	ELAPSED	No data written to file.
Iterations - Number of linear iterations for each gradient calculation (Gradient Option).	HLINEARS	OPM Flow currently has not implemented the Gradient option.
Iterations - Total number of linear iterations for each gradient pressure (Gradient Option).	HSUMLINS	OPM Flow currently has not implemented the Gradient option.
Maximum Courant-Friedrichs-Lewy value for each time step.	MAXCFL	Compositional keyword
Maximum variation in energy for each time step.	MAXDE	Compositional keyword
Maximum variation in pressure for each time step.	MAXDPR	Unsupported
Maximum variation in gas saturation for each time step.	MAXDSG	Unsupported
Maximum variation in oil saturation for each time step.	MAXDSO	Unsupported
Maximum variation in water saturation for each time step.	MAXDSW	Unsupported
Maximum variation in temperature for each time step.	MAXDT	Compositional keyword
Memory - Operating system reported maximum current memory usage across all processors.	MEMORYTS	Unsupported
CPR Solver - Number of pressure iterations for each time step.	MLINEARP	Unsupported
Iterations - Number linear iterations for each time step.	MLINEARS	
Messages - Cumulative number of BUG messages.	MSUMBUG	Unsupported
Messages - Cumulative number of COMMENT messages.	MSUMCOMM	Unsupported
Messages - Cumulative number of ERROR messages.	MSUMERR	Unsupported
CPR Solver - Cumulative number of pressure iterations.	MSUMLINP	Unsupported
Iterations - Cumulative number of linear iterations.	MSUMLINS	
Messages - Cumulative number of MESSAGES messages.	MSUMMESS	Unsupported
Iterations - Cumulative number of Newton iterations.	MSUMNEWT	

OPM Flow Simulation Performance		
Variable Description	**Variable**	**Comment**
Messages - Cumulative number of PROBLEM messages.	MSUMPROB	Unsupported
Messages - Cumulative number of WARNING messages.	MSUMWARN	Unsupported
Number of flash calculations for each time step	NBAKFL	Compositional keyword
Memory - Peak usage of dynamically allocated memory reported by OPM Flow. For parallel cases this is the maximum across all processors.	NBYTOT	Unsupported
CPR Solver - Average number of pressure iterations per linear iteration for each time step.	NCPRLINS	Unsupported
Iterations – Cumulative average number of Newton iterations per cell in flash calculations..	NEWTFL	Compositional keyword
Iterations - Number of Newton iterations used for each time step.	NEWTON	
CPR Solver - Average number of pressure iterations per Newton iteration per time step.	NLINEARP	Unsupported
Iterations - Average number of linear iterations per Newton iteration for each time step.	NLINEARS	For runs with LGRs, LLINEARS will automatically be exported for each LGR.
Iterations - Maximum number of linear iterations in the Newton iterations per time step.	NLINSMAX	
Iterations - Minimum number of linear iterations in the Newton iterations per time step.	NLINSMIN	
Cumulative number of flash calculations.	NNUMFL	Compositional keyword
Cumulative number of stability tests.	NNUMST	Compositional keyword
Non-linear residual total.	NLRESSUM	Compositional keyword
Non-linear residual maximum value.	NLRESMAX	Compositional keyword
Number of time steps – taken.	NTS	Compositional keyword
Number of time steps – pressure converged.	NTSPCL	Compositional keyword
Number of time steps – molar density converged.	NTSMCL	Compositional keyword
Number of time steps – energy density converged.	NTSECL	Compositional keyword
Time Step – Criteria used to select the length of the time step, set to one of the following:	STEPTYPE	No data written to file.

OPM Flow Simulation Performance				
Variable Description			Variable	Comment
Value	Criteria	Definition		Note if the RUNSUM keyword has been activated in the SUMMARY section, then the mnemonics are written to the RSM file instead of integer values.
1	INIT	Initial time step.		
2	TRNC	Controlled via time truncation error.		
3	MINF	Minimum ratio between one time step and the next.		
4	MAXF	Maximum ratio between one time step and the next.		
5	MINS	Minimum permitted time step.		
6	MAXS	Maximum permitted time step.		
7	REPT	Report time step		
8	HALF	Half of the time set to reach the next report time step.		
9	CHOP	Time step chopped due non-convergence of non-linear equations.		
10	SATM	Maximum saturation change.		
11	PCHP	Time step chopped due to maximum pressure change in IMPES formulation.		
12	DIFF	Difficult time step after a time step CHOP.		
13	LGRC	Determined by LGR fluid in-place error targets.		
14	SURF	Set by maximum expected grid block surfactant concentration change.		
15	NETW	Time step determined by network balancing controls.		
16	THRP	Time step controlled by maximum through put ratio.		
17	EMTH	Time step selected to match the end of a month.		
18	MAXP	Set by maximum expected grid block pressure change.		

OPM Flow Simulation Performance				
Variable Description			**Variable**	**Comment**
19	WCYC	Determined by well cycling on/off time.		
20	MAST	Chosen by the master simulation in a reservoir coupling run.		
21	SLVR	Selected by a slave simulation to match a slave reporting time step, in a reservoir coupling run.		
22	SLVC	Time step selected by a slave simulation due to a slave's expected flow rate change in a reservoir coupling run.		
23	MAXW	Maximum time step size after a well control event.		
24	EFF+	Selected by ZIPPY optimum time selection algorithm.		
25	EFF-			
26	NLTR			
27	EFFT			
28	DLYA			
29	ACTN			
30	RAIN			
CPU - Current CPU usage in seconds.			TCPU	Does not consider the time taken by inter-process communications in parallel runs, whereas ELAPSED does. Thus, for parallel jobs, ELAPSED is the most relevant time measurement.
CPU - CPU time per day (or hour in lab units depending on run units system).			TCPUDAY	
CPU – Cumulative CPU time for each gradient calculation (Gradient Option).			TCPUH	OPM Flow currently has not implemented the Gradient option.
CPU - Cumulative CPU time for all gradient calculations (Gradient Option).			TCPUHT	OPM Flow currently has not implemented the Gradient option.
CPU - Cumulative CPU time used in SCHEDULE section.			TCPUSCH	Unsupported
CPU - CPU time per time step in seconds.			TCPUTS	
CPU - CPU time per time step for each gradient calculation (Gradient Option).			TCPUTSH	OPM Flow currently has not implemented the Gradient option.

OPM Flow Simulation Performance		
Variable Description	**Variable**	**Comment**
CPU - CPU time per time step for all gradient calculations (Gradient Option).	TCPUTSHT	OPM Flow currently has not implemented the Gradient option.
CPU – Cumulative Tracer CPU time.	TCPUTR	Compositional keyword
CPU – Cumulative Tracer implicit CPU time.	TCPUTRSV	Compositional keyword
CPU – Cumulative Tracer Lagrangian tracer stramline CPU time.	TCPULGTR	Compositional keyword
CPU – Cumulative Tracer Lagrangian tracer solver CPU time.	TCPULGSV	Compositional keyword
Elapsed - Elapsed time per day (or hour in lab units).	TELAPDAY	Unsupported
Elapsed - Elapsed time per linear iteration in seconds.	TELAPLIN	No data written to file.
Elapsed - Elapsed time per time step in seconds.	TELAPTS	Unsupported
Time Step - Length of time step.	TIMESTEP	
Iterations - Number of well Newton iterations taken in the last global Newton iteration. A negative value indicates that the well failed to converge.	WNEWTON	Unsupported
Time Step - Predicted efficiency of the time step (developer use).	ZIPEFF	Unsupported
Time Step - Predicted efficiency of the time step divided by the actual efficiency (developer use).	ZIPEFFC	Unsupported

Notes:

1) Cells under the Variable column not colored indicate that the summary variable is available in OPM Flow.
2) Cells under the Variable column colored in gray indicate that the summary variable is recognized by the parser but the summary variable is not available.
3) Cells under the Variable column colored in orange indicate that the summary variable is not recognized by the parser and the summary variable is not available. These summary variables may cause the simulator to abort.

Table 11.22: OPM Flow Simulator Performance Summary Variables

See also the PERFORMA and NMESSAGE keywords in section *11.3 Keyword Definitions* that write out a selection of the variables in Table 11.22.

11.2.26 OPTION SPECIFIC VARIABLES – POLYMER MODEL

The summary variables in this section are associated with Polymer phase and the polymer flood model. The feature is activated by the POLYMER keyword in the RUNSPEC section.

Note that not all these variables are available in OPM Flow; however, the simulator will issue a warning messages if this is indeed the case. It is anticipated that the number of recognized summary variables will increase in future releases of OPM Flow.

Polymer Model Summary Variables									
Type	Variable	Root	Field	Group	Well	Well Connection	Region	Block	Comment
Flow	Polymer Flow Rate	CFR				CCFR			
Flow	Polymer Flow Total (Inter-Region)	CFT					RCFT		
Flow	Polymer Injection Rate	CIR	FCIR	GCIR	WCIR	CCIR			
Flow	Polymer Injection Total	CIT	FCIT	GCIT	WCIT	CCIT			
Flow	Polymer Production Rate	CPR	FCPR	GCPR	WCPR	CCPR			
Flow	Polymer Production Total	CPT	FCPT	GCPT	WCPT	CCPT			
Flow	Salt Flow Rate	SFR				CSPR			
Flow	Salt Injection Rate	SIR	FSIR	GSIR	WSIR	CSIR			
Flow	Salt Injection Total	SIT	FSIT	GSIT	WSIT	CSIT			
Flow	Salt Production Rate	SPR	FSPR	GSPR	WSPR	CSPR			
Flow	Salt Production Total	SPT	FSPT	GSPT	WSPT	CSPT			
Property	Polymer Adsorbed (PLYTRRFA)	CABnnn						BCABnnn	By highest temperature band at which residual resistance factor was calculated, see keyword PLYTRRFA. The band number nnn can range from 001 to 999, but must be less than or equal to the argument on the PLYTRRFA keyword.
Property	Polymer Adsorption Total	CAD	FCAD				RCAD	BCAD	Block is concentration
Property	Polymer Solution (Effective Viscosity)	EPVIS						BEPVIS	Also BVPOLY
Property	Polymer Solution Shear Viscosity (Inter-Block I+ Direction)	SHWVISI						BSHWVISI	
Property	Polymer Solution Shear Viscosity (Inter-Block J+ Direction)	SHWVISJ						BSHWVISJ	

Polymer Model Summary Variables									
Type	Variable	Root	Field	Group	Well	Well Connection	Region	Block	Comment
Property	Polymer Solution Shear Viscosity (Inter-Block K+ Direction)	SHWVISK						BSHWVISK	
Property	Polymer Thermal Degradation	CDCS						BCDCS	Total mass degraded in previous time step
Property	Polymer Thermal Degradation Rate	CDCR						BCDCR	Total mass degraded in previous time step
Property	Polymer Thermal Degradation Rate (Adsorbed)	CDCA						BCDCA	
Property	Polymer Thermal Degradation Rate (Solution)	CDCP						BCDCP	
Property	Polymer Water (Effective Viscosity)	EMVIS						BEMVIS	Based on block center properties.
Property	Water Flow Rate Times Shear Multiplier (Inter-Block I+ Direction)	FLOW0I						BFLOW0I	Multiplied by the corresponding shear multiplier (PLYSHLOG and PLYSHEAR options only)
Property	Water Flow Rate Times Shear Multiplier (Inter-Block J+ Direction)	FLOW0J						BFLOW0J	
Property	Water Flow Rate Times Shear Multiplier (Inter-Block K+ Direction)	FLOW0K						BFLOW0K	
Property	Water Shear Rate After Shear Effects (Inter-Block I+ Direction)	SRTWI						BSRTWI	
Property	Water Shear Rate After Shear Effects (Inter-Block J+ Direction)	SRTWJ						BSRTWJ	
Property	Water Shear Rate After Shear Effects (Inter-Block K+ Direction)	SRTWK						BSRTWK	
Property	Water Shear Rate Prior to Shear Effects (Inter-Block I+ Direction)	SRTW0I						BSRTW0I	
Property	Water Shear Rate Prior to Shear Effects (Inter-Block J+ Direction)	SRTW0J						BSRTW0J	
Property	Water Shear Rate Prior to Shear Effects (Inter-Block K+ Direction)	SRTW0K						BSRTW0K	
Property	Water Velocity (Inter-Block I+ Direction)	VELW0I						BVELW0I	
Property	Water Velocity (Inter-Block J+ Direction)	VELW0J						BVELW0J	
Property	Water Velocity (Inter-Block K+ Direction)	VELW0K						BVELW0K	
Property	Water Viscosity (Effective)	EWV_POL						BEWV_POL	
Property	Water Viscosity Sheared Factor (Inter-Block I+ Direction)	PSHLZI						BPSHLZI	

Polymer Model Summary Variables									
Type	**Variable**	**Root**	**Field**	**Group**	**Well**	**Well Connection**	**Region**	**Block**	**Comment**
Property	Water Viscosity Sheared Factor (Inter-Block J+ Direction)	PSHLZJ						BPSHLZJ	
Property	Water Viscosity Sheared Factor (Inter-Block K+ Direction)	PSHLZK						BPSHLZK	
Volume	Polymer Concentration (Block)	CCN						BCCN	
Volume	Polymer Concentration (Injection)	CIC	FCIC	GCIC	WCIC	CCIC			
Volume	Polymer Concentration (Production)	CPC	FCPC	GCPC	WCPC	CCPC			
Volume	Polymer Solution (In Solution)	CIP	FCIP				RCIP	BCIP	
Volume	Salt Cell Concentration	SCN						BSCN	
Volume	Salt In Place	SIP	FSIP				RSIP	BSIP	
Volume	Salt inter-region Flow Total	SFT					RSFT		

Notes:

1) Cells colored in gray with a mnemonic indicate that the summary variable is not available in OPM Flow.
2) Cells colored in gray with no mnemonic indicate that the summary variable is not available for this combination.
3) Cells colored orange show combinations that are not available in OPM Flow because the underlying feature is not available.

Table 11.23: Polymer Model Summary Variables

11.2.27 OPTION SPECIFIC VARIABLES – PSEUDO STEADY STATE

This feature has not been implemented in OPM Flow.

11.2.28 OPTION SPECIFIC VARIABLES – RESERVOIR COUPLING

This feature has not been implemented in OPM Flow.

11.2.29 OPTION SPECIFIC VARIABLES – SOLVENT MODEL

The SOLVENT keyword in the RUNSPEC section activates the solvent phase in the model and activates the four component solvent model for the run. In addition to this keyword, the oil, water and gases phases should also be declared for the run using the OIL, WATER and GAS keywords. The summary variables for this option are outlined in Table 11.24.

Note that not all these variables are available in OPM Flow; however, the simulator will issue a warning messages if this is indeed the case. It is anticipated that the number of recognized summary variables will increase in future releases of OPM Flow.

Solvent Model Summary Variables									
Type	**Variable**	**Root**	**Field**	**Group**	**Well**	**Well Connection**	**Region**	**Block**	**Comment**
Flow	Solvent Flow Rate	NFR				CNFR			Field, group and wells gas rates and cumulative volumes include the solvent gas.
Flow	Solvent Flow Total (Inter-Region)	NFT					RNFT		
Flow	Solvent Injection Rate	NIR	FNIR	GNIR	WNIR				
Flow	Solvent Injection Total	NIT	FNIT	GNIT	WNIT	CNIT			
Flow	Solvent Production Rate	NPR	FNPR	GNPR	WNPR				
Flow	Solvent Production Total	NPT	FNPT	GNPT	WNPT	CNPT			
Property	Solvent Relative Permeability	NKR						BNKR	
Volume	Solvent In-Place	NIP	FNIP				RNIP	BNIP	
Volume	Solvent Saturation	NSAT						BNSAT	

Notes:

1) Cells colored in gray with a mnemonic indicate that the summary variable is not available in OPM Flow.
2) Cells colored in gray with no mnemonic indicate that the summary variable is not available for this combination.
3) Cells colored orange show combinations that are not available in OPM Flow because the underlying feature is not available.

Table 11.24: Solvent Model Summary Variables

11.2.30 OPTION SPECIFIC VARIABLES – SURFACTANT MODEL

This feature has not been implemented in OPM Flow.

11.2.31 OPTION SPECIFIC VARIABLES – THERMAL MODEL

The temperature option (TEMP keyword in the RUNSPEC section) and the thermal option (THERMAL keyword in the RUNSPEC section) are two separate modeling facilities in the commercial simulator, although some keywords can be used by both options. OPM Flow's thermal implementation is based on solving the energy equation fully coupled with the black-oil equations so the results are not directly equivalent to the commercial simulator's black-oil TEMP or compositional THERMAL formulations. The THERMAL keyword is used to invoke OPM Flow's thermal option. The summary variables for this option are listed in Table 11.25.

Note that not all these variables are available in OPM Flow; however, the simulator will issue a warning messages if this is indeed the case. It is anticipated that the number of recognized summary variables will increase in future releases of OPM Flow.

Thermal Model Summary Variables

Type	Variable	Root	Field	Group	Well	Well Connection	Region	Block	Comment
Flow	Energy Injection Rate	TIRHEA	FTIRHEA	GTIRHEA	WTIRHEA				
Flow	Energy Injection Total	TITHEA	FTITHEA	GTITHEA	WTITHEA				
Flow	Energy Production Rate	TPRHEA	FTPRHEA	GTPRHEA	WTPRHEA				
Flow	Energy Production Total	TPTHEA	FTPTHEA	GTPTHEA	WTPTHEA				
Property	Temperature (Block)	TCNFHEA						BTCNFHEA	
Property	Temperature (Injection)	TICHEA	FTICHEA	GTICHEA	WTICHEA				
Property	Temperature (Production)	TPCHEA	FTPCHEA	GTPCHEA	WTPCHEA			BTPCHEA	
Volume	Energy In-Place Difference	TIPTHEA	FTIPTHEA				RTIPTHEA 4	BTIPTHEA	Difference in energy in-place from start of run to current time.

Notes:

1) Cells colored in gray with a mnemonic indicate that the summary variable is not available in OPM Flow.
2) Cells colored in gray with no mnemonic indicate that the summary variable is not available for this combination.
3) Cells colored orange show combinations that are not available in OPM Flow because the underlying feature is not available.
4) RTIPTHEA is currently not reported as unsupported by the simulator.

Table 11.25: Thermal Model Summary Variables

See the THERMAL keyword in the RUNSPEC section for additional information on this feature.

11.2.32 OPTION SPECIFIC VARIABLES – USER DEFINE QUANTITIES

The UDQ keyword in the SCHEDULE section defines variables and operations used to access the User Defined Quantities features in OPM Flow. UDQ variables can be constants, summary variables, as defined in the SUMMARY section, or a formula using various mathematical functions together with constants and summary variables.

The variable names defined by the UDQ keyword in the SCHEDULE section consist of a character string of length eight, that stipulates the name of the user defined variable based on the type of variable. The variable type is defined by the first two characters of the variable name and must be set to one of the following:

1) CU: For variables that are associated with connections, for example summary variable COFR (Connection Oil Flow Rate).
2) FU: For variables that are associated with field data, for example summary variable FOPR (Field Oil Production Rate).
3) GU: For variables that are associated with groups, for example summary variable GLPR (Group Liquid Production Rate).
4) RU: For variables that are associated with regions, for example summary variable RPR (Region Pressure).
5) SU: For variables that are associated with multi-segment wells, for example summary variable SOFR (Segment Oil Flow Rate).
6) WU: For variables that are associated with wells, for example summary variable WWCT (Well Water Cut).
7) AU: For variables that are associated with aquifers, for example summary variable AAQP (Analytical Aquifer Pressure).
8) BU: For variables that are associated with blocks, for example summary variable BPR (Block oil phase Pressure).

The summary variable format for UDQ defined variables is presented in Table 11.26.

User Define Quantities Summary Variables							
Variable	**Root**	**Field**	**Group**	**Well**	**Well Connection**	**Region**	**Well Segment**
UDQ Variable	XXXXXX	FUXXXXXX	GUXXXXXX	WUXXXXXX	CUXXXXXX	RUXXXXXX	SUXXXXXX

Notes:

1) Where XXXXXX is the variable name defined by the UDQ keyword in the SCHEDULE section.
2) Output of aquifer (AU) and block (BU) summary variables is not supported.

Table 11.26: User Define Quantities Summary Variables

The following example illustrates the use of this type of summary variable.

Examples

The example shows how to define some constant field variables used for calculating facilities corrected condensate yields in a wet gas model in the SCHEDULE section. The corrected condensate rate is stored as variable named FU_FNGLR.

```
-- ================================================================================
--
-- SCHEDULE SECTION
--
-- ================================================================================
SCHEDULE
--
-- DEFINE START OF USER DEFINED QUANTITY SECTION
--
UDQ
--
-- OPERATOR VARIABLE  EXPRESSION
--
ASSIGN      FUNGLYLD  1.100000                / Condensate Yield Correction
--
-- OPERATOR VARIABLE  EXPRESSION
--
DEFINE      FU_FNGLR  FGPR *(FOGR * FUNGLYLD)  / Calculate Condensate Rate Field
UPDATE      FU_FNGLR  ON                       /
UNITS       FU_FNGLR  STBD                     /

/  DEFINE END OF USER DEFINED QUANTITY SECTION
```

In order to report the field corrected condensate values, one would declare FU_FNGLR in the SUMMARY section, as shown below.

```
-- ================================================================================
--
-- SUMMARY SECTION
--
-- ================================================================================
SUMMARY

FU_FNGLR
```

Note that the normal summary variables syntax rules still apply to groups, regions and wells, etc. So for example, in addition, if the same calculations were performed on two wet gas producers, GP01 and GP02, then one would use:

```
-- ================================================================================
--
-- SUMMARY SECTION
--
-- ================================================================================
SUMMARY

FU_FNGLR

WU_FNGLR
GP01
GP02
/
```

Assuming of course that WU_FNGLR was defined in the SCHEDULE section.

11.2.33 OPTION SPECIFIC VARIABLES – WELLBORE FRICTION MODEL

This feature has not been implemented in OPM Flow.

11.3 KEYWORD DEFINITIONS

11.3.1 ALL – EXPORT STANDARD SUMMARY VARIABLE VECTORS TO FILE

Description

This keyword activates the writing out of a standard set of summary production and injection data vectors for the field, group and well objects to the SUMMARY (*.SMSPEC and *.UNSMRY) and RSM (*.RSM) files. Table 11.27 lists the production, injection, pressure and volume summary variables written out by the ALL keyword, and Table 11.28 list the aquifer variables.

Standard Production, Injection, and Pressures Summary Variables

Variable	Root	Field	Group	Well	Comment
Gas Injection Rate	GIR	FGIR	GGIR	WGIR	
Gas Injection Total	GIT	FGIT	GGIT	WGIT	
Gas Production Rate	GPR	FGPR	GGPR	WGPR	Produced reservoir gas only, gas lift gas is excluded.
Gas Production Total	GPT	FGPT	GGPT	WGPT	
Oil Injection Rate	OIR	FOIR	GOIR	WOIR	
Oil Injection Total	OIT	FOIT	GOIT	WOIT	
Oil Production Rate	OPR	FOPR	GOPR	WOPR	
Oil Production Total	OPT	FOPT	GOPT	WOPT	
Res.Vol. Injection Rate	VIR	FVIR	GVIR	WVIR	
Res.Vol. Injection Total	VIT	FVIT	GVIT	WVIT	
Res.Vol. Production Rate	VPR	FVPR	GVPR	WVPR	
Res.Vol. Production Total	VPT	FVPT	GVPT	WVPT	
Water Injection Rate	WIR	FWIR	GWIR	WWIR	
Water Injection Total	WIT	FWIT	GWIT	WWIT	
Water Production Rate	WPR	FWPR	GWPR	WWPR	
Water Production Total	WPT	FWPT	GWPT	WWPT	
Bottom-Hole Pressure	BHP			WBHP	
Productivity	PI			WPI	Preferred Phase
Tubing Head Pressure	THP			WTHP	
Gas-Oil Ratio	GOR	FGOR	GGOR	WGOR	
Water -Gas Ratio	WGR	FWGR	GWGR	WWGR	
Water Cut	WCT	FWCT	GWCT	WWCT	

Standard Production, Injection, and Pressures Summary Variables					
Pressures and Volumes					
Variable	**Root**	**Field**	**Group**	**Well**	**Comment**
Oil Phase Pressure	PR	FPR			Field and Region HCPV weighted
Gas In-Place	GIP	FGIP			Liquid and gas phases
Gas In-Place (Gas Phase)	GIPG	FGIPG			
Gas In-Place (Liquid Phase)	GIPL	FGIPL			Only for cases that have dissolved gas in the water phase
Oil In-Place	OIP	FOIP			Liquid and gas phases
Oil In-Place (Gas Phase)	OIPG	FOIPG			
Oil In-Place (Liquid Phase)	OIPL	FOIPL			
Water In-Place	WIP	FWIP			

Notes:

1) Cells colored in gray indicate that the summary variable is not available for this combination.
2) Cells colored orange under the Type column indicate that this set of summary variables are not available in OPM Flow because the underlying feature is not available in OPM Flow.

Table 11.27: Standard Production, Injection, and Pressure Summary Variables

Standard Aquifer Summary Variables						
Variable	**Root**	**Field**	**Analytical Aquifer**	**Analytical Aquifer List**	**Numerical Aquifer**	**Comment**
Aquifer Influx Rate (Water Aquifers)	QR	FAQR	AAQR			
Aquifer Influx Total (Water Aquifers)	QT	FAQT	AAQT			
Aquifer Influx Rate (Gas Aquifers)	QRG	FAQRG	AAQRG			
Aquifer Influx Total (Gas Aquifers)	QTG	FAQTG	AAQTG			

Notes:

1) Cells colored in gray with a mnemonic indicate that the summary variable is not available in OPM Flow.
2) Cells colored in gray with no mnemonic indicate that the summary variable is not available for this combination.
3) Cells colored orange show combinations that are not available in OPM Flow because the underlying feature is not available.

Table 11.28: Standard Aquifer Summary Variables

There is no data required for this keyword and there is no terminating "/" for this keyword.

Examples

```
-- ===============================================================================
--
-- SUMMARY SECTION
--
-- ===============================================================================
SUMMARY
--
--       EXPORT STANDARD SUMMARY VARIABLE VECTORS TO FILE
--
ALL
--
--       ACTIVATE COLUMNAR SUMMARY DATA REPORTING OPTION
--
RUNSUM
--
--       ACTIVATE SUMMARY DATA RSM FILE OUTPUT OPTION
--
SEPARATE
```

Note the SEPARATE keyword is not required for OPM Flow as this is the default behavior; however, it is probably good practice to include it if the same input decks are being run with commercial simulator.

11.3.2 COLUMNS – DEFINE INPUT FILE COLUMN MARGINS

The COLUMNS keyword defines the input file column margins; characters outside the margins are ignored by the input parser.

See *COLUMNS – Define Input File Column Margins* in the GLOBAL section for a full description.

11.3.3 DATE - ACTIVATE THE DATE OPTION FOR THE SUMMARY FILE

Description

This keyword activates the writing out of the date of each time step to the SUMMARY file. Normally only the time in days and decimal years are written out to the SUMMARY file, activating the DATE option also results in the DATE being written out to the SUMMARY file as well. This option is normally used when the RUNSUM keyword in the SUMMARY section has been activated to produce a RSM file.

There is no data required for this keyword and there is no terminating "/" for this keyword.

Example

The following example shows an example RSM file output when the DATE option has not been activated.

```
 ---------------------------------------------------------------------------
 SUMMARY OF RUN NO-DATE-KEYWORD
 ---------------------------------------------------------------------------
 TIME          YEARS         FPR           FOEW          FOPR          FOPT
 DAYS          YEARS         PSIA                        STB/DAY       STB



 ---------------------------------------------------------------------------
        0             0     4467.125             0             0             0
 1.000000      0.002738     4466.943      0.000239      3235.662      3235.662
 31.00000      0.084873     4464.476      0.007407      3230.117      100256.4
 60.00000      0.164271     4462.717      0.014291      3193.902      193421.5
 91.00000      0.249144     4460.813      0.021523      3127.557      291306.3
 121.0000      0.331280     4458.909      0.028362      3055.878      383879.7
 152.0000      0.416153     4456.914      0.035262      2982.212      477271.4
.....
```

And activating the SUMMARY file DATE option with:

```
--
--        ACTIVATE DATE SUMMARY FILE OPTION
--
DATE
```

Results in the following example RSM file output.

```
 ---------------------------------------------------------------------------
 SUMMARY OF RUN WITH-DATE-KEYWORD
 ---------------------------------------------------------------------------
 DATE          YEARS     DAY   MONTH   YEAR   FPR           FOEW          FOPR
               YEARS                          PSIA                        STB/DAY



 ---------------------------------------------------------------------------
  1-JAN-98           0   19    10      1992   4467.125            0             0
  2-JAN-98    0.002738   20    10      1992   4466.943      0.000239      3235.662
 31-JAN-98    0.084873   21    10      1992   4464.476      0.007407      3230.117
 28-FEB-98    0.164271   24    10      1992   4462.717      0.014291      3193.902
 31-MAR-98    0.249144   28    10      1992   4460.813      0.021523      3127.557
 30-APR-98    0.331280    3    11      1992   4458.909      0.028362      3055.878
 31-MAY-98    0.416153   14    11      1992   4456.914      0.035262      2982.212
.....
```

11.3.4 DEBUG – DEFINE THE DEBUG DATA TO BE PRINTED TO FILE

This keyword defines the debug data to be written to the debug file (*.DBG). This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

See *DEBUG – Define the Debug Data to be Printed to File* in the GLOBAL section for a full description.

11.3.5 ECHO – ACTIVATE ECHOING OF USER INPUT FILES TO THE PRINT FILE

Turns on echoing of all the input files to the print file; note that this keyword is activated by default and can subsequently be switched off by the NOECHO activation keyword.

See *ECHO – Activate Echoing of User Input Files to the Print File* in the GLOBAL section for a full description.

11.3.6 END – DEFINE THE END OF THE INPUT FILE

This keyword marks the end of the input file and can occur in any section. Any keywords and data after the END keyword are ignored.

See *END – Define the End of the Input File* in the GLOBAL section for a full description.

11.3.7 ENDINC – DEFINE THE END OF AN INCLUDE FILE

This keyword marks the end of an include file specified on the INCLUDE keyword. When the ENDINC keyword is encountered in the INCLUDE file, input data is read from the next keyword in the current file. Any keywords and data after the ENDINC keyword in the INCLUDE file are ignored.

See *ENDINC – Define the End of an Include File* in the GLOBAL section for a full description.

11.3.8 ENDSKIP – DEACTIVATE SKIPPING OF KEYWORDS AND INPUT DATA

Turns off skipping of keywords that was activated by the SKIP, SKIP100, or SKIP300 keywords. Note that for each SKIP keyword activated there must be a corresponding ENDSKIP keyword.

See *ENDSKIP – DeActivate Skipping of Keywords and Input Data* in the GLOBAL section for a full description.

11.3.9 EXCEL - ACTIVATE THE EXCEL OPTION FOR THE SUMMARY FILE

Description

This keyword activates the writing out of the RSM file data in a format that can easily be loaded into Microsoft's EXCEL spreadsheet program or LibreOffice's CALC spreadsheet program. The RSM file output is activated by the RUNSUM keyword in the SUMMARY section.

There is no data required for this keyword and there is no terminating "/" for this keyword.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

Examples

```
-- ==========================================================================
--
-- SUMMARY SECTION
--
-- ==========================================================================
SUMMARY
--
--       ACTIVATE COLUMNAR SUMMARY DATA REPORTING OPTION
--
RUNSUM
--
--       ACTIVATE EXCEL SUMMARY FILE OPTION
--
EXCEL
```

The above example activates the SUMMARY file EXCEL option for directly loading the RSM file into either Microsoft's EXCEL or LibreOffice's CALC spreadsheet programs.

11.3.10 EXTRAPMS – ACTIVATE EXTRAPOLATION WARNING MESSAGES

The EXTRAPMS keyword activates extrapolation warning messages for when OPM Flow extrapolates the PVT or VFP tables. Frequent extrapolation warning messages should be investigated and resolved as this would indicate possible incorrect data and may result in the simulator extrapolating to unrealistic values.

See *EXTRAPMS – Activate Extrapolation Warning Messages* in the GLOBAL section for a full description.

11.3.11 FMWSET - EXPORT WELL STATUS VECTORS FOR THE FIELD TO FILE

Description

This keyword is similar to the ALL keyword in the SUMMARY section, in that it results in a group of summary variables to be written out to the SUMMARY file. In this case the keyword activates the writing out of a set of data vectors that give the production and injections status of all the wells in the model, as well as the number of wells in the drilling queue and the number of workover events occurring within a time step. Both instantaneous and cumulative well counts and events are written out as listed in Table 11.29.

There is no data required for this keyword.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

Field and Group Well Status Summary Variables

Variable	Root	Field	Group	Comment
Number of abandoned injection wells	MWIA	FMWIA	GMWIA	
Number of abandoned production wells	MWPA	FMWPA	GMWPA	
Number of drilling events in total	MWDT	FMWDT	GMWDT	
Number of drilling events this time step	MWDR	FMWDR	GMWDR	
Number of injection wells currently flowing	MWIN	FMWIN	GMWIN	
Number of injectors on group control	MWIG	FMWIG	GMWIG	
Number of injectors on own reservoir volume rate limit control	MWIV	FMWIV	GMWIV	
Number of injectors on own surface rate limit control	MWIS	FMWIS	GMWIS	
Number of injectors on pressure control	MWIP	FMWIP	GMWIP	
Number of producers controlled by own oil rate limit	MWPO	FMWPO	GMWPO	
Number of producers on group control	MWPG	FMWPG	GMWPG	
Number of producers on own reservoir volume rate limit control	MWPV	FMWPV	GMWPV	
Number of producers on own surface rate limit control	MWPS	FMWPS	GMWPS	
Number of producers on pressure control	MWPP	FMWPP	GMWPP	
Number of producers using artificial lift (with ALQ > 0.0)	MWPL	FMWPL	GMWPL	
Number of production wells currently flowing	MWPR	FMWPR	GMWPR	
Number of unused injection wells	MWIU	FMWIU	GMWIU	
Number of unused production wells	MWPU	FMWPU	GMWPU	
Number of workover events in total	MWWT	FMWWT	GMWWT	
Number of workover events this time step.	MWWO	FMWWO	GMWWO	
Total number of injection wells	MWIT	FMWIT	GMWIT	
Total number of production wells	MWPT	FMWPT	GMWPT	

Field and Group Well Status Summary Variables				
Variable	Root	Field	Group	Comment

Notes:

1) Cells colored in gray with a mnemonic indicate that the summary variable is not available in OPM Flow.
2) Cells colored in gray with no mnemonic indicate that the summary variable is not available for this combination.
3) Cells colored orange show combinations that are not available in OPM Flow because the underlying feature is not available.

Table 11.29: FMWSET - Standard Field and Group Well Status Summary Variables

See also the GMWSET keyword in the SUMMARY schedule that has similar functionality but at a group level.

Example

```
-- ==========================================================================
--
-- SUMMARY SECTION
--
-- ==========================================================================
SUMMARY
--
--       EXPORT WELL STATUS VECTORS FOR THE FIELD TO FILE
--
FMWSET
--
--       ACTIVATE COLUMNAR SUMMARY DATA REPORTING OPTION
--
RUNSUM
--
--       ACTIVATE SUMMARY DATA RSM FILE OUTPUT OPTION
--
SEPARATE
```

The above example exports the field standard well status variables to the SUMMARY file.

11.3.12 FORMFEED – DEFINED THE PRINT FILE FORM-FEED CHARACTER

The FORMFEED keyword defines the form-feed character, or carriage control character, for the output print (*.PRT) run summary (*.RSM) files. The keyword should be place at the very top of the input file.

See *FORMFEED – Defined the Print File Form-Feed Character* in the GLOBAL section for a dull description.

11.3.13 GMWSET - EXPORT WELL STATUS VECTORS BY GROUP TO FILE

Description

This keyword is similar to the ALL keyword in the SUMMARY section, in that it results in a group of summary variables to be written out to the SUMMARY file. In this case the keyword activates the writing out of a set of data vectors that give the production and injections status of all the wells in the named groups, as well as the number of wells in the drilling queue and the number of workover events occurring within a time step for the requested groups. Both instantaneous and cumulative well counts and events for the groups are written out as tabulated in Table 11.30.

Note that GMWSET should be followed by a list of group names enclosed in quotes and therefore a terminating "/" is required to end the list of groups. A blank list requests output for all groups.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

Field and Group Well Status Summary Variables

Variable	Root	Field	Group	Comment
Number of abandoned injection wells	MWIA	FMWIA	GMWIA	
Number of abandoned production wells	MWPA	FMWPA	GMWPA	
Number of drilling events in total	MWDT	FMWDT	GMWDT	
Number of drilling events this timestep	MWDR	FMWDR	GMWDR	
Number of injection wells currently flowing	MWIN	FMWIN	GMWIN	
Number of injectors on group control	MWIG	FMWIG	GMWIG	
Number of injectors on own reservoir volume rate limit control	MWIV	FMWIV	GMWIV	
Number of injectors on own surface rate limit control	MWIS	FMWIS	GMWIS	
Number of injectors on pressure control	MWIP	FMWIP	GMWIP	
Number of producers controlled by own oil rate limit	MWPO	FMWPO	GMWPO	
Number of producers on group control	MWPG	FMWPG	GMWPG	
Number of producers on own reservoir volume rate limit control	MWPV	FMWPV	GMWPV	
Number of producers on own surface rate limit control	MWPS	FMWPS	GMWPS	
Number of producers on pressure control	MWPP	FMWPP	GMWPP	
Number of producers using artificial lift (with ALQ > 0.0)	MWPL	FMWPL	GMWPL	
Number of production wells currently flowing	MWPR	FMWPR	GMWPR	
Number of unused injection wells	MWIU	FMWIU	GMWIU	
Number of unused production wells	MWPU	FMWPU	GMWPU	
Number of workover events in total	MWWT	FMWWT	GMWWT	
Number of workover events this time step.	MWWO	FMWWO	GMWWO	
Total number of injection wells	MWIT	FMWIT	GMWIT	

Field and Group Well Status Summary Variables				
Variable	Root	Field	Group	Comment
Total number of production wells	MWPT	FMWPT	GMWPT	

Notes:

1) Cells colored in gray with a mnemonic indicate that the summary variable is not available in OPM Flow.
2) Cells colored in gray with no mnemonic indicate that the summary variable is not available for this combination.
3) Cells colored orange show combinations that are not available in OPM Flow because the underlying feature is not available.

Table 11.30: GMWSET - Standard Field and Group Well Status Summary Variables

See also the FMWSET keyword in the SUMMARY schedule that has similar functionality but at the field level.

Examples

The first example below exports all the group standard well status variables to the SUMMARY file.

```
-- ================================================================================
--
-- SUMMARY SECTION
--
-- ================================================================================
SUMMARY
--
--       EXPORT WELL STATUS VECTORS FOR NAMED GROUPS TO FILE
--
GMWSET
/
--
--       ACTIVATE COLUMNAR SUMMARY DATA REPORTING OPTION
--
RUNSUM
--
--       ACTIVATE SUMMARY DATA RSM FILE OUTPUT OPTION
--
SEPARATE
```

The second example exports all the group standard well status variables for just the PLAT1 and PLT2 groups only to the SUMMARY file.

```
--
--       EXPORT WELL STATUS VECTORS FOR NAMED GROUPS TO FILE
--
GMWSET
'PLAT1' 'PLAT2'
/
--
--       ACTIVATE COLUMNAR SUMMARY DATA REPORTING OPTION
--
RUNSUM
--
--       ACTIVATE SUMMARY DATA RSM FILE OUTPUT OPTION
--
SEPARATE
```

11.3.14 INCLUDE – LOAD ANOTHER DATA FILE AT THE CURRENT POSITION

The INCLUDE keyword informs OPM Flow to continue reading input data from the specified INCLUDE file. When the end of the INCLUDE file is reached, or the ENDINC keyword is encountered in the included file, input data is read from the next keyword in the current file.

See *INCLUDE – Load Another Data File at the Current Position* in the GLOBAL section for a full description.

11.3.15 MESSAGE – OUTPUT USER MESSAGE

The MESSAGE keyword outputs a user message to the terminal, as well as to the print (*.PRT) and debug (*.DBG) files. Note this is different to the MESSAGES keyword, that defines OPM Flows message print limits and stop limits generated by the simulator.

See *MESSAGE – Output User Message* in the GLOBAL section for a full description.

11.3.16 MESSAGES – DEFINE MESSAGE PRINT LIMITS AND STOP LIMITS

The MESSAGES keyword defines the print and stops levels for various messages. The "print limits" set the maximum number of messages that will be printed, after which no more messages will be printed and the "stop limits" terminate the run when these limits are exceeded. There are six levels of message that increase in severity from informative all the way to programming errors, as outlined in Error: Reference source not found.

See *MESSAGES – Define Message Print Limits and Stop Limits* in the GLOBAL section for a full description.

11.3.17 MONITOR – ACTIVATE OUTPUT OF THE MONITORING DATA AND FILE

The MONITOR keyword activates the writing out of the run time monitoring information used by post-processing graphics software to display run time information, for example the simulated production and injection rates and cumulative values. OPM Flow does not have this functionality.

See *MONITOR – Activate Output of the Monitoring Data and File* in the RUNSPEC section for a full description.

11.3.18 NARROW – ACTIVATE RUN SUMMARY NARROW COLUMN OUTPUT OPTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The NARROW keyword activates the Run Summary Narrow Column Output option, for when printed SUMMARY data has been requested by the RUNSUM keyword in the SUMMARY section. The option increases the number of columns "printed on the page".

There is no data required for this keyword and there is no terminating "/" for this keyword.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

11.3.19 NEWTON – ACTIVATE NEWTON ITERATION SUMMARY OUTPUT

Description

This keyword activates the writing out of the Newton iteration vector (the number of non-linear iterations per time step) to the SUMMARY file, and the RSM file if the RSM file option has been requested by the RUNSUM keyword in the SUMMARY section,

There is no data required for this keyword and there is no terminating "/" for this keyword.

Although the keyword is recognized by OPM Flow only zeros are written to the SUMMARY file.

Example

```
-- ================================================================================
--
-- SUMMARY SECTION
--
-- ================================================================================
SUMMARY
--
--       ACTIVATE COLUMNAR SUMMARY DATA REPORTING OPTION
--
RUNSUM
--
--       ACTIVATE SUMMARY DATA RSM FILE OUTPUT OPTION
--
SEPARATE       --
--       ACTIVATE NEWTON ITERATION SUMMARY OUTPUT
--
NEWTON
```

The above example actives the writing out of the Newton iteration vector to the SUMMARY file.

11.3.20 NMESSAGE – EXPORT CUMULATIVE MESSAGE SUMMARY VARIABLES TO FILE

Description

This keyword activates the writing out of a standard set of summary OPM Flow simulation performance summary variables to the SUMMARY (*.SMSPEC and *.UNSMRY) and RSM (*.RSM) files, namely the number of messages written per message class. Error: Reference source not found lists the summary variables written out by the NMESSAGE keyword.

The keyword is recognized by OPM Flow but none of the variables are currently supported.

OPM Flow Simulator Performance Summary Variables

Cumulative Message Variables

Variable Description	Variable	Comment
Messages - Cumulative number of BUG messages.	MSUMBUG	Unsupported.
Messages - Cumulative number of COMMENT messages.	MSUMCOMM	Unsupported.
Messages - Cumulative number of ERROR messages.	MSUMERR	Unsupported.
Messages - Cumulative number of MESSAGES messages.	MSUMMESS	Unsupported.
Messages - Cumulative number of PROBLEM messages.	MSUMPROB	Unsupported.
Messages - Cumulative number of WARNING messages.	MSUMWARN	Unsupported.

Notes:

1) Cells under the Variable column not colored indicate that the summary variable is available in OPM Flow.
2) Cells under the Variable column colored in gray indicate that the summary variable is recognized by the parser but the summary variable is not available.
3) Cells under the Variable column colored in orange indicate that the summary variable is not recognized by the parser and the summary variable is not available. These summary variables may cause the simulator to abort.

Table 11.31: Simulator Performance Summary Variables (Cumulative Messages)

Example

```
-- ================================================================================
--
-- SUMMARY SECTION
--
-- ================================================================================
SUMMARY
--
--       EXPORT PERFORMANCE CUMULATIVE MESSAGE VARIABLE VECTORS TO FILE
--
NMESSAGE
--
--       ACTIVATE COLUMNAR SUMMARY DATA REPORTING OPTION
--
RUNSUM
--
--       ACTIVATE SUMMARY DATA RSM FILE OUTPUT OPTION
--
SEPARATE
```

Note the SEPARATE keyword is not required for OPM Flow as this is the default behavior; however, it is probably good practice to include it if the same input decks are being run with the commercial simulator.

11.3.21 NOECHO – DEACTIVATE ECHOING OF USER INPUT FILES TO THE PRINT FILE

Turns off echoing of all the input files to the print file. Note by default echoing of the inputs files is active. but can subsequently be switched off by the NOECHO activation keyword.

See *NOECHO – Deactivate Echoing of User Input Files to the Print File* in the GLOBAL section for a full description.

11.3.22 NOMONITO – DEACTIVATE OUTPUT OF THE MONITORING DATA AND FILE

The NOMONITO keyword deactivates the writing out of the run time monitoring information used by post-processing graphics software to display run time information, for example the simulated production and injection rates and cumulative values. OPM Flow does not have this functionality.

See *NOMONITO – Deactivate Output of the Monitoring Data and File* in the RUNSPEC section for a full description.

11.3.23 NOWARN – DEACTIVATE WARNING MESSAGES

Turns off warning messages to be printed to the print file; note that this keyword is deactivated by default and can subsequently be switched off by the WARN activation keyword. The warning messages may be turned on and off using keywords WARN and NOWARN.

See *NOWARN – Deactivate Warning Messages* in the GLOBAL section for a full description.

11.3.24 OFM – ACTIVATE OFM FILE OUTPUT OF THE SUMMARY DATA

Description

This keyword activates the writing out of the SUMMARY file data in the Oil Field Manager ("OFM") file format to enable the simulated data to be directly loaded into OFM.

There is no data required for this keyword and there is no terminating "/" for this keyword.

See also the EXCEL, RPTONLY, RUNSUM and SEPARATE keywords in the SUMMARY section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

11.3.25 PERFORMA – EXPORT STANDARD SIMULATOR PERFORMANCE SUMMARY VARIABLES TO FILE

Description

The PERFORMA keyword activates the writing out of a standard set of summary OPM Flow simulation numerical performance summary variables to the SUMMARY (*.SMSPEC and .*UNSMRY) and RSM (*.RSM) files. Error: Reference source not found lists the summary variables written out by the keyword.

Note that not all these variables are available in OPM Flow; however, the simulator will issue a warning message if this is indeed the case. It is anticipated that the number of recognized summary variables will increase in future releases of OPM Flow.

OPM Flow Simulator Performance Summary Variables Numerical Performance Variables		
Variable Description	**Variable**	**Comment**
Elapsed - Elapsed time in seconds.	ELAPSED	No data written to file.
Iterations - Number linear iterations for each time step.	MLINEARS	
Iterations - Cumulative number of linear iterations.	MSUMLINS	
Iterations - Cumulative number of Newton iterations.	MSUMNEWT	
Iterations - Number of Newton iterations used for each time step.	NEWTON	
Iterations - Average number of linear iterations per Newton iteration for each time step.	NLINEARS	For runs with LGRs, LLINEARS will automatically be exported for each LGR.
Iterations - Maximum number of linear iterations in the Newton iterations per time step.	NLINSMAX	
Iterations - Minimum number of linear iterations in the Newton iterations per time step.	NLINSMIN	
Time Step – Criteria used to select the length of the time step. See section *11.2.25 Option Specific Variables – OPM Flow Simulation Performance* for the definition of of the STEPTYPE mnemonics.	STEPTYPE	No data written to file.
CPU - Current CPU usage in seconds.	TCPU	Does not consider the time taken by inter-process communications in parallel runs, whereas ELAPSED does. Thus, for parallel jobs, ELAPSED is the most relevant time measurement.
CPU - CPU time per day (or hour in lab units depending on run units system).	TCPUDAY	No data written to file.
CPU - CPU time per time step in seconds.	TCPUTS	No data written to file.

OPM Flow Simulator Performance Summary Variables Numerical Performance Variables		
Variable Description	**Variable**	**Comment**
Elapsed - Elapsed time per linear iteration in seconds.	TELAPLIN	No data written to file.
Time Step - Length of time step.	TIMESTEP	

Notes:

1) Cells under the Variable column not colored indicate that the summary variable is available in OPM Flow.
2) Cells under the Variable column colored in gray indicate that the summary variable is recognized by the parser but the summary variable is not available.
3) Cells under the Variable column colored in orange indicate that the summary variable is not recognized by the parser and the summary variable is not available. These summary variables may cause the simulator to abort.

Table 11.32: Simulator Performance Summary Variables (Numerical Performance)

Note

If the summary vector data is unavailable then zeros are written at each time step to the SUMMARY and RSM file.

Example

```
-- ================================================================================
--
-- SUMMARY SECTION
--
-- ================================================================================
SUMMARY
--
--       EXPORT NUMERICAL PERFORMANCE SUMMARY VARIABLES TO FILE
--
PERFORMA
--
--       ACTIVATE COLUMNAR SUMMARY DATA REPORTING OPTION
--
RUNSUM
--
--       ACTIVATE SUMMARY DATA RSM FILE OUTPUT OPTION
--
SEPARATE
```

Note the SEPARATE keyword is not required for OPM Flow as this is the default behavior; however, it is probably good practice to include it if the same input decks are being run with commercial simulator.

11.3.26 PYEND – END THE DEFINITION OF A PYINPUT SECTION

The PYINPUT and PYEND keywords are a part of OPM Flow's Python scripting facility that processes standard Python commands that can be used to manipulate and define the simulators input parameters during processing of the input deck. The main purpose of the facility is to script the construction of the various keywords. A PYINPUT Definition Section is terminated by a PYEND keyword on a separate single line.

See *PYEND – End the Definition of a PYINPUT Section* in the GRID section for a full description.

11.3.27 PYINPUT – DEFINE THE START OF A PYINPUT SECTION

The PYINPUT and PYEND keywords are a part of OPM Flow's Python scripting facility that processes standard Python commands that can be used to manipulate and define the simulators input parameters during processing of the input deck. The main purpose of the facility is to script the construction of the various keywords used by the simulator. PYINPUT declares the start of a PYINPUT Definition Section on a single separate line, which is then followed by various standard Python commands, with one command per line.

See *PYINPUT – Define the Start of a PYINPUT Section* in the GRID section for a full description.

11.3.28 RPTONLY - ACTIVATE THE REPORT TIME STEPS ONLY OPTION FOR THE SUMMARY FILE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword activates the writing out of the SUMMARY file and the RSM file data, if the RSM file option has been requested by the RUNSUM keyword in the SUMMARY section, at report time steps only. The default is for the data to be written out for all time steps to the SUMMARY files. This keyword reduces the file size at the expense of lower resolution in the time domain.

There is no data required for this keyword and there is no terminating "/" for this keyword.

The option can be deactivated by the RPTONLYO keyword in the SUMMARY section.

Example

```
-- ==================================================================================
--
-- SUMMARY SECTION
--
-- ==================================================================================
SUMMARY
--
--        EXPORT STANDARD SUMMARY VARIABLE VECTORS TO FILE
--
ALL
--
--        ACTIVATE COLUMNAR SUMMARY DATA REPORTING OPTION
--
RUNSUM
--
--        ACTIVATE SUMMARY DATA RSM FILE OUTPUT OPTION
--
SEPARATE
--
--        ACTIVATE REPORT TIME STEPS ONLY SUMMARY FILE OPTION
--
RPTONLY
```

The above example activates the writing out of the SUMMARY file at report time steps only.

11.3.29 RPTONLYO - DEACTIVATE THE REPORT TIME STEPS ONLY OPTION FOR THE SUMMARY FILE

Description

This keyword deactivates the writing out of the SUMMARY file and the RSM file data, if the RSM file option has been requested by the RUNSUM keyword in the SUMMARY section, at report time steps only, and switches on writing out all the time steps to the files. This option is the default behavior for when RPTONLY has not been activated.

There is no data required for this keyword and there is no terminating "/" for this keyword.

The option can be deactivated by the RPTONLY keyword in the SUMMARY section that will switch on writing the data at every report time step instead of every time step.

Example

```
-- ================================================================================
--
-- SUMMARY SECTION
--
-- ================================================================================
SUMMARY
--
--        EXPORT STANDARD SUMMARY VARIABLE VECTORS TO FILE
--
ALL
--
--        ACTIVATE COLUMNAR SUMMARY DATA REPORTING OPTION
--
RUNSUM
--
--        ACTIVATE SUMMARY DATA RSM FILE OUTPUT OPTION
--
SEPARATE
--
--        DEACTIVATE REPORT TIME STEPS ONLY SUMMARY FILE OPTION
--
RPTONLYO
```

The above example deactivates the writing out of the SUMMARY file at report time steps only, and switches on writing out all the time steps to the file.

11.3.30 RPTSMRY - ACTIVATE OR DEACTIVATE SUMMARY LIST REPORT

Description

This keyword activates or deactivates a listing of all the summary variables that are going to be written to the SUMMARY file and the RSM file, if the RSM file option has been requested by the RUNSUM keyword in the SUMMARY section.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

No.	Name	Description	Default
1	RPTSMRY	An integer value set to zero for no report, or one to produce the report.	0

Notes:

1) The keyword is terminated by a "/".

Table 11.33: RPTSMRY Keyword Description

Example

```
-- ======================================================================
--
-- SUMMARY SECTION
--
-- ======================================================================
SUMMARY
--
--        EXPORT STANDARD SUMMARY VARIABLE VECTORS TO FILE
--
ALL
--
--        ACTIVATE COLUMNAR SUMMARY DATA REPORTING OPTION
--
RUNSUM
--
--        ACTIVATE SUMMARY DATA RSM FILE OUTPUT OPTION
--
SEPARATE
--
--        ACTIVATE OR DEACTIVATE SUMMARY LIST REPORT
--
RPTSMRY
         1                                                  /
```

The example switches on the summary list report.

11.3.31 RUNSUM – ACTIVATE RSM FILE OUTPUT OF THE SUMMARY DATA

Description

This keyword activates the writing out of the SUMMARY file data in a columnar format to the PRT file. Normally the SEPARATE keyword in the SUMMARY section is invoked in the same run to direct the data stream to a separate RSM file for easy loading into other programs, for example, Microsoft's EXCEL or LibreOffice's CALC spreadsheet programs.

There is no data required for this keyword and there is no terminating "/" for this keyword.

See also the EXCEL, RPTONLY and SEPARATE keywords in the SUMMARY section.

Example

```
-- ==========================================================================
--
-- SUMMARY SECTION
--
-- ==========================================================================
SUMMARY
--
--        ACTIVATE COLUMNAR SUMMARY DATA REPORTING OPTION
--
RUNSUM
--
--        ACTIVATE SUMMARY DATA RSM FILE OUTPUT OPTION
--
SEPARATE
```

Note unlike the commercial simulator, OPM Flow always writes out the data to a separate file.

11.3.32 SEPARATE – ACTIVATE THE SEPARATE RSM FILE OUTPUT OPTION

Description

This keyword activates the writing out of the SUMMARY file date in a columnar format to the RSM file, if the RUNSUM keyword has also been activated in the SUMMARY section. Both the SEPARATE and the RUNSUM keywords need to be invoked. If the SEPARATE option is not activated then the RSM output is directed to the end of the PRT file. Normally the both the SEPARATE and RUNSUM keywords are invoked in the same run to enable easy loading of the data into Microsoft's EXCEL or LibreOffice's CALC spreadsheet programs.

There is no data required for this keyword and there is no terminating "/" for this keyword.

See also the EXCEL, RPTONLY and RUNSUM keywords in the SUMMARY section.

Example

```
-- ================================================================================
--
-- SUMMARY SECTION
--
-- ================================================================================
SUMMARY
--
--        ACTIVATE COLUMNAR SUMMARY DATA REPORTING OPTION
--
RUNSUM
--
--        ACTIVATE SUMMARY DATA RSM FILE OUTPUT OPTION
--
SEPARATE
```

Note unlike the commercial simulator, OPM Flow always writes out the data to a separate file; however, it is probably good practice to include it if the same input decks are being run with commercial simulator.

11.3.33 SUMMARY - DEFINE THE START OF THE SUMMARY SECTION OF KEYWORDS

Description

The SUMMARY activation keyword marks the end of the SOLUTION section and the start of the SUMMARY section that defines the variables to be written out to the SUMMARY file for reporting and plotting of grid block data, production data, etc.

There is no data required for this keyword and there is no terminating "/" for this keyword.

Example

```
-- ================================================================================
--
-- SUMMARY SECTION
--
-- ================================================================================
SUMMARY
```

The above example marks the end of the SOLUTION section and the start of the SUMMARY section in the OPM Flow data input file.

11.3.34 SUMTHIN – DEFINE SUMMARY DATA REPORTING TIME STEPS

Description

This keyword defines a time interval for writing out the SUMMARY data to the SUMMARY file and the RSM file, if the RUNSUM keyword has also been activated in the SUMMARY section. Only the data for the first time step in the time interval is written out and the other time steps are skipped until the next time interval, except for report time steps. Note that report time steps data are always written out regardless of the setting on this keyword. This enables the size of the SUMMARY files to be reduced depending on the size of the time interval. However, the keyword will produce irregular time steps reports of the SUMMARY data.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SUMSTEP	SUMSTEP is a real positive number that defines the time interval for which the first time step of data will be written to the SUMMARY file (and the RSM file if RSM output has been activated). For example, if SUMSTEP is set to 30 days, and if the simulator takes time steps of 0, 5, 10, 16, 24, 30, 40, 45, 60, 90 days. Then the SUMMARY data will be written out at time steps 0, 30, 40 and 60 days.			
		days	days	hours	None
Notes: 1) The keyword is terminated by a "/".					

Table 11.34: SUMTHIN Keyword Description

See also the RPTONLY keyword in the SUMMARY section that forces the SUMMARY data to be only written out at report time steps, as oppose to all time steps or SUMSTEP time intervals.

Example

```
--
--        DEFINE SUMMARY DATA REPORTING TIME STEP INTERVAL
--
--        SUMSTEP
SUMTHIN
          30.0                                                  /
/
```

The above example defines the SUMMARY file time step interval to be 30 days for both field and metric units.

11.3.35 WARN – ACTIVATE WARNING MESSAGES

Turns on warning messages to be printed to the print file (*.PRT); note that this keyword is activated by default and can subsequently be switched off by the NOWARN activation keyword. The warning messages may be turned on and off using keywords WARN and NOWARN. OPM Flow always prints error messages.

See *WARN – Activate Warning Messages* in the GLOBAL section for a full description.

CHAPTER 12: SCHEDULE SECTION

12.1 INTRODUCTION

The SCHEDULE section is used to specify the production and injection targets and constraints for the entire model, advance the model through time, and stipulate any other data that depends on time. For example, drilling wells at a particular time, incorporating production and/or injection rate changes as the model progresses through time, and opening and shutting wells and well connections/completions at particular times during the history of the reservoir. Field development and operating strategy are also incorporated in the SCHEDULE section, for example the timing and effect of upgrading of the field facilities liquid and water handling capacities. All these actions must be specified in the SCHEDULE section at the time they occur, that is all the events in the SCHEDULE section are time dependent. This is different to the other sections of the input deck were the order of keywords is largely unimportant; in the SCHEDULE section it is essential because the order in which events take place in the field have to be preserved. For instance, the keyword defining wellhead locations via the WELSPECS keyword must precede the keyword defining the locations and properties of the well connections using the COMPDAT keyword. The rules on keyword order generally follow the same sequence as the order of events in a production plan, that is we drill wells first (WELSPECS), then complete them (COMPDAT), and put them on production (WCONHIST, WCONPROD or WCONINJE).

In most cases a significant part of the contents of the SCHEDULE section is often imported from other applications. For instance, the Vertical Lift Performance ("VLP") tables used to convert from Bottom-Hole Pressure ("BHP") to Tubing Head Pressure ("THP") and entered via the VFPPROD keyword, are always generated by an external Nodal Analysis application. These tables are then included in the input deck via the INCLUDE keyword. Note as the VFP tables are assigned to a well via the WCONHIST, WCONPROD or WCONINJE keywords, then the tables need to have been previously entered prior to assigning them to a well, that is the data is order dependent. Well historical production data is also normally "included" from an external source, typically the production data base, and may have to be edited to ensure that the correct connections/completions are opened and shut at the appropriate time.

Note

The term well connection is used to describe individual connections from the wellbore to the reservoir grid cell, as opposed to a well completion. A well completion is used to describe a set of connections that are "grouped" into well completions using the COMPLUMP keyword in the SCHEDULE section. For instance, a well may consist of several completions with each completion consisting of multiple connections.

The reservoir management flexibility offered in the SCHEDULE section is indicated by over 300 keywords available in the section and thus makes quality assurance of the data challenging. This means that one should not assume that the data included from third party applications are error free and therefore suitable error checking should be performed.

The majority of the keywords are related to controlling how wells and groups are operated, by applying production/injection targets and constraints, economic criteria and associated actions if a criteria is violated, drilling of additional wells to meet production targets or constraints, as well as the relationship between the different wells and groups in the model. In addition, the DATES, TIME and TSTEP keywords that are used to progress the simulation through time and the TUNING series of keywords control the numerical parameters in defining numerical convergence of each time step. Note that OPM Flow can be instructed to read the first record of the TUNING keyword if the appropriate command line parameter has been activated (see section *2.2 Running OPM Flow 2023-10 From The Command Line*).

Note

When the DATES, TIME and TSTEP keywords are used to advanced the simulation through time this "new" time step is known as a "report time step", as either an action (open a well for example) or a report request will be made. This is different to a simulator's automatic time step used to advance the simulator to the next "reporting time step".

For instance, if the simulation is currently at January 1, 2020 and the next reporting time step is January 1, 2021, then the simulator may not be able to advance directly in one time step to January 1, 2021. Instead, a series of time steps of various duration will be automatically selected by the simulator based on the current set of numerical convergence controls until the required "report time step" is reached.

The next section summarizes the data requirements for the SCHEDULE section and offers some guidance on structuring the format of the keywords in order to increase readability of the input deck and to minimize potential input errors.

12.2 DATA REQUIREMENTS

Apart from the keywords that advance the simulator through time (the DATES, TIME and TSTEP keywords), the minimum required data is associated with defining a well (WELSPECS), defining a well's connection to the reservoir (COMPDAT), and the operating and production targets and constraints for the well (WCONHIST, WCONPROD, WCONINJH or WCONINJE). Well connections can be "grouped" into completions using the COMPLUMP keyword to represent actual physical well completions in the model. Wells can either operate independently or under group control. That is when a well is allocated to a group, then the group can dictate how the wells in the group under group control are operated. For example, a group may have production targets and constraints and all wells under group control within the group will be operated in such a manner as to satisfy the group's targets and constraints. Note that wells can belong to a group but do not necessary have to be under group control. The top level group, level one, has the name FIELD and under this level can be wells, groups and sub groups to the higher level groups. By default three group levels are defined that sets the wells as level three, reporting directly to defined groups at level two, and the level two groups reporting to the FIELD group at level one. If a different configuration is required, then the GRUPTREE keyword should be used to define the group hierarchy by defining a lower level group that reports directly to a higher level group.

The major well specification keywords are summarized in Table 12.1 for ease of reference.

Major Well Specification Keywords

Purpose	Keywords	Description
Vertical Flow Tubing Performance Table Specification	VFPINJ	*VFPINJ – Define Injection Vertical Flow Performance Tables*. VFPINJ declares similar data to VFPPROD but for injection wells.
	VFPPROD	*VFPPROD – Define Production Vertical Flow Performance Tables*. The VFPPROD keyword defines production Vertical Flow Performance ("VFP") tables that are used to determine the outflow or downstream pressure based on the inlet or upstream pressure and the phases flowing through the system. For a well this means the table relates the flowing bottom-hole pressure ("FBHP") to the well's tubing head pressure ("THP") based on the oil, gas and water rates (and any artificial lift quantities like gas lift gas), or phases ratios, flowing up the wellbore.

Major Well Specification Keywords		
Purpose	**Keywords**	**Description**
Well Specification	WELSPECS	*WELSPECS – Define Well Specifications*. The WELSPECS keyword defines the general well specification data for all well types, and must be used for all wells before any other well specification keywords are used in the input file. The keyword declares the name of well, the initial group the well belongs to, the wellhead location and other key parameters.
Well Completion Data	COMPDAT	*COMPDAT – Define Well Connections to the Grid*. COMPDAT defines how a well is connected to the reservoir by defining or modifying existing well connections.
	COMPLUMP	*COMPLUMP – Assign Well Connections to Completions*. The COMPLUMP keyword assigns connections, as defined by the COMPDAT keyword, to completion intervals. This "lumping" or "grouping" of the connections to various completion intervals allows automatic workovers and economic criteria to be applied to the completions (that is a set of connections) as opposed to the individual connections.
	COMPTRAJ	*COMPTRAJ – Define Well Trajectory Connections to the Grid*. COMPTRAJ keyword defines how a well that has been declared as a trajectory well, using the WELTRAJ keyword, is connected to the reservoir model by defining or modifying existing well perforation depths. The keyword can only be used for wells defined by the WELTRAJ keyword, This is an OPM Flow specific keyword and will therefore cause an error in the commercial simulator.
	WELTRAJ	*WELTRAJ – Define Well Trajectory Data*. WELTRAJ defines a trajectory well together with the well trajectory data, and is used in conjunction with the COMPTRAJ keyword to define the well connections to the simulation grid blocks. The keyword can only be used for trajectory wells that employ the COMPTRAJ keyword to define the connections to the grid, This is an OPM Flow specific keyword and will therefore cause an error in the commercial simulator.
Historical Production and Injection Data	WCONHIST	*WCONHIST – Define Well Historical Production Rates and Pressures*. WCONHIST defines production rates and pressures for wells that have been declared history matching wells by the use of this keyword. History matching wells are handled differently than ordinary wells that use the WCONPROD keyword for controlling their production targets and constraints.
	WCONINJH	*WCONINJH – Well Historical Observed Injection Rates and Pressures*. WCONINJH declares similar data as the WCONHIST keyword but for injection wells.
Predictive Production and Injection Data	WCONPROD	*WCONPROD – Define Well Production Targets and Constraints*. The WCONPROD keyword defines production targets and constraints for wells that have previously been defined by the WELSPECS keyword.

Major Well Specification Keywords		
Purpose	**Keywords**	**Description**
	WCONINJE	*WCONINJE – Well Injection Targets and Constraints*. WCONINJE declares similar data as the WCONPROD keyword but for injection wells.
Well Control	WELCNTL	*WELCNTL – Modify Well Control and Targets*. The WELCNTL keyword modifies a well's target control and value, both rates and pressures, for previously defined wells without having to define all the variables on the well control keywords: WCONPROD, WCONHIST, WCONINJE, or WCONINJH keywords. See also the WELTARG keyword below.
	WECON	*WECON – Well Economic Criteria for Production Wells*. WECON defines the economic criteria for production wells that have previously been defined by the WELSPECS and WCONPROD keywords.
	WEFAC	*WEFAC – Define Well Efficiency*. WEFAC defines a well's efficiency or up-time as opposed to setting the efficient factors at the group level.
	WHISTCTL	*WHISTCTL - Define Well Historical Target Phase*. The WHISTCTL keyword changes the target control phase for wells declared as history match wells via the WCONHIST keyword. The target phase is set on the WCONHIST keyword and WHISTCTL overrides this value for all subsequent entries on the WCONHIST keyword
	WELOPEN	*WELOPEN – Define Well and Well Connections Flowing Status*. WELOPEN defines the status of wells and the well connections, and is used to open and shut previously defined wells and well connections without having to re-specify all the data on the well control keywords: WCONPROD, WCONHIST, WCONINJE, or WCONINJH keywords.
	WELTARG	*WELTARG – Modify Well Target and Constraint Values*. The WELTARG keyword modifies the target and constraints values of both rates and pressures for previously defined wells without having to define all the variables on the well control keywords: WCONPROD, WCONHIST, WCONINJE, or WCONINJH keywords. See also the WELCNTL keyword above.
Well Productivity Adjustments (Normally used in History Matching)	WELPI	*WELPI – Define Well Productivity and Injectivity Indices*. WELPI is used to define a well's productivity or injectivity index and the values enter on this keyword for a given well will override any previously calculated values, or values previously entered values using this keyword.
	WPIMULT	*WPIMULT – Define Well Connection Multipliers*. The WPIMULT keyword defines a well connection multiplier factor that scales the existing well connection values. The resulting effect is to scale the well's productivity at the reporting time step the keyword is entered.

Notes:

1) Cells colored orange under the Keywords column are currently not supported by OPM Flow.

Table 12.1: Major Well Specification Keywords

Wells are initially allocated to groups via the WELSPECS keyword and groups have a similar set of keywords as for wells, as outlined in Table 12.2. However only a limited set of keywords have been implemented in OPM Flow compared with the commercial simulator's set of keywords.

Major Group Specification Keywords		
Purpose	**Keywords**	**Description**
Group Specification	GRUPTREE	*GRUPTREE – Define Group Tree Hierarchy*. GRUPTREE defines the group hierarchy of groups that have been created by having wells assigned to them via the WELSPECS keyword. By default three group levels are defined that sets the wells as level three, reporting directly to defined groups at level two, and the level two groups reporting to the FIELD group at level one.
Predictive Production and Injection Data	GCONPROD	*GCONPROD – Group Production Targets and Constraints*. The GCONPROD keyword defines production targets and constraints for groups, including the top most group in the group hierarchy known as the FIELD group.
	GCONINJE	*GCONINJE – Group Injection Targets and Constraints*. GCONINJE declares similar data as for GCONPROD but for group injection targets and constraints.
Group Control	GECON	*GECON – Group Economic Criteria for Production Groups*. The GECON keyword defines economic criteria for production groups, including the field level group FIELD, that have previously been defined by the WELSPECS and GCONPROD keyword.
	GEFAC	*GEFAC – Define Group Efficiency*. GEFAC defines a group's efficiency or up-time factor as opposed to setting the efficiency factors for individual wells.
	GRUPTARG	*GRUPTARG – Modify Group Targets and Constraints Values*. GRUPTARG keyword modifies the production target and constraints values of both rates and pressures for previously defined groups without having to define all the variables on the group production control keywords: GCONPROD or GCONPRI keywords.
Notes: 1) Cells colored orange under the Keywords column are currently not supported by OPM Flow.		

Table 12.2: Major Group Specification Keywords

The final set of keywords define various controls for the model, the keywords available to advance the simulator through time, and the available reporting keywords, as illustrated in Table 12.3.

Schedule Advancement, Control And Reporting Keywords		
Purpose	**Keywords**	**Description**
Control RS and RV Behavior	DRSDT	*DRSDT – Solution Gas (Rs) Maximum Rate of Increase Parameters*. DRSDT defines the maximum rate at which the solution gas-oil ratio (Rs) can be increased in a grid cell. The keyword is similar in functionality to the DRSDTR keyword, that defines the maximum rate at which Rs can be increased in a grid cell by region.

Schedule Advancement, Control And Reporting Keywords		
Purpose	**Keywords**	**Description**
	DRSDTR	*DRSDTR – Solution Gas (Rs) Maximum Rate of Increase Parameters by Region*. DRSDTR defines the maximum rate at which the solution gas-oil ratio (Rs) can be increased in a grid cell for various regions in the model. The keyword is similar in functionality to the DRSDT keyword, that defines the maximum rate at which Rs can be increased in a grid cell for all cells in the model.
	DRVDT	*DRVDT – Solution Oil (Rv) Maximum Rate of Increase Parameters*. DRVDT defines the maximum rate at which the solution oil-gas ratio or condensate-gas ratio (Rv) can be increased in a grid cell. The keyword is similar in functionality to the DRVDTR keyword, that defines the maximum rate at which Rv can be increased in a grid cell by region.
	DRVDTR	*DRVDTR – Solution Oil (Rv) Maximum Rate of Increase Parameters by Region*. DRVDTR defines the maximum rate at which the solution oil-gas ratio or condensate-gas ratio (Rv) can be increased in a grid cell for various regions in the model. The keyword is similar in functionality to the DRVDT keyword, that defines the maximum rate at which Rv can be increased in a grid cell for all cells in the model.
Schedule Advancement	DATES	*DATES – Advance Simulation by Reporting Date*. DATES advances the simulation to a given report date after which additional keywords may be entered to instruct OPM Flow to perform additional functions via the SCHEDULE section keywords, or further DATES keywords may be entered to advance the simulator to the next report date.
	TIME	*TIME – Advance Simulation by Cumulative Reporting Time*. TIME advances the simulation to a given cumulative report time after which additional keywords may be entered to instruct OPM Flow to perform additional functions via the SCHEDULE section keywords, or further TIME keywords may be entered to advance the simulator to the next report time.
	TSTEP	*TSTEP – Advance Simulation by Reporting Time*. TSTEP keyword advances the simulation to a given report time after which additional keywords may be entered to instruct OPM Flow to perform additional functions via the SCHEDULE section keywords, or further TSTEP keywords may be entered to advance the simulator to the next report time
Schedule Advancement Control	TUNING	*TUNING - Numerical Tuning Control*. TUNING defines the parameters used for controlling the commercial simulator's numerical convergence parameters for the global grid. The keyword is mostly ignored by OPM Flow; however, the simulator can be instructed to read the first record of the TUNING keyword if the appropriate command line parameter has been activated (see section *2.2 Running OPM Flow 2023-10 From The Command Line*).
	NEXTSTEP	*NEXTSTEP – Maximum Next Time Step Size* . NEXTSTEP defines the maximum time step size the simulator should take for the next time step. This keyword can be used to reset the time step for when known large changes to the model are taking place that may result in time step chops.

Schedule Advancement, Control And Reporting Keywords		
Purpose	**Keywords**	**Description**
Reporting	RPTSCHED	*RPTSCHED – Define SCHEDULE Section Reporting*. RPTSCHED keyword defines the data in the SCHEDULE section that is to be printed to the output print file in human readable format.
	RPTRST	*RPTRST – Define Data to be Written to the RESTART File*. RPTRST keyword defines the data and frequency of the data to be written to the RESTART file at each requested restart point. In addition to the solution data arrays required to restart a run, the user may request additional data to be written to the restart file for visualization in OPM ResInsight.
Notes: 1) Cells colored orange under the Keywords column are currently not supported by OPM Flow.		

Table 12.3: Schedule Advancement, Control and Reporting Keywords

In terms of structuring the format of the keywords in the SCHEDULE section it is advisable to declared all the VLP tables, wells and groups at the start of SCHEDULE section as oppose to declaring the items as they are needed, or when they come on stream at various times during the simulation. This produces a cleaner input deck and tends to limit unforeseen errors, as all items are declared upfront and only the operational changes (opening wells, changing group and well targets, etc.) are needed. The example SCHEDULE section given on the following pages illustrates a typical SCHEDULE based on this philosophy.

The first segment of the example shows the start of the SCHEDULE section and the group definitions and controls. Here there are controls on the group level only at the FIELD level and there is both production and injection rate targets, as well as water and liquid handling constraints applied on the FIELD level. These keywords are activated from the start of the simulation as set by the START keyword in the RUNSPEC section, in this case January 1, 2020.

```
-- ================================================================================
--
-- SCHEDULE SECTION
--
-- ================================================================================
SCHEDULE
-- --------------------------------------------------------------------------------
-- GROUP PRODUCTION AND INJECTION CONTROLS
-- --------------------------------------------------------------------------------
--
--       DEFINE GROUP TREE HIERARCHY
--
--       LOWER     HIGHER
--       GROUP     GROUP
GRUPTREE
         FLTBLK1  FIELD                                                        /
         FLTBLK2  FIELD                                                        /
         FLTBLK3  FLTBK2                                                       /
/
--
--       GROUP PRODUCTION CONTROLS
--
-- GRUP  CNTL  OIL    WAT    GAS    LIQ    CNTL  GRUP  GUIDE  GUIDE  CNTL
-- NAME  MODE  RATE   RATE   RATE   RATE   OPT   CNTL  RATE   DEF    WAT
GCONPROD
FIELD    GRAT  1*     8E3   125E3   10E3   1*     1*    1*     1*     1*       /
/
```

```
--
--        GROUP INJECTION TARGETS AND CONSTRAINTS
--
-- GRUP   FLUID CNTL    SURF    RESV    REINJ   VOID   GRUP   GUIDE   GUIDE GRUP   GRUP
-- NAME   TYPE  MODE    RATE    RATE    FRAC    FRAC   CNTL   RATE    DEF   REINJ RESV
GCONINJE
FIELD     GAS   RATE   100E3   1*      1*      1.0    YES    1*      1*    1*    1*   /
/
```

The next segment covers the well specification and includes the loading of the VFP tables via include files, declaring the wells using the WELSPECS keyword, and connecting the wells to the reservoir using the COMPDAT keyword. In addition, the COMPLUMP keyword is invoked to assign the well connections to well completions.

In multi-stacked reservoirs it is a good idea to associate the completion number with a given reservoir for all the wells. So for example completion number one is always associated with the Lower Talang Akar Formation Unit A, completion number two with the Lower Talang Akar Formation Unit B, and completion number three with the Upper Talang Akar Formation Unit C, etc. In this way one can easily identify which zone a well is producing from or completed in.

Finally, the WCONPROD keyword is used to define the operating conditions for the gas producers(GP01 and GP02) and to assign the VFP tables to the producing wells. Whereas the WCONINJE keyword performs a similar function for the single gas injector, GI01.

Notice also the well naming nomenclature that easily identifies the type of well: the letter G for a gas well, O for an oil well and W for a water well, and the function of the well, P for a producer and I for an injector.

```
-- ------------------------------------------------------------------------
-- WELL SPECIFICATIONS AND COMPLETIONS
-- ------------------------------------------------------------------------
--
--       LOAD INCLUDE FILES WITH VLP TABLES
--
INCLUDE
         'WEL-P50-VLP01.inc'     / VFP TABLE #1 5 1/2 inch tubing

INCLUDE
         'WEL-P50-VLP02.inc'     / VFP TABLE #2 4 1/2 inch tubing
--
--       WELL SPECIFICATION DATA
--
-- WELL  GROUP     LOCATION  BHP    PHASE  DRAIN  INFLOW  OPEN  CROSS  PVT
-- NAME  NAME        I    J  DEPTH  FLUID  AREA   EQUANS  SHUT  FLOW   TABLE
WELSPECS
GI01     FIELD      14   13   1*     GAS    1*     GPP    SHUT   NO     1*     /
GP01     FLTBLK1    64   80   1*     GAS    1*     GPP    SHUT   NO     1*     /
GP02     FLTBLK1    24  110   1*     GAS    1*     GPP    SHUT   NO     1*     /
/
--
--       WELL CONNECTION DATA
--
-- WELL  --- LOCATION ---  OPEN   SAT   CONN   WELL   KH    SKIN   D     DIR
-- NAME   II  JJ  K1  K2   SHUT   TAB   FACT   DIA    FACT  FACT   FACT  PEN
COMPDAT
GI01      1*  1*   1  60   SHUT   1*    1*    0.708   1*    0.0    1*    'Z'   /
GP01      1*  1*   1  60   SHUT   1*    1*    0.708   1*    0.0    1*    'Z'   /
GP01      1*  1*   1  60   SHUT   1*    1*    0.708   1*    0.0    1*    'Z'   /
/
--
--       ASSIGN WELL CONNECTIONS TO COMPLETIONS
--
-- WELL  --- LOCATION ---  COMPL
-- NAME   II  JJ  K1  K2   NO.
COMPLUMP
GI01      14  13   1  20    1                              / COMPLETION NO. 01
GI01      14  13  35  60    2                              / COMPLETION NO. 02
GP01      64  80   1  20    1                              / COMPLETION NO. 01
GP01      64  80  20  60    2                              / COMPLETION NO. 02
GP02      24 110   1  20    1                              / COMPLETION NO. 01
GP02      24 110  25  60    2                              / COMPLETION NO. 02
/
--
--       WELL PRODUCTION WELL CONTROLS
--
-- WELL  OPEN/  CNTL   OIL    WAT    GAS   LIQ    RES    BHP   THP   VFP    VFP
-- NAME  SHUT   MODE   RATE   RATE   RATE  RATE   RATE   PRES  PRES  TABLE  ALFQ
WCONPROD
GP01     SHUT   GRUP   1*     1*     70E3  1*     1*     500.0 120.0 1         /
GP02     SHUT   GRUP   1*     1*     70E3  1*     1*     500.0 120.0 2         /
/
--
--       WELL INJECTION CONTROLS
--
-- WELL  FLUID  OPEN/  CNTL  SURF   RESV   BHP   THP   VFP
-- NAME  TYPE   SHUT   MODE  RATE   RATE   PRSES PRES  TABLE
WCONINJE
GI01     GAS    SHUT   RATE 125E3   1*     10E3   1*    1*                    /
/
```

The final part of the well specification segment is shown below and sets the well efficiency for the wells via the WEFAC keyword, the producing wells economic limits via the WECON keyword, and then advances the simulation to June 25, 2020. Thus, there is no production up to January 25, 2020 as all the wells are shut-in.

```
--
--       WELL EFFICIENCY FACTORS
--
-- WELL  EFF    NETWK
-- NAME  FACT   OPTN
WEFAC
'GI*'    0.950                                                          /
'GP*'    0.950                                                          /
/
--
--       WELL ECONOMIC CRITERIA FOR PRODUCTION WELLS
-- WELL  MIN    MIN    MAX    MAX    MAX    CNTL    END   WELL
-- NAME  ORAT   GRAT   WCUT   GOR    WGR    MODE    RUN   NAME
WECON
'GP*'    1*     5.0E3  1*     1*     1*    'WELL'  'NO'                  /
/
DATES
 25 JAN   2020           /
/
```

At January 25, 2020 yearly restarts are requested (RPTRST) together with various printed reports (RPTSCHED), as well as the maximum time step size of 31 days being set via the TUNING keyword. More importantly, well GP01 is opened together with completion number one to put the well on production. Note that one needs to open both the well and the completion; hence, the two lines in the WELOPEN keyword.

```
-- ------------------------------------------------------------------------
-- SCHEDULE SECTION FOR PHASE 1 DEVELOPMENT
-- ------------------------------------------------------------------------
--
--       RESTART CONTROL BASIC = 4 (ALL=2, YEARLY=4, MONTHLY=5, TSTEP=6)
--
RPTRST
         BASIC=4                                                        /
--
--       DEFINE SCHEDULE SECTION REPORT OPTION
--
RPTSCHED
         'WELLS=2'    'WELSPECS'    'CPU=2'     'FIP=2'                 /
--
-- DEFAULT TUNING PARAMETERS (SET MAX TSTEP = 31 days)
--
TUNING
1.0      31.0     0.1     0.15     3     0.3     0.1     1.25     0.75  /
                                                                        /
/
--
--       DEFINE WELL AND WELL CONNECTIONS FLOWING STATUS
--
--  WELL WELL   --LOCATION--  COMPLETION
--  NAME STAT     I   J    K  FIRST LAST
WELOPEN
GP01     OPEN                                                           /
GP01     OPEN     0   0    0     1     1                               /
/

DATES
 1  FEB   2020  /
/
```

At February 1, 2020 further SCHEDULE keywords are processed. The first keyword is the RPTSCHED keyword that in this case switches off all reports printed to the *.PRT file, as only annual reports are required for this particular model.

Next the gas producer GP02 is put on production using two statements in the WELOPEN keyword, and the simulator is requested to advance to report time step of March 1, 2020.

```
--
--       DEFINE SCHEDULE SECTION REPORT OPTION
--
RPTSCHED
'NOTHING'                                                                    /
--
--       DEFINE WELL AND WELL CONNECTIONS FLOWING STATUS
--
--  WELL WELL    --LOCATION--  COMPLETION
--  NAME STAT      I    J    K  FIRST LAST
WELOPEN
GP02     OPEN                                                                /
GP02     OPEN      0   0    0     1     1                                    /
/
DATES
 1  MAR   2020  /
/
```

Then finally at March 1, 2020 the gas injector is placed on stream and the simulation advances on a monthly report basis to the end of the year.

```
--
--       DEFINE WELL AND WELL CONNECTIONS FLOWING STATUS
--
--  WELL WELL    --LOCATION--  COMPLETION
--  NAME STAT      I    J    K  FIRST LAST
WELOPEN
GI01     OPEN                                                                /
GI01     OPEN      0   0    0     1     1                                    /
/
DATES
 1  APR   2020  /
 1  MAY   2020  /
 1  JUN   2020  /
 1  JLY   2020  /
 1  AUG   2020  /
 1  SEP   2020  /
 1  OCT   2020  /
 1  NOV   2020  /
 1  DEC   2020  /
/
```

Next a detailed report is produced as of January 1, 2021, and reporting is then switch off and the simulation continues to the end of the run.

```
-- ------------------------------------------------------------------------
-- SCHEDULE SECTION - 2021-01-01
-- ------------------------------------------------------------------------
RPTSCHED
'WELLS=2'    'WELSPECS'    'CPU=2'     'FIP=2'                                /

DATES
 1  JAN   2021  /
/

RPTSCHED
```

```
'NOTHING'                                                                 /

DATES
 1  FEB   2021  /
 1  MAR   2021  /
 1  APR   2021  /
 ...……..……..……...
/
```

12.2.1 HISTORY MATCHING RUNS

History matching is the process of matching actual historical production data versus the model's production performance. During history matching the engineer specifies the actual field measured production and injection rates for a reservoir which has been producing for a period of time. Although other reservoir characteristics such as permeability, layering structure, aquifer strength and individual well performance are specified as normal in the remainder of the data file, they are in only one realization out of possibly thousands of realizations. There is therefore significant uncertainty associated with the data set. Thus in most cases, the simulated past performance of the reservoir, that is the well rates calculated by OPM Flow up to the present day, will not in general correspond to the measured rates specified by the user. History matching then becomes an exercise in identifying those reservoir properties subject to the greatest uncertainty and adjusting them to bring the simulated and measured rates to an acceptable degree of agreement.

Prior to beginning the matching process the production data should be reviewed and quality checked. This step is needed mainly to:

- Allocate production data areally and vertically to the pertinent areas and zones, as well as ensuring that the correct well completions have been implemented into the model's schedule section. Particular care should be taken with multiple completed wells to ensure that the wells produce from the correct zones at the correct times. Wells that are completed across fault planes also need be checked so that the completions are on the right side of the fault plane.
- Correct field production data to separator conditions consistent with model conditions, or vice versa.
- Correct reported pressures for simulation use (e.g., Peaceman correction), or use the weighted areal average well bottom-hole pressures from the simulator to match against actual data.

Note also that not all production volumes have the same level of quality assurance. Product streams that are sold are going have a higher level of quality assurance because they have value, compared with production streams that are disposed off or re-injected, as they have no value. Thus, in an oil field field selling the produced oil but flaring or re-injecting the produced gas, the oil volumes will be back allocated based on "ticketed sales", and thus the field volumes are reasonable certain. However, even here there are uncertainties associated with allocating production volumes back to the individual wells and well completions. As the gas volumes are not sold they have no value and thus have greater uncertainty than the oil volumes, especially when back allocating back to the wells.

History matching is an iterative process, in which steps are repeated a number of times with variations in reservoir characterization (permeability, pore volume, relative permeability modifications, etc.), until a reasonable match is obtained. There are no precise rules for conducting a history match but the methodology is well established. Generally one first matches pressures globally, then overall saturations (and well rates) and repeat until an overall match is achieved, as the process of matching pressures also effects the saturations and vice versa. Which is why this is an iterative process. This methodology is then repeated on a regional level until the process is complete and reasonable matches have been obtained at the field, region and well levels.

Generally, group control, targets and constraints are not utilized in the history matching part of a simulation study. Instead, the WCONHIST keyword is used to enter the well production (oil, gas and water rates), pressure data (BHP and THP when available) and VFP table assignment, on a well by well basis and on discrete time intervals, normally monthly. WCONHIST allows one to set the "target" phase and rate the well will attempt to produce at, if possible, and to produce the other phases that come with the target rate. That is there is no constraints applied to the other phases, unlike when using the WCONPROD keyword. If well productivity has been matched then the target phase rates will be satisfied and the quality of the history match is determined by the accuracy of the other phases, together with the pressure match. Thus, in an oil field the target phase will normally be the oil phase and the oil rate, and the quality of the history would be determined by the produced gas and water volumes plus the pressure match at various levels (field, region, reservoir and well). If water breakthrough has not occurred and the field is producing above the saturation

pressure then there is very little to match, as the gas production in this case is a direct function of the oil rate.

The WCONHIST keyword also has a RESV target phase where the target is set to the in situ reservoir volume rate which is calculated by the simulator using the oil, water, gas and liquid rates declared on the keyword. This is useful in the initial part of the history matching study where one is trying to match the overall pressure behavior, as the option forces the simulator to produce the correct number of reservoir barrels (or m3), although the actual oil, gas and water rates will most likely not match at this stage of the study. Note that it is not necessary to edit the WCONHIST keywords for a run to change the target phase, as the WHISTCNTL keyword can be used do this from the time this keyword is invoked, thus avoiding changing the control mode on all subsequent WCONHIST keywords.

As mentioned above, the rates and volumes of product streams that are sold are more reliable then those that are being re-injected, flared or otherwise disposed of, and therefore in an oil field the preferred target phase is oil and not liquid, which some engineers prefer to use.

The WCONINJH keyword has similar functionality as the WCONHIST keyword, but is used for history matching injection wells instead.

More recent developments in history matching incorporate Assisted History Matching ("AHM") techniques [294] [295] [296] [297] [298] [299] and [300] in which various statistical techniques are applied in specialized software to match the field data by the user supplying the parameters to vary, the range of the parameters values, and the objective function to minimize. This type of software can also generate a range of "reasonable" history match models that can then be used to quantify the uncertainty range in the prediction phase.

[294] *History Matching and Uncertainty Quantification: Multi-objective Particle Swarm Optimisation Approach (SPE 143067), L. Mohamed, M. Christie, V. Demyanov, Vienna, Austria, 23–26 May 2011.*

[295] *Field-Scale Assisted History Matching Using a Systematic, Massively Parallel Ensemble Kalman Smoother Procedure (SPE00182617), Binghuai Lin, Paul I Crumpton, and Ali H. Dogru, Society of Petroleum Engineers (February 2017).*

[296] *Enhancing the Geological Models Consistency in Ensemble-Based History Matching - An Integrated Approach (SPE00186049), A. Perrone, F Pennadoro, A Tiani, E Della Rossa, and J. Saetrom, Society of Petroleum Engineers (May, 2017).*

[297] *Correlation-Based Adaptive Localization for Ensemble-Based History Matching - Applied to the Norne Field Case Study (SPE00191305), Xiaodong Luo, Rolf Lorentzen, Randi Valestrand, and Geir Evensen, , Society of Petroleum Engineers (April, 2018).*

[298] *Rapid Forecast Calibration Using Nonlinear Simulation Regression with Localization (SPE-193845-MS), Jincong He, Wenyue Sun, and Xian-Huan Wen, SPE Reservoir Simulation Conference held in Galveston, Texas, USA, 10–11 April 2019.*

[299] *Multilevel Strategies and Geological Parameterizations for History Matching Complex Reservoir Models (SPE-193895-MS), Yimin Liu and Louis J. Durlofsky, SPE Reservoir Simulation Conference held in Galveston, Texas, USA, 10–11 April 2019.*

[300] *Improved Estimation and Forecast Through Model Error Estimation – Norne Field Example (IPTC-19142-MS), Minjie Lu and Yan Chen, International Petroleum Technology Conference held in Beijing, China, 26 – 28 March 2019.*

12.2.2 PREDICTION RUNS

Prediction runs are used to optimize future production from the field and can take two forms:

- Prediction runs may follow directly from a history match case using the reservoir description estimated during the history matching phase. The history matching and prediction runs are frequently in separate data files, with the prediction cases "restarting" from the final history match run, as the history match restart file contains sufficient data to continue the simulation into the future. Therefore, the prediction data set contains the same reservoir description as the history match but is initialized using the restart file output from the history match case. In general, the base case scenario is the No Further Activity ("NFA") scenario. This is just a continuation of the history match into the future using the well controls in force at the end of the history matching period. This is the simplest way to run the field since the surface facilities remain unchanged and no workovers take place. However, the switch from history matching mode to prediction mode should be treated with care in order to ensure a smooth transition between the two phases. Subsequent cases will then be based on the NFA scenario, and will attempt to optimize recovery by examining the impact of drilling infill wells, modifying facilities constraints, varying the reservoir management strategy, etc.
- Prediction cases may also be based on predicting production from structures that have not been put on production at the time of the study, that is there is no historical production available. In this case, the reservoir description is based on the best available data set and interpretation, although there may be various variations of the models to cover the range of uncertainties in the static earth modeling. Typically, Low, Best and High models are generated to capture the range of uncertainty in the static modeling, although full uncertainty analysis is becoming more common as computational computer power becomes more available. Additional dynamic sensitivities are then run to determine the Low, Best and High models that address both the static and dynamic uncertainties. These models then form the basis of the supplementary predictions using a variety of different production strategies that are used to improve the petroleum recovery and reduce production costs. For each prediction case, an aspect of the surface facilities or well management policy is changed and the case is compared to the base prediction. Most simulators, including OPM Flow, have a wide range of well, group and surface facility modeling features and it is likely that several will be invoked during recovery optimization to model events such as drilling infill wells, modifying facilities constraints, investigating various depletion strategies, etc.

In both scenarios more recent developments within the industry apply a full uncertainty analysis to the models, in which multiple cases are run under various parameter changes to derive a distribution of recoverable volumes, which in turn are used to select the Low, Best and High scenarios. This can result in hundreds of cases being run and is computationally expensive.

As mentioned above, the switch from history matching mode to prediction mode should be treated with care in order to ensure a smooth transition between the two phases. History matching of a well's BHP has always been somewhat problematic, as the field measured data consists of shutting-in the well for a specific period for the BHP to build backup to a well's average drainage pressure (usually assumed to be the reservoir pressure in the surrounding area). History matching time steps are normally based on monthly time steps to coincide with the monthly back allocated field reported production volumes. Thus, to correctly match the measured Shut-in Bottom-Hole Pressure ("SBHP") data in the model, one would have to shut-in the well in the model as per the field, and then open up the well back up at the end of the BHP survey. One would also have to re-adjust the well rates to match the cumulative volumes produced for the time the well was shut-in. These steps are impracticable and engineers originally handled the problem by assigning the wells a high productivity index, so that the difference between the Flowing Bottom-Hole Pressure ("FBHP") and the SBHP was small.

This would lead to a mismatch in rates when switching from history matching mode to prediction mode as shown in Figure 12.1. Thus, the final step in history matching was to calibrate the well rates based on adjusting the Productivity Indices ("PI") of the wells, in order to get a smooth transition between the two modes of operating the wells.

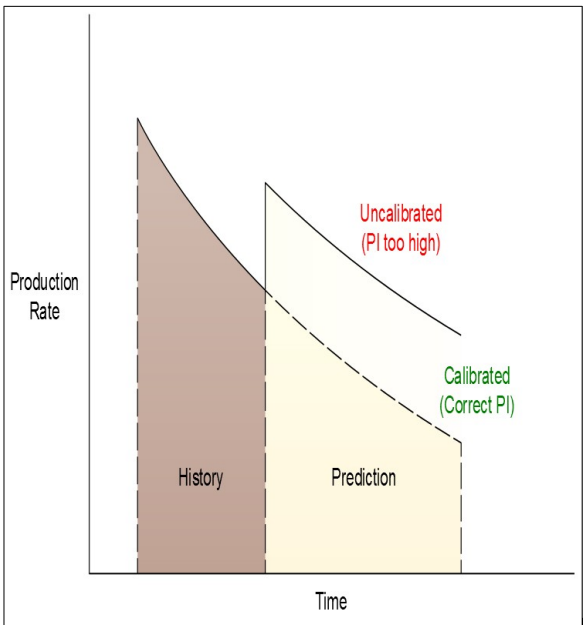


Figure 12.1: Calibration of Well Productivity Index

Calibration is achieved by running the model at the back pressures held against the wells at one or several time-steps. The well PIs and/or wellbore flow parameters are adjusted to duplicate field-observed rates. Although the calibration step was usually done after the history-matching step, there are times when it must be done beforehand (e.g., when modeling multi-layer flow into a well). This is because in stratified reservoirs the correct allocation of rates among layers is dependent on the absolute value of the well's PI (and not solely on the ratios of PIs of the various layers).

Although well PI matching is still performed as part of the general history matching methodology, it is no longer necessary to set a well's PI to a high value in order to mimic the field measured SBHP. This is because most simulators are now capable of reporting an "average" bottom-hole pressure around the well, including one, four, five and nine grid cell averaging, as well as alternative averaging schemes. Thus, the engineer now matches the field measured SBHP with one of the simulator's average grid cell BHPs, normally either the five or nine grid cell average BHP values.

This leaves the question on what controls should be used when moving from history matching mode to prediction mode and ensuring a smooth transition between the two, and at the same time ensuring the resulting predictive forecast is reasonable. There are basically three approaches to this problem:

1) set the well's prediction constraint to the last observed rate,
2) set the well's prediction constraint to the average FBHP and adjusting a well's PI, and thirdly,
3) set the well's prediction constraint to the last FBHP or THP.

The next three sections discusses these options in some detail.

(1) Prediction Constraint Based on a Well's Last Observed Rate

In the history matching mode the wells are controlled by a rate target, so using the last observed well rate would seem a sensible approach. Thus, this approach uses the last observed well rates as well constraints in the prediction run. The typical approach is to constrain production based on a preferred phase (oil rate, gas rate or liquid rate), and then set a reasonable flowing BHP value as a limit for when the well can no longer meet the rate target set. The main requirement for using this approach is therefore that the history-matched model reproduces the rates and flowing bottom-hole pressures fairly exactly.

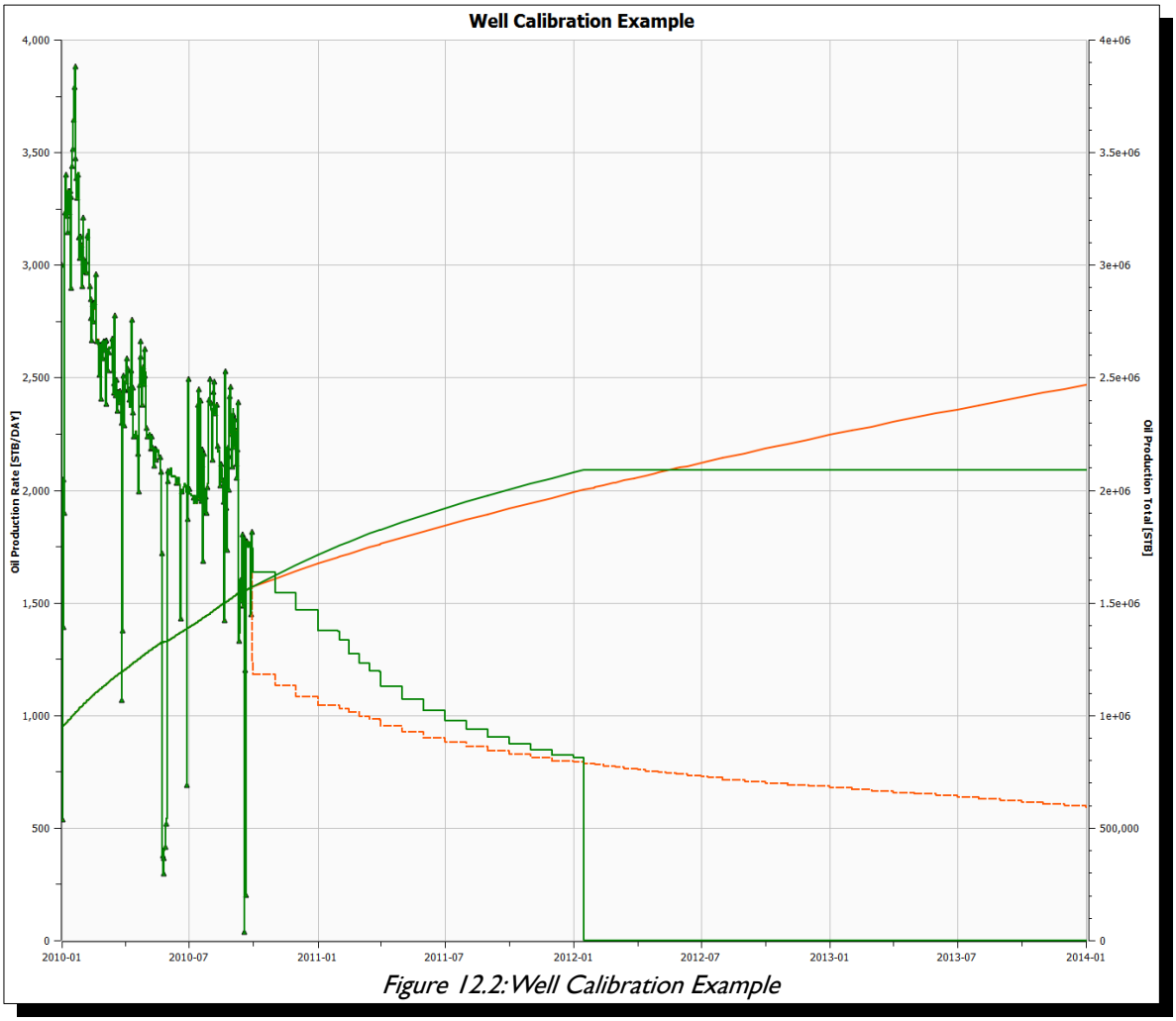


Figure 12.2: Well Calibration Example

However, the are some drawbacks to this approach as one can observe in Figure 12.2, which shows the oil rate and cumulative oil for a well, the symbols on the plot are the actual historical oil production rates. If the last period of production history were not matched successfully, a smooth transition from the history simulation to prediction phase will not be achieved, as shown by the orange lines in Figure 12.2.

A second drawback to this approach, is that the well and field production rate profiles will tend have a flat trend for a few months (or even years) when the well can still deliver the rate target. In real field management such rate profiles can be optimistic if the specified boundary conditions are not representative of actual operating conditions in the field.

(2) Prediction Constraint Based on a Well's Average FBHP and Adjusting a Well's PI

This approach has been adopted by some reservoir engineers as a result of the issues mentioned for the first approach.

Here, an average FBHP of each well, based on most recent production history, is used as a target to control the wells during the prediction phase. Usually, this will lead to abrupt changes in rate and pressure profiles during the transition. The next step is therefore to dampen these abrupt changes in the well rate profiles by applying well PI multipliers (at the well and/or completion level) until a smooth transition is obtained in the rate profiles, similar to that shown by the green lines after the history matching period in Figure 12.1.

The problem with this approach is that it is synonymous to re-calibrating a supposedly history-matched simulation model (with a given permeability distribution based on a certain geological characterization), and thus this approach is rather questionable and is not recommended.

(3) Prediction Constraint Based on a Well's Last Flowing BHP or THP

This methodology is the recommended approach for switching between historical and prediction controls in a reservoir model. It involves automatically applying the last flowing bottom-hole pressure at the instant the transition takes place as the production target, and therefore setting the latest flow rates to be constraints. This ensures a smooth transition between history and prediction phases without having to resort to unreasonable changes to the model. In OPM Flow this is accomplished by using the WELTARG keyword to specify that all the wells should use the bottom-hole pressure as the target at the beginning of the restart run, and defaulting the actual value for the BHP. By defaulting the BHP value the simulator will use the current value of FBHP for a well as the constraint. Note that the WELTARG keyword only defines the variable to be changed, it does not change how a well is controlled. However, by setting the BHP to equal to the current FBHP this has a similar effect as changing the operating target.

Using this approach, all wells will continue flowing at the last flowing bottom-hole pressure from the previous time step, using this value of FBHP as the effective target while the last historical rates are kept as limits (if BHP is requested as shown in the example). This approach achieves two important objectives, firstly it results in a smooth transition from the history-match phase to prediction phase in well and field production profiles And secondly, it gives a smooth decline in well production rate, as is expected of real wells in the field.

Although this discussion has been based on using the BHP as the effective control mechanism, one could also use THP, and to a lesser extent, liquid rate, instead.

The following example below shows how to apply the recommended method to obtain a smooth transition to the predictive phase using the BHP as the controlling mechanism.

Example

In the example, December 2019 is the last month of historical data and therefore the predication phase starts from January 1, 2020.

```
DATES
01 DEC 2019 /
/
--
--       WELL HISTORICAL PRODUCTION CONTROLS
--
-- WELL  OPEN/  CNTL   OIL    WAT    GAS    VFP    VFP   THP   BHP
-- NAME  SHUT   MODE   RATE   RATE   RATE   TABLE  ALFQ  PRES  PRES
WCONHIST
OP01     OPEN   ORAT  15.5E3  100.0  1550   10      1*   900.0 1*     /
OP02     OPEN   ORAT  10.3E3  500.0  1550   10      1*   700.0 1*     /
OP03     OPEN   ORAT   5.4E3 2100.0  1550   10      1*   500.0 1*     /
/
-- ------------------------------------------------------------------------
-- 01 JAN 2020 START OF PREDICTION SCHEDULE SECTION
-- ------------------------------------------------------------------------
DATES
01 JAN 2020 /
/
--
--       WELL PRODUCTION AND INJECTION TARGETS
--
--  WELL WELL   TARGET
--  NAME TARG   VALUE
WELTARG
 OP*     BHP                                              /
/
```

From December 1, 2019 to January 1, 2020 the three oil producing wells will be under history matching control with the oil rate being the target rate for the wells. On January 1, 2020 the wells will be converted to normal producers as the WELTARG keyword changes a well status from a history match well to a regular well. At the same time the WELTARG keyword sets all the wells to have their current FBHP as a constraint, by setting the target BHP values to the default values, which effectively makes the FBHP a target as it is now the "active" constraint.

The alternative keyword for implementing the same approach is the WELCNTL keyword, but unfortunately this is currently not implemented in OPM Flow.

12.2.3 WELL PRODUCTIVITY

Throughout this section the equations are presented in terms of oil production, - or liquid flow, i.e., oil and water. Note that similar equations exists for gas flow in terms of p^2 and pseudo pressure formulations ($\psi(p)$).

Under the assumption of (1) one-dimensional, radial flow into the wellbore, (2) steady-state pressure behavior, and (3) single-phase flow, it can be shown that the flow rate of a well is given by:

$$q = (0.001127)\,(2\pi)\,\frac{kh}{\mu}\left[\frac{p_e - p_w}{\left(\ln\left(\frac{r_e}{r_w}\right) + S\right)}\right] \tag{12.1}$$

Where:

q	= production rate, res.bbl./day
p_e	= pressure at the drainage radius of the well, psia
p_w	= bottom-hole well pressure, psia
r_e	= drainage radius, ft.
r_w	= wellbore radius, ft.
S	= skin factor, dimensionless
kh	= effective permeability thickness, md-ft.

This is extended to multi-phase flow by introduction of relative permeabilities and adjusted to surface conditions by use of the formation volume factor resulting in the following expression for oil production rate:

$$q_o = (0.001127)\,(2\pi)\,\frac{kk_{ro}\,h}{B_o\,\mu_o}\left[\frac{p_e - p_w}{\left(\ln\left(\frac{r_e}{r_w}\right) + S\right)}\right] \tag{12.2}$$

If the modified radial flow equation is written for each perforated interval and summed over all intervals, the resulting equation is the one used to calculate oil production rates in the simulator. One way to specify the flow capacities of wells is to enter r_e, r_w, s and p_w values for the well, with the kh values for each perforated interval. The simulator then determines the other parameter values from calculated pressure and saturation distributions.

Because the pressure at the drainage radius is not readily determined in the simulator, it substitutes p_b (a representative pressure of each perforation grid block) for p. That is readily accommodated by replacing the drainage radius by a probe radius as defined by Peaceman[301]

$$r_b = 0.28\,\frac{\left[\sqrt{\frac{K_y}{K_x}}\,\Delta x^2 + \sqrt{\frac{K_x}{K_y}}\,\Delta y^2\right]^{0.5}}{\left(\frac{K_y}{K_x}\right)^{0.25} + \left(\frac{K_x}{K_y}\right)^{0.25}} \tag{12.3}$$

Where:

K_x and K_y	=	x- and y- direction absolute permeabilities
Δx and Δy	=	grid block sizes (feet)

[301] *Peaceman, D.W. "Interpretation of Well-Block Pressures in Numerical Reservoir Simulation with Nonsquare Grid Blocks and Anisotropic Permeability." SPE 10528 presented at the Sixth SPE Symposium on Reservoir Simulation, New Orleans, (February 1982), pp. 553-569.*

The simulator then uses the modified radial flow equation, written for each connection:

$$q_o = \frac{(0.001127)\,(2\pi)}{\left(\ln\left(\frac{r_b}{r_w}\right) + S\right)}\left(\frac{k\,k_{ro}\,h}{B_o\,\mu_o}\right)\left(p_b - p_w\right) \tag{12.4}$$

The connection term on the COMPDAT keyword in the SCHEDULE section, CONFACT, is simply the transmissibility portion of equation (12.4) for a given connection that is:

$$T_{xy} = \frac{(0.001127)\,(2\pi)(kh)}{\left(\ln\left(\frac{r_b}{r_w}\right) + S\right)} \tag{12.5}$$

And the oil phase mobility term is defined as:

$$M_o = \left(\frac{k_{ro}}{B_o\,\mu_o}\right) \tag{12.6}$$

Substituting (12.5) and (12.6) into (12.4) and summing all the connections for the well to obtain the oil rate for the well gives:

$$q_o = \sum_1^N \left(T_{xy}\,M_o\left(p_b - p_w\right)\right) \tag{12.7}$$

Note in the case of partially completed wells, the effective permeability thickness of a perforated interval may be substantially greater than the average permeability thickness of the formation opposite the perforation.

Engineers commonly use the Productivity Index ("PI") as a parameter to defined an oil well's deliverability, as it is derived relatively easy from field measurements, that is:

$$q_o = \frac{PI}{\left(p_e - p_w\right)} \tag{12.8}$$

or in terms of the productivity index:

$$PI = \frac{q_o}{\left(p_e - p_w\right)} \tag{12.9}$$

It is significant that this equation is written in terms of the pressure at the drainage radius, p_e, rather than grid block pressure, p_b. It must be written this way since PI is a property that is determined by field production tests. One can also relate the above equation to the multilayered extension of the steady-state radial flow equation (12.7), that is

$$PI = \frac{q_o}{\left(p_b - p_w\right)} = \sum_1^N \left(T_{xy}\,M_o\right) \tag{12.10}$$

On very important point to note is that the productivity index is, in general, not constant for a given well since both relative permeability and drainage radius can be time dependent. This makes PI an inappropriate quantity for computer prediction of production rates, but it is often the only type of data available on well performance. To overcome this limitation, most simulators accept PI data as input, and then immediately convert this assuming steady state radial flow with a uniform mobility throughout the drainage area, resulting in the following equation.

$$PI = \sum_{n=1}^{N} \left(T_{xy} \; M_o \left(\frac{\ln\left(\frac{r_e}{r_w}\right) + S}{\ln\left(\frac{r_b}{r_w}\right) + S} \right) \right)_n \tag{12.11}$$

Since the drainage radius, r_e, is generally different from the grid block radius, r_b, the ratio of logarithms may be significant; hence, the use of the PI option normally requires specification of both r_e and r_b.

Equation is (12.11) is used to calculate the PI for the well and is used to print the PI on the WELLS production report requested via the RPTSCHED keyword in the SCHEDULE section. Note that gas wells with non-zero D-factors, the non-Darcy skin factor is added to to the skin (S) in equation (12.11) in both the nominator and the denominator. Secondly, r_e is often undetermined or unknown and it is common to default the DRADIUS parameter on the WELSPECS keyword, in the SCHEDULE section. If the well drainage radius (DRADIUS) is defaulted then equation (12.11) simplifies to:

$$PI = \sum_{n=1}^{N} \left(T_{xy} \; M_o \; \right)_n \tag{12.12}$$

In this case the PI should be considered a grid block productivity index and not a well drainage area productivity index. Note also that if DRADIUS is set to a negative number, then this results in a well's potential being written out to the WELLS production report instead of the productivity index. In this instance this is the rate based on only applying the BHP and THP constraints only, all other constraints are ignored.

12.3 KEYWORD DEFINITIONS

12.3.1 ACTION – DEFINE ACTION CONDITIONS AND COMMAND PROCESSING (FIELD)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The ACTION keyword defines a series of conditions that invoke run time processing of ACTION functions and is similar to executing a run time script for conditions and variables at the field level.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped; use the ACTIONX keyword instead.

12.3.2 ACTIONG – DEFINE ACTION CONDITIONS AND COMMAND PROCESSING (GROUPS)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The ACTIONG keyword defines a series of conditions that invoke run time processing of ACTION functions and is similar to executing a run time script for conditions and variables at the group level

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped; use the ACTIONX keyword instead.

12.3.3 ACTIONR – DEFINE ACTION CONDITIONS AND COMMAND PROCESSING (REGIONS)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The ACTIONR keyword defines a series of conditions that invoke run time processing of ACTION functions and is similar to executing a run time script for conditions and variables at the region level.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped; use the ACTIONX keyword instead.

12.3.4 ACTIONS – DEFINE ACTION CONDITIONS AND COMMAND PROCESSING (WELL SEGMENTS)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The ACTIONS keyword defines a series of conditions that invoke run time processing of ACTION functions and is similar to executing a run time script for conditions and variables associated with well segments.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped; use the ACTIONX keyword instead.

12.3.5 ACTIONW – DEFINE ACTION CONDITIONS AND COMMAND PROCESSING (WELLS)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword starts the definition of an ACTIONW section that stipulates the Boolean conditions to test the nominated well parameters, and the resulting SCHEDULE keywords to be executed, if the Boolean condition evaluates to true. An ACTIONW Definition Section is terminated by an ENDACTIO keyword on a separate single line. Here, the keyword defines a series of conditions applied to wells only, that invoke run time processing of ACTION functions, and is similar to executing a run time script for conditions and variables at the well level. The ACTION series of keywords (ACTION, ACTIONG, ACTIONR, ACTIONS and ACTIONW) can apply Boolean conditional tests to variables at the field, group, region, well segment and well levels.

Note that one should used caution using this facility as it may result in the simulator aborting, because the ACTIONW keyword enables the user to implement complex functionality and therefore it is advisable to start with simple expressions before adding the desired complexity.

See also the PYACTION keyword, also in the SCHEDULE, that implements OPM Flow's Python scripting facility using the Python scripting language.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped; use the ACTIONX keyword instead.

No.	Name	Description	Default
ACTIONW		Defines the start of an ACTIONW Definition Section. This is then followed on a new line by the ACTIONW record that stipulates the conditions for which the defined action will be executed and the various operations to be performed if the condition is satisfied.	
1-1	ACTNAME	ACTNAME is a character sting of up to length eight, that defines the name of this action definition. If ACTNAME has been previously used by any ACTION series keyword, then the previous ACTION series definitions will be replace by the definition declared by this ACTIONW Definition Section.	
1-2	ACTWELL	A character string of up to eight characters in length that defines the well name for which the ACTIONW Definition Section is being defined. Note that the well name (ACTWELL) must have been declared previously using the WELSPECS keyword in the SCHEDULE section, otherwise an error may occur.	
1-3	ACTLHS	ACTLHS is a character string of up to eight characters in length, that stipulates a well SUMMARY variable on the left hand side of a Boolean conditional test. For example, WOPR that stands for the Well Oil Production Rate. In addition, ACTLHS can also be a well User Define Quantity ("UDQ") define by the *UDQ - Declare User Define Quantities ("UDQ")* keyword in the SCHEDULE section. In this case, the first two characters of ACTLHS must be set to WU and the UDQ variable must have previously been declared with the UDQ keyword. A complete list of well SUMMARY vectors that can be used with ACTIONW keyword by OPM Flow and the commercial black-oil simulator is summarized in Table 12.5.	Not Applicable

No.	Name	Description	Default
1-4	ACTTEST	ACTTEST is a defined character string that the states the Boolean operator and must be set to one of the following Boolean conditionals: 1) >: Greater than. 2) <: Less than. 3) >=: Greater than or equal to. 4) <=: Less than or equal to. 5) =: Equals to. 6) !=: Not equal to Note that OPM Flow implementation of ACTTEST has been enhanced, as the commercial simulator only support the following Boolean conditionals: 1) >: Greater than. 2) <: Less than. whereas, OPM Flow fully supports the aforementioned items (1) to (6). Thus, one should be mindful of this fact, if the input deck is to be run by both OPM Flow and the commercial black-oil simulator.	Not Applicable

1-5	ACTRHS	ACTRHS is a numeric value that defines a constant on the right hand side of a Boolean conditional test.
		The set of wells matching the well quantities on the right hand side of the Boolean condition is captured and can be used as a general "well list" with the symbol '?' in subsequent well keywords. For example, to test if the gas production rate is less than 5 MMscf/d for all the wells, one would use:
		`ACTIONW 'GP*' WGPR < 5.0 10000 /`
		`WELOPEN`
		`'?' SHUT /`
		`/`
		`…..`
		`ENDACTIO`
			Not Applicable

1-6	ACTNSTEP	ACTNSTEP is a positive integer that defines the number times that the ACTNAME definition is executed. ACTIONW definitions are activated at the end of a time step and this parameter is used to set how many time steps the ACTNAME definition will be invoked. The default value of one means that the definition will be executed only once. One can use a large value, for example 10,000 for the definition to be executed at every time step. Noted that the counter only affects successful evaluations; i.e. if ACTNSTEP is set equal to one (the default), then the simulator will test the action at the end of every time step until it evaluates to true.	1

No.	Name	Description	Default

1-3	ACTINCR	ACTINCR is a real negative or positive value that stipulates a value to increment ACTRHS every time the Boolean condition evaluates to true.
		For example, if ACTINCR is set to 0.5:
		`ACTIONW 'GP*' WGPR < 5.0 10000 0.5 /`
		`WELOPEN`
		`'?' SHUT /`
		`/`
		`…..`
		`ENDACTIO`
		Then after the third time the Boolean conditional has evaluated to true, the condition would effectively be:
		`ACTIONW 'GP*' WGPR < 6.5 10000 0.5 /`
		`…..`
		`ENDACTIO`
			0.0

1-4	/	Record terminated by a "/"	Not Applicable

		The next section contains any number of standard SCHEDULE keywords that will be executed if the Boolean expression evaluates to true. For example, to reduce the tubing head pressure constraint for when any of the oil producers' oil rate drop below 100 stb/d then one could use:
		`ACTIONW 'OP*' WOPR < 100.0 10000 /`
		`--`
		`-- FLOW WELLS THROUGH LOW PRESSURE SEPARATOR`
		`--`
		`-- WELL WELL TARGET`
		`-- NAME TARG VALUE`
		`WELTARG`
		`'OP*' THP 150 /`
		`/`
		`ENDACTIO`
		In theory, most SCHEDULE keywords can be used in an ACTIONW Definition Section here, except for the time stepping keywords, i.e, TSTEP and DATES.
		See Error: Reference source not found under the ACTIONX keyword definition, for a list of the SCHEDULE keywords that are known to work with the ACTION series of keywords.
			Not Applicable

ENDACTIO		Define the end of ACTIONW Definition Section.	Not Applicable
Notes: 1) There is no terminating "/" for this keyword, instead the ENDACTIO keyword terminates the keyword.			

Table 12.4:ACTIONW Keyword Description

The well SUMMARY vectors that can be used with the ACTIONW keyword by OPM Flow and the commercial black-oil simulator are outlined in Table 12.5.

No.	Well SUMMARY Vector Description	SUMMARY Mnemonic
1	Bottom-Hole Pressure	WBHP
2	Gas Injection Rate	WGIR
3	Gas Injection Total	WGIT
4	Gas-Liquid Ratio	WGLR
5	Gas-Liquid Ratio (Bottom Hole)	WBGLR
6	Gas-Oil Ratio	WGOR
7	Gas Production Rate	WGPR
8	Gas Production Total	WGPT
9	Liquid Production Rate	WLPR
10	Liquid Production Rate	WLPT
11	Oil Injection Rate	WOIR
12	Oil Injection Total	WOIT
13	Oil Production Rate	WOPR
14	Oil Production Total	WOPT
15	Polymer Production Concentration	WCPC
16	Polymer Production Rate	WCPR
17	Salt Production Concentration	WSPC
18	Salt Production Rate	WSPR
19	Tracer Production Concentration (Tracer name should be added to WTPC)	WTPC
20	Tracer Production Rate (Tracer name should be added to WTPC)	WTPR
21	Tubing Head Pressure	WTHP
22	Voidage Injection Rate	WVIR
23	Voidage Injection Total	WVIT
24	Voidage Production Rate	WVPR
25	Voidage Production Total	WVPT
26	Water Cut	WWCT
27	Water Injection Rate	WWIR
28	Water Injection Total	WWIT
29	Water Production Rate	WWPR

No.	Well SUMMARY Vector Description	SUMMARY Mnemonic
30	Water Production Total	WWPT
31	Water-Gas Ratio	WWGR
32	User Define Quantity	WUXXXXXX

Notes:

1) Cells under the No. column not colored show that the SUMMARY vector is functional within an ACTIONW block.
2) Cells under the No. column colored orange show that the SUMMARY vector is currently not supported within an ACTIONW block.

Table 12.5:ACTIONW Supported Well SUMMARY Variables

See also the ACTDIMS and UDADIMS keyword in the RUNSPEC section to define the dimensions for the ACTION series of keywords and associated variables. In addition, the EXIT keyword in the SCHEDULE section that allows for terminating the simulation for when a condition within an ACTIONW definition is satisfied

Although most SCHEDULE section keywords should work with the ACTIONW keyword, Error: Reference source not found under the ACTIONX keyword definition - *ACTIONX – Define Action Conditions and Command Processing*, shows the status of keywords that have been tested and known to work, together with keywords that are currently planned to be implemented.

As mentioned previously, the UDQ keyword stipulates the variables and operations used to access the User Defined Quantities features in OPM Flow. UDQ variables can be constants, SUMMARY variables, as defined in the SUMMARY section, or a formula using various mathematical functions together with constants and SUMMARY variables.

Note

Within an ACTIONW Definition Section any UDQ variables utilizing well variables, must have their associated wells previously fully defined in the commercial simulator, otherwise an error will occur. For example, if a well's GOR is being used as part of a UDQ definition, then the well must be fully characterized prior to declaring the UDQ definition.

This restriction does not apply to OPM Flow; however, it should be considered if the same deck is to be run with both simulators.

User Defined Quantities and also be used as User Defined Arguments ("UDA") in the SCHEDULE section with various group, well, and connection keywords. In this case, the UDA variables are used to replace numerical values on these keywords by UDA variables that have been defined by the UDQ keyword. For example, if we wish to make the oil rate for certain wells to be a function of their water cut, then one can define the function using the UDQ keyword that results in a UDQ variable, WU_WCUT say, and then use WU_WCUT as a UDA variable on the WCONPROD keyword for the ORAT parameter. See Error: Reference source not found for a list of keywords that can be used with UDA variables in the *UDQ - Declare User Define Quantities ("UDQ")* keyword section.

Examples

The first example uses the ACTIONW keyword to re-complete a gas injection well, GI01, when the well's bottom-hole pressure exceeds the fracture pressure of the formation, and to open up a structurally higher zone in the well. The keyword NEXTSTEP is used to set the next step to 0.1 days to avoid convergence issues due to a well event.

```
--
--       ACTIONW WELL COMMANDS
--
ACTIONW
--       ACTION     WELL     ACTION   ACTION  ACTION  MAX      ICREMENT
--       NAME       NAME     LHS      TEST    RHS     ACTIONS  RHS
         ACTW-1     GI01     WBHP      >      6800.0                     /
--
--       ACTION COMMANDS TO BE EXECUTED
--
--       NEXT   ALL
--       STEP   TIME
NEXTSTEP
         0.1    'NO'                                                    /
--
-- WELL PRODUCTION STATUS
--
--  WELL WELL    --LOCATION--  COMPLETION
--  NAME STAT     I   J    K  FIRST LAST
WELOPEN
GI01     SHUT                                                          /
GI01     SHUT     0   0    0     2     2                               /
GI01     OPEN                                                          /
GI01     OPEN     0   0    0     1     1                               /
/
ENDACTIO
```

The second example shows how to reduce a producing well deliverability due to water production. Here, well OP01's productivity index is reduced by 0.98 when the well's liquid production is greater than 100,000 stb of liquid. The reduction is repeated 300 times and each time the action is executed the ACTRHS constant is increased by 10,000 stb.

```
--
--       ACTIONW WELL COMMANDS
--
ACTIONW
--       ACTION     WELL     ACTION   ACTION  ACTION  MAX      ICREMENT
--       NAME       NAME     LHS      TEST    RHS     ACTIONS  RHS
         ACTW-01    OP01     WLPT       >     100E3   300      10E3     /
--
--       DEFINE WELL CONNECTION MULTIPLIERS
--
-- WELL  PI     --LOCATION--  COMPLETION
-- NAME  MULT    I   J    K  FIRST LAST
WPIMULT
OP01     0.980    1*  1*  1*    1*    1*                              /
/

ENDACTIO
```

The final example is similar to the previous example, and shows how to nest ACTIONW blocks and how to apply ACTIONW to all the oil producers when the Boolean condition are satisfied.

```
--
--       ACTIONW WELL COMMANDS
--
ACTIONW
--       ACTION     WELL     ACTION   ACTION  ACTION  MAX      ICREMENT
--       NAME       NAME     LHS      TEST    RHS     ACTIONS  RHS
         ACTW-01A   'OP*'    WLPT       >     100E3   300      10E3     /

--
--       ACTIONW WELL COMMANDS
--
ACTIONW
--       ACTION     WELL     ACTION   ACTION  ACTION  MAX      ICREMENT
--       NAME       NAME     LHS      TEST    RHS     ACTIONS  RHS
         ACTW-01B   'OP*'    WPI       >      5.0     1        0.0      /

--
--       DEFINE WELL CONNECTION MULTIPLIERS
--
-- WELL  PI     --LOCATION--  COMPLETION
-- NAME  MULT     I   J    K  FIRST LAST
WPIMULT
'?'      0.980    1*  1*  1*    1*    1*                              /
/

ENDACTIO
ENDACTIO
```

In this case the outer ACTIONW checks if the liquid production for each OP* well is great than the calculated ACTRHS constant, if it is, then inner ACTIONW reduces all the selected wells' productivity indices by 0.980, provided a well's productivity index is greater than five.

12.3.6 ACTIONX – DEFINE ACTION CONDITIONS AND COMMAND PROCESSING

Description

The ACTIONX keyword defines a series of conditions that invoke run time processing of ACTION functions and is similar to executing a run time script. This is the general purpose version of the ACTION series of keywords that can apply Boolean conditional tests to variables at the field, group, region, well segment and well levels. The ACTION, ACTIONG, ACTIONR, ACTIONS and ACTIONW keywords are not implemented in OPM Flow and are unlikely to be so, as the ACTIONX keyword implements their functionality with greater flexibility.

This keyword starts the definition of a ACTIONX section that stipulates the Boolean conditions to test and the resulting SCHEDULE keywords to be executed if the Boolean condition evaluates to true. An ACTIONX Definition Section is terminated by an ENDACTIO keyword on a separate single line.

Although this keyword is read by OPM Flow and the ACTION and UDQ computational logic and calculations have been implemented, one should used caution using this facility as it may result in OPM Flow aborting. This is because the ACTIONX keyword enables the user to implement complex functionality and therefore it is advisable to start with simple expressions before adding the desired complexity.

See also the PYACTION keyword, also in the SCHEDULE, section that implements OPM Flow's Python scripting facility using the Python scripting language.

No.	Name	Description	Default
ACTIONX		Define the start of ACTIONX Definition Section. This is then followed on a new line by any number of ACTIONX records that define the conditions for which the defined action will be executed and the various operations to be performed if the conditions are satisfied.	
1-1	ACTNAME	ACTNAME is a character sting of up to length eight, that defines the name of this action definition. If ACTNAME has been previously used by any ACTION series keyword, then the previous ACTION series definition will be replace by the definition declared by this ACTIONX Definition Section.	
1-2	ACTNSTEP	ACTNSTEP is a positive integer that defines the number times that the ACTNAME definition is executed. ACTIONX definitions are activated at the end of a time step and this parameter is used to set how many time steps the ACTNAME definition will be invoked. The default value of one means that the definition will be executed only once. Use a large value, for example 10,000 for the definition to be executed at every time step. Noted that the counter only affects successful evaluations; i.e. if ACTNSTEP is set equal to one (the default), then the simulator will test the action at the end of every time step until it evaluates to true.	1
1-3	ACTDELTA	ACTDELTA is a real positive value that stipulates the duration of time that the conditions defined on the second record to be satisfied before the ACTIONX action are executed. For example, if ACTDELTA is defaulted the actions will be executed at the end of the time step for which the conditions are met. If set to say 30, then a minimum of 30 days will pass before the actions are executed (assuming field or metric units).	
		days / days / hours	0.0

No.	Name	Description	Default
1-4	/	Record terminated by a "/"	Not Applicable
2-1	ACTLHS	ACTLHS is a series of character strings, each up to eight characters in length, that defines a constant, UDQ defined value, or a SUMMARY variable on the left hand side of a Boolean conditional test. The format for ACTLHS is dependent on the SUMMARY variable type: Aquifer, Block, Field, Group, Region, Time, Well, Well Connection, Well Local Grid Refinement Connection, or a Well Segment. In addition to SUMMARY variables, an UDQ defined value or a Constant variable can be used. The format for the various data types is given in Table 12.7.	Not Applicable

2-2	ACTTEST	ACTTEST is a defined character string that the states the Boolean operator and must be set to one of the following Boolean conditionals:
		1) >: Greater than.
		2) <: Less than.
		3) >=: Greater than or equal to.
		4) <=: Less than or equal to.
		5) =: Equals to.
		6) !=: Not equal to
		For example to test if the field's gas production rate is less than 600 MMscf/d then one would use:
		`ACTIONX`
		`PHASE2 1 /`
		`GGPR 'FIELD' < 600E3 /`
		`/`
		`….`
		`ENDACTIO`
			Not Applicable

2-3	ACTRHS	ACTRHS is a numeric value or a series of character strings, each up to eight characters in length, that defines a constant, an UDQ defined value, or a SUMMARY variable on the right hand side of a Boolean conditional test, as outlined in Table 12.7 (see also ACTLHS).
		In the case of well quantities on the right hand side the set of matching wells is captured and can be used as a general "well list" with the symbol '?' in subsequent well keywords. For example, to shut-in all oil producing wells ('OP*') with a water cut greater than 90% for every time the field water production rate exceeds 60,000 stb/d one would use:
		`ACTIONX`
		`MXWATER 10000 /`
		`GWPR 'FIELD' > 60E3 AND /`
		`WCUT 'OP*' > 0.90 /`
		`/`
		`-- WELL PRODUCTION STATUS`
		`--`
		`-- WELL WELL --LOCATION-- COMPLETION`
		`-- NAME STAT I J K FIRST LAST`
		`WELOPEN`
		`'?' SHUT /`
		`/`
		`ENDACTIO`
			Not Applicable

No.	Name	Description	Default

2-4	ANDOR	An optional defined character string that stipulates a Boolean operator that must be set to either AND or OR if included on this record, that links this record with additional records of this type. For example, to test if the field's gas production rate is less than 600 MMscf/d after 2020 then one would use:
		`ACTIONX`
		`PHASE2 1 /`
		`GGPR 'FIELD' < 600E3 AND /`
		`YEAR > 2020 /`
		`/`
		`……..`
		`ENDACTIO`
		This item should be left blank if not required.
			Not Applicable

2.5	/	Termination of an ACTIONX Boolean condition record. Note that multiple numbers of records of this type can be entered with each record terminated by a "/", as illustrated above.	Not Applicable
3-1	/	The Boolean condition section of the ACTIONX keyword is terminated by an empty line with a single "/".	Not Applicable

		The next section contains any number of standard SCHEDULE keywords that will be executed if the Boolean expression evaluates to true. For example, to test if the field's gas production rate is less than 600 MMscf/d after 2020 and to open up additional wells if this occurs, then one would use:
		`ACTIONX`
		`PHASE2 1 /`
		`GGPR 'FIELD' < 600E3 AND /`
		`YEAR > 2020 /`
		`/`
		`-- WELL PRODUCTION STATUS`
		`--`
		`-- WELL WELL --LOCATION-- COMPLETION`
		`-- NAME STAT I J K FIRST LAST`
		`WELOPEN`
		`GP10 OPEN /`
		`GP11 OPEN /`
		`/`
		`ENDACTIO`
		In theory, most SCHEDULE keywords can be used in an ACTIONX Definition Section here, except for the time stepping keywords, i.e, TSTEP and DATES. See Table 12.8 for a list of the SCHEDULE keywords that are known to work with the ACTIONX keyword.
			Not Applicable

ENDACTIO		Define the end of ACTIONX Definition Section.	Not Applicable
Notes: 1) There is no terminating "/" for this keyword, instead the ENDACTIO keyword terminates the keyword.			

Table 12.6:ACTIONX Keyword Description

The variable types and the associated definitions that are available for use with Boolean conditionals are outlined in Table 12.7.

Variable Type	Description
AQUIFER	AQUIFER variable consists of two parameters the: 1) Aquifer SUMMARY variable; for example, Analytical Aquifer Influx Rate, AAQR, and the 2) Aquifer number consisting of a positive integer greater than zero that defines the aquifer to be used.
BLOCK	BLOCK variable consists of four parameters: 1) Block SUMMARY variable; for example Block Oil Saturation, BOSAT. 2) Block I location which should be a positive integer greater than or equal to zero and less than or equal to NX that defines the connection location in the I-direction. 3) Block J location which should be a positive integer greater than or equal to zero and less than or equal to NY that defines the connection location in the J-direction. 4) Block K location which should be a positive integer greater than or equal to one and less than or equal to NZ that defines the connection location in the K-direction. The NX, NY, and NZ parameters are defined on the DIMENS keyword in the RUNSPEC section.
CONSTANTS	CONSTANTS can be any numerical value and can also include an integer constant as a counter secondary variable. This type of variable has the following form: 1) Any numerical value. 2) ACTIONX counter as defined by ACTNSTEP in the ACTIONX Definition Section. The default value is zero, but this parameter can be any integer value. For example, if one wanted to activate the action after the third time the Boolean condition was passed then setting ACTNSTEP to one and this parameter to -2 would accomplish this.

FIELD	The FIELD variable consists of any field SUMMARY variable; for example the Field average Pressure, as shown below:
	`ACTIONX`
	`WIPHASE 1 /`
	`FPR < 2500 /`
	`/`
	`…...`
	`ENDACTIO`
	The above would action a set of SCHEDULE keywords if the field average pressure fell below 2,500 psia for a run using FIELD units.

GROUP	GROUP variable definition consists of:
	1) Group SUMMARY variable; for example, Group Oil Production Rate, GOPR.
	2) Group Name which is a character string of up to eight characters in length that defines an existing group, note that the group named FIELD is the top most group.
	To enable an action for when the field's oil production rate drops below 20,000 stb/d then one could use.
	`ACTIONX`
	`OILMIN 1 /`
	`GOPR 'FIELD' < 20.0E3 /`
	`/`
	`…...`
	`ENDACTIO`

Variable Type	Description

REGION	REGION variable definition consists of:
	1) Region SUMMARY variable; selected from one of the following only: RPR, RGSAT, ROSAT, RWSAT, RGIP, ROIP, and RWIP. No other region summary fields are permitted in the expressions.
	2) Fluid In-Place region number which is a positive integer greater than or equal to zero that defines the region number. The value should less than or equal to the NTFIP variable on either REGDIMS or TABDIMS keywords in the RUNSPEC section. Note that a zero value indicates the whole model.
	3) Fluid In-Place region family (not used by OPM Flow).
	For example,
	`ACTIONX`
	`WIPHASE 1 /`
	`RRP 0 < 2500 /`
	`/`
	`….…`
	`ENDACTIO`
	Would action a set of SCHEDULE keywords if the field average pressure fell below 2,500 psia for a run using FIELD units.

TIME	TIME variables consists of one parameter that can have three values: DAY for the current simulation day of the month, MNTH for the current simulation month, and YEAR for the current simulation year.
	Thus, to set an action for April 1, 2025 one would use:
	`ACTIONX`
	`DAY = 1 AND /`
	`MNTH ='APR' AND /`
	`YEAR = 2025 /`
	`/`
	`……..`
	`ENDACTIO`
	Note that the value for the MNTH variable, 'APR', in the example, can also be entered without the quotes, that is:
	`MNTH = APR AND /`
	In addition, numerical values for MNTH, similar to DAY and YEAR, are also permitted and are converted to the nearest integer for comparison, so for example:
	`ACTIONX`
	`DAY = 0.95 AND /`
	`MNTH = 4.40 AND /`
	`YEAR = 2024.9 /`
	`/`
	`……..`
	`ENDACTIO`
	Would again result in the action taking place on April 1, 2025.

Variable Type	Description

WELL	WELL variable definition consists of:
	1) Well SUMMARY variable; for example, Well Oil Production Rate, WOPR.
	2) Well Name which is a character string of up to eight characters in length that defines the well, which must have been declared previously using the WELSPECS keyword in the SCHEDULE section, otherwise an error may occur.
	To reduce the tubing head pressure constraint for when any of the oil producers' oil rate drop below 100 stb/d then one could use.
	`ACTIONX`
	`WOILMIN 1 /`
	`WOPR 'OP*' < 100.0 /`
	`/`
	`--`
	`-- FLOW WELLS THROUGH LOW PRESSURE SEPARATOR`
	`--`
	`-- WELL WELL TARGET`
	`-- NAME TARG VALUE`
	`WELTARG`
	`'OP*' THP 150 /`
	`/`
	`ENDACTIO`

WELL CONNECTION	WELL CONNECTION variable definition is comprised of: 1) Well connection SUMMARY variable; for example, Connection Oil Flow Rate, COFR. 2) Well Name which is a character string of up to eight characters in length that defines the well, which must have been declared previously using the WELSPECS keyword in the SCHEDULE section, otherwise an error may occur. 3) I- Connection: A positive integer greater than or equal to zero and less than or equal to NX that defines the connection location in the I-direction 4) J-Connection: A positive integer greater than or equal to zero and less than or equal to NY that defines the connection location in the J-direction. 5) K- Connection: A positive integer greater than or equal to one and less than or equal to NZ that defines the connection location in the K-direction. The NX, NY, and NZ parameters are defined on the DIMENS keyword in the RUNSPEC section.

Variable Type	Description
WELL LOCAL GRID REFINEMENT CONNECTION	WELL LOCAL GRID REFINEMENT CONNECTION variable definition is comprised of: 1) Well local grid refinement connection SUMMARY variable; for example, the Local Grid Refinement Connection Oil Flow Rate, LCOFR. 2) Well Name which is a character string of up to eight characters in length that defines the well, which must have been declared previously using the WELSPECS keyword in the SCHEDULE section, otherwise an error may occur. 3) Local Grid Refinement Name which is a character string of up to eight characters in length that defines the Local Grid Refinement ("LGR"), which must have been declared previously using the CARFIN or RADFIN keywords in the GRID section, otherwise an error may occur. 4) I- Connection: A positive integer greater than or equal to zero and less than or equal to NX that defines the connection location in the I-direction within the LGR. 5) J-Connection: A positive integer greater than or equal to zero and less than or equal to NY that defines the connection location in the J-direction within the LGR. 6) K- Connection: A positive integer greater than or equal to one and less than or equal to NZ that defines the connection location in the K-direction within the LGR. The NX, NY, and NZ parameters are defined on either the CARFIN or RADFIN keywords in the GRID section depending upon whether a Cartesian or radial local grid refinement is being utilized. **Note Local Grid Refinements are currently not implemented in OPM Flow.**
WELL SEGMENT	WELL SEGMENT variable definition consists of: 1) Well Segment SUMMARY variable; for example, Segment Oil Flow Rate, SOFR. 2) Multi-Segment Well which is a character string of up to eight characters in length that defines the well name which must have been declared previously using the WELSPECS and WELSEGS keywords in the SCHEDULE section, otherwise an error may occur. 3) Segment Number, which is a positive integer greater than or equal to two and less than or equal to MXSEGS on WSEGDIMS keyword in the RUNSPEC section that defines the segment. Note that the total number of wells should be defined via the WELLSDIMS keyword and the number of multi-segment wells should be declared on the WSEGDIMS keyword, both keywords are in the RUNSPEC section.

Table 12.7: ACTIONX Variable Definitions

See also the ACTDIMS and UDADIMS keyword in the RUNSPEC section to define the dimensions for the ACTIONX keyword and associated variables. In addition, the EXIT keyword in the SCHEDULE section that allows for terminating the simulation for when a condition within an ACTIONX definition is satisfied

Although most SCHEDULE keywords should work with the ACTIONX keyword, Table 12.8 shows the status of keywords that have been tested and known to work, together with keywords that are currently planned to be implemented.

ACTIONX Schedule Section Keywords Status					
Action Keywords	Group Keywords	Well Keywords		Connection Keywords	Miscellaneous Keywords
ACTIONX	GCONINJE	WCONINJE	WPIMULT	COMPDAT	BOX
UDQ	GCONPROD	WCONPROD	WSEGVALV	COMPLUMP	ECHO
	GCONSUMP	WECON	WTEST	COMPSEGS	ENDBOX
	GLIFTOPT	WEFAC	WTMULT		EXIT5
	GRUPNET	WELOPEN			INCLUDE[6]
	GRUPTARG	WELPI			MULTX
	GRUPTREE	WELSEGS			MULTX-
	GSATINJE	WELSPECS6			MULTY
	GSATPROD	WELTARG			MULTY-
		WGRUPCON			MULTZ
		WINJMULT			MULTZ-
					NEXT
					NEXTSTEP
					NOECHO

Notes:

1) Cells not colored show that the keyword has been tested and is functional within an ACTIONX block.
2) Cells colored in gray indicate that the keyword has been tested in OPM Flow but the results are currently different to the commercial simulator.
3) Cells colored orange show keywords that are currently available in OPM Flow but have not been tested; however, testing is ongoing.
4) Cells colored red show keywords currently unavailable in OPM Flow because the underlying feature is not available; however, there are plans to implement this functionality.
5) OPM Flow specific keyword.
6) If a standard well is fully declared in an ACTIONX block which is then activated at a later date, and later the well is modified to be a multi-segment well using the WELSEGS and COMPSEGS keywords, then this will cause the simulator to abort with an assert failure.
7) The INCLUDE keyword is permissible in an OPM Flow ACTIONX block, but this will cause an exception in the commercial simulator.

Table 12.8:ACTIONX Schedule Section Keyword Status

As mentioned previously, the UDQ keyword stipulates the variables and operations used to access the User Defined Quantities features in OPM Flow. UDQ variables can be constants, SUMMARY variables, as defined in the SUMMARY section, or a formula using various mathematical functions together with constants and SUMMARY variables.

Note

Within an ACTIONX Definition Section any UDQ variables utilizing group and well variables, must have their associated groups and wells previously fully defined in the commercial simulator, otherwise an error will occur. For example, if a well's GOR is being used as part of a UDQ definition, then the well must be fully characterized prior to declaring the UDQ definition.

This restriction does not apply to OPM Flow; however, it should be considered if the same deck is to be run with both simulators.

User Defined Quantities and also be used as User Defined Arguments ("UDA") in the SCHEDULE section with various group, well, and connection keywords. In this case, the UDA variables are used to replace numerical values on these keywords by UDA variables that have been defined by the UDQ keyword. For example, if we wish to make the oil rate for certain wells to be a function of their water cut, then one can define the function using the UDQ keyword that results in a UDQ variable, WU_WCUT say, and then use WU_WCUT as a UDA variable on the WCONPROD keyword for the ORAT parameter. See Error: Reference source not found for a list of keywords that can be used with UDA variables in the *UDQ - Declare User Define Quantities ("UDQ")* keyword section.

Examples

The first example uses the UDQ keyword to sort the oil wells from high water cut to low, via the WU_WLIST variable, and then use the ACTIONX keyword to shut-in the worst offending well when the field's water production is greater than 30,000 stb/d.

```
--
--      DEFINE START OF USER DEFINED QUANTITY SECTION
--
UDQ
--
--      OPERATOR VARIABLE  EXPRESSION
--
DEFINE  WU_WCUT   1/(WWCT 'OP*')                    / WELL WWCT LIST
DEFINE  WU_LIST   SORT(WU_WCUT)                     / WELL WWCT LIST SORTED
/                                                     END OF UDQ SECTION
--
--      DEFINE START OF ACTIONX SECTION
--
ACTIONX
           WSHUT        10
/
        GWPR  'FIELD' > 30E3  AND                                          /
        WU_LIST 'OP*' > 1                                                  /
/

--
--      DEFINE WELL AND WELL CONNECTIONS FLOWING STATUS
--
--  WELL WELL    --LOCATION--  COMPLETION
--  NAME STAT     I   J    K  FIRST LAST
WELOPEN
'?'      SHUT                                                              /
'?'      SHUT     0   0    0     0     0                                   /
/

ENDACTIO
```

Apart from checking that the field's water production rate is greater than 30,000 stb/d the Boolean conditional also checks that there is more than one well in the sorted well list. Notice also the use of '?'

symbol as a substitution of the well name and that the ACTIONX WSHUTIN series of commands will be executed a total of ten times.

The second example checks to see if the field's gas rates is below 600 MMscf/d and if the simulation time is greater that January 1, 2030. If it is, then compression is installed by re-setting all the gas producing well's THP and BHP pressures to 450 psia and 300 psia respectively. In addition all gas wells currently shut-in are tested to see if they can be opened up under the new THP and BHP constraints.

```
--
--      START ACTIONX FIELD PHASE-3 AUTOMATIC COMPRESSION
--
ACTIONX
       PHASE-3      1                                                    /
        GGPR  'FIELD' < 600E3 AND                                        /
        DAY           = 1     AND                                        /
        MNTH          = JAN   AND                                        /
        YEAR          = 2030                                             /
/

--
--      INSTALL COMPRESSION AND RESET WELL THP AND BHPS
--
--  WELL    WELL   TARGET
--  NAME    TARG   VALUE
WELTARG
'GP*'       THP     450                                                  /
'GP*'       BHP     300                                                  /
/

--
--      TEST AND OPEN ALL WELLS UNDER COMPRESSION CONSTRAINTS
--
--  WELL    TEST   CLOSE   NO.     START
--  NAME    INTV   CHECK   CHECK   TIME
WTEST
'GP*'       1.0    PE      1       3                                     /
/

--
--      END OF ACTIONX FIELD PHASE-3 AUTOMATIC COMPRESSION DEFINITION
--
ENDACTIO
```

12.3.7 AITS – ACTIVATE INTELLIGENT TIME STEPPING

Turns on the commercial simulator's intelligent time stepping.

See *AITS – Activate Intelligent Time Stepping* in the RUNSPEC section.

12.3.8 AITSOFF – DEACTIVATE INTELLIGENT TIME STEPPING

Turns off the commercial simulator's intelligent time stepping.

See *AITSOFF – Deactivate Intelligent Time Stepping* in the RUNSPEC Section

12.3.9 APILIM – DEFINE API TRACKING GRID BLOCK LIMITS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The APILIM keyword defines API Tracking grid block limits for when API Tracking has been activated via the API keyword in the RUNSPEC section. The keyword enables the simulator to monitor the grid blocks outside the limits defined on the keyword, as well as to optionally contrain the values within a given range.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.10 AQUCHGAS – DEFINE CONSTANT PRESSURE GAS ANALYTICAL AQUIFER PROPERTIES

The AQUCHGAS keyword defines the properties of constant pressure gas analytical aquifers.

See *AQUCHGAS – Define Constant Pressure Gas Analytical Aquifer Properties* in the SOLUTION section for a full description.

12.3.11 AQUCHWAT – DEFINE CONSTANT PRESSURE WATER ANALYTICAL AQUIFER PROPERTIES

The AQUCHWAT keyword defines the properties of constant pressure water analytical aquifers.

See *AQUCHWAT – Define Constant Pressure Water Analytical Aquifer Properties* in the SOLUTION section for a full description.

12.3.12 AQUCT – DEFINE CARTER-TRACY ANALYTICAL AQUIFERS

The AQUCT keyword defines Carter-Tracy[302] analytical aquifers, the properties of the aquifer, including the Carter-Tracy aquifer influence function associated with the aquifer, defined by the AQUTAB keyword in the PROPS section.

See *AQUCT – Define Carter-Tracy Analytical Aquifers* in the GRID section.

[302] *Carter, R. D. and Tracy, G. W. "An Improved Method for Calculating Water Influx," Transactions of AIME (1960) 219, 215-417.*

12.3.13 AQUCWFAC – MODIFY CONSTANT PRESSURE WATER ANALYTICAL AQUIFER PROPERTIES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The AQUCWFAC keyword modifies the datum depth and pressure for all aquifers specified by the AQUCHWAT keyword in the SOLUTION or SCHEDULE sections.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.14 AQUFETP – DEFINE FETKOVICH ANALYTICAL AQUIFERS

The AQUFETP keyword defines Fetkovich Analytical aquifers and the aquifer properties. Each row entry in the AQUFETP keyword defines one Fetkovich Analytical aquifer. In order to fully define this type of aquifer, the aquifer must be connected to the reservoir using the AQUANCON keyword in the GRID or SOLUTION sections.

See *AQUFETP – Define Fetkovich Analytical Aquifers* in the SOLUTION section for a full description.

12.3.15 AQUFLUX - DEFINE CONSTANT FLUX ANALYTICAL AQUIFER

The AQUFLUX keyword defines the properties of Constant Flux Analytical Aquifers, that allows for a constant water influx to the connected grid blocks

See *AQUFLUX - Define Constant Flux Analytical Aquifer* in the SOLUTION section for a full description.

12.3.16 BCPROP – DEFINE BOUNDARY CONDITIONS PROPERTIES

Description

The BCPROP keyword defines the type and properties of the boundary conditions.

Together the BCCON and BCPROP keywords define the boundary conditions for the model, and can be used to set boundary conditions for when external influx or efflux volumes are influencing the reservoir pressure and production history. For example, when the average reservoir pressure remains constant throughout the production period due to water influx, or gas migration from an external source.

Note

This is an OPM Flow specific keyword and will therefore cause an error in the commercial simulator.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	INDEX	A positive integer that identifies the boundary condition.			None
2	TYPE	A defined character string that defines the type of boundary condition to be applied, and should be set to one of the following character strings: 1) DIRICHLET: for user defined boundary conditions. In this case, COMPONENT for the fluid type, and PRESS and TEMP for the constant pressure and temperature boundary conditions must be specified. 2) FREE: for the initial state of the boundary to be kept throughout the simulation, that is a constant boundary condition. The remaining items should be defaulted. 3) RATE: for the boundary to have a constant influx or efflux rate. In this case, COMPONENT for the fluid type, and RATE to set the mass rate per unit area must be specified. 4) THERMAL: Constant temperature boundary condition (no-flow of mass). 5) NONE: No flow boundary condition.			None
3	COMPONENT	A defined character string that sets fluid type used in the boundary calculations, and should be set to one of the following character strings: 1) GAS: the gas phase will be used to control the boundary conditions for when the TYPE has been set to DIRICHLET or RATE. 2) OIL: the oil phase will be used to control the boundary conditions for when the TYPE has been set to DIRICHLET or RATE . 3) WATER or WAT: the water phase will be used to control the boundary conditions for when the TYPE has been set to DIRICHLET or RATE. 4) SOLVENT: the solvent component will be used to control the boundary conditions for when the TYPE has been set to RATE. 5) POLYMER: the polymer component will be used to control the boundary conditions for when the TYPE has been set to RATE. 6) NONE: the fluid type is undefined. COMPONENT must be declared and not equal to NONE if TYPE has been set to either DIRICHLET or RATE.			NONE

No.	Name	Description			Default
		Field	Metric	Laboratory	
4	RATE	A real value that defines the constant mass rate per unit area of the specified COMPONENT to be injected or withdrawn at the boundary, when TYPE has been set to RATE. Note a negative value implies an influx rate, whereas, a positive value indicates an efflux.			
		lb/day/ft^2	kg/day/m^2	gm/hour/cm^2	0.0
5	PRESS	PRESS is a real positive value that defines the constant pressure boundary condition. PRESS should only be entered if TYPE has been set to DIRICHLET. If the pressure at the boundary is less than PRESS, then the fluid type declared via PHASE will flow across the boundary. The default value of 1* will use the simulator's calculated value based on data entered via the EQUIL keyword in the SOLUTION section.			
		psia	barsa	atma	1*
6	TEMP	TEMP is a real positive number that defines the constant temperature boundary condition. TEMP should only be entered if TYPE has been set to DIRICHLET or THERMAL. The default value of 1* will use the simulator's calculated value based on data entered via one of the following reservoir temperature keywords: RTEMP, RTEMPA, RTEMPVD, TEMPI, or TEMPVD, in the SOLUTION section. Note that all of the aforementioned reservoir temperature keywords, except for TEMPI, may also be used in the PROPS section as well.			
		°F	°C	°C	1*
7	MECHTYPE	A defined character string that defines the type of geo-mechanical boundary condition to be applied, and should be set to one of the following character strings: 1) FREE: Surface is free to move with no external stress. 2) FIXED: Surface is fixed in the x/y/z-directions if FIXEDX / FIXEDY / FIXEDZ > 0. The displacement in each direction defined by DISPX / DISPY / DISPZ. The external stess is defined by STRESSXX / STRESSYY / STRESSZZ. 3) NONE: No geo-mechanical boundary condition.			NONE
8	FIXEDX	A positive integer that identifies the whether the boundary is free or fixed in the x-direction. A value of 0 implies the boundary is free to move in the x-direction. A value > 0 implies the boundary is fixed in the x-direction with displacement defined by DISPX,			1
9	FIXEDY	A positive integer that identifies the whether the boundary is free or fixed in the y-direction. A value of 0 implies the boundary is free to move in the y-direction. A value > 0 implies the boundary is fixed in the y-direction with displacement defined by DISPY,			1

No.	Name	Description			Default
		Field	Metric	Laboratory	
10	FIXEDZ	A positive integer that identifies the whether the boundary is free or fixed in the z-direction. A value of 0 implies the boundary is free to move in the z-direction. A value > 0 implies the boundary is fixed in the z-direction with displacement defined by DISPZ,			1
	STRESSXX	A real value that defines the diagonal component σ_{xx} of the stress tensor.			
		psia	barsa	atma	0
	STRESSYY	A real value that defines the diagonal component σ_{yy} of the stress tensor.			
		psia	barsa	atma	0
	STRESSZZ	A real value that defines the diagonal component σ_{zz} of the stress tensor.			
		psia	barsa	atma	0
	DISPX	A real value that defines the x-direction displacement boundary condition.			
		feet	m	cm	0
	DISPY	A real value that defines the y-direction displacement boundary condition.			
		feet	m	cm	0
	DISPZ	A real value that defines the z-direction displacement boundary condition.			
		feet	m	cm	0

Notes:

1) The boundary condition connections must have been specified using the BCCON keyword.
2) Cells colored orange indicate parameters for the geo-mechanical boundary conditions that are only supported by the experimental geo-mechanical simulator.
3) Each record must be terminated by a "/" and the keyword is terminated by a "/".

Table 12.9: BCPROP Keyword Description

See also the AQUFLUX keyword that is supported by OPM Flow in both the SOLUTION and SCHEDULE sections, to define a constant flux analytical aquifer.

If the BCCON and BCPROP keywords are not present in the input deck, then the boundary conditions for the model are set to be no flow, which is the normal behavior in both OPM Flow and the commercial simulator.

The BC keyword has been replaced by the BCCON and BCPROP keywords.

Examples

The first example shows how to set a constant pressure boundary using TYPE equal to FREE:

```
--
--       DEFINE BOUNDARY CONDITIONS CONNECTION (OPM FLOW SOLUTION GRID KEYWORD)
--
--INDEX  ---------- BOX ---------    BC
```

```
--       I1  I2   J1  J2   K1  K2   DIRC
BCCON
1        1   1    1   1*   1   1*   X-    /
2        1   1*   1   1    1   1*   Y     /
/

--
--       DEFINE BOUNDARY CONDITIONS PROPERTIES (OPM FLOW SOLUTION GRID KEYWORD)
--
--INDEX  BC         BC     BC    BC     BC
--       TYPE       PHASE  RATE  PRESS  TEMP
BCPROP
1        FREE       1*     1*    1*     1*    /
2        FREE                                 /
/
```

With this option it is only necessary to define the boundary cells and all the other parameters (PHASE, RATE, PRESS, and TEMP) can be defaulted, as they are ignored when TYPE equals FREE.

The next example is based on NX, NY and NZ equal to 20, 1, 10 respectively, on the DIMENS keyword in the RUNSPEC section, and shows how different boundary types can be assigned to different parts of the model.

```
--
--       DEFINE BOUNDARY CONDITIONS CONNECTION (OPM FLOW SOLUTION GRID KEYWORD)
--
--INDEX  ---------- BOX ---------   BC
--       I1  I2   J1  J2   K1  K2   DIRC
BCCON
1        1   1    1   1    1   10   X-    /
2        20  20   1   1    1   10   X     /
/

--
--       DEFINE BOUNDARY CONDITIONS PROPERTIES (OPM FLOW SOLUTION GRID KEYWORD)
--
--INDEX  BC         BC     BC    BC     BC
--       TYPE       PHASE  RATE  PRESS  TEMP
BCPROP
1        RATE       GAS    1*    256.0  100.0 /
2        FREE       4*                       /
/
```

The last example shows how the DIRICHLET boundary condition option may be used:

```
--
--       DEFINE BOUNDARY CONDITIONS CONNECTION (OPM FLOW SOLUTION GRID KEYWORD)
--
--INDEX  ---------- BOX ---------   BC
--       I1  I2   J1  J2   K1  K2   DIRC
BCCON
1        1   1    1   1*   1   1*   X-    /
2        1   1*   1   1    1   1*   Y     /
/

--
--       DEFINE BOUNDARY CONDITIONS PROPERTIES (OPM FLOW SOLUTION GRID KEYWORD)
--
--INDEX  BC         BC     BC    BC     BC
--       TYPE       PHASE  RATE  PRESS  TEMP
BCPROP
1        DIRICHLET  WAT    1*    256.0  100.0 /
2        DIRICHLET  WAT    1*    1*     100.0 /
```

/

Here, the first line sets both the pressure and temperature at the boundary, and the second line defaults the pressure entry, so that the simulator calculated initial boundary pressure will be used.

12.3.17 BOUNDARY – DEFINE A BOUNDARY BOX FOR PRINTING

The BOUNDARY keyword defines a rectangular grid for printing various arrays to the output print file (*.PRT); thus, avoiding printing all the elements in the selected array.

See *BOUNDARY – Define a Boundary Box for Printing* in the GRID section for a full description.

12.3.18 BOX - DEFINE A RANGE OF GRID BLOCKS TO ENTER PROPERTY DATA

BOX defines a range of grid blocks for which subsequent data is assigned for all the cells in the defined BOX. Note that the BOX grid is reset by the keyword ENDBOX by resetting the current defined BOX to be the whole grid. The keyword can be used for any array and for all grid types.

See *BOX - Define a Range of Grid Blocks to Enter Property Data* in the GRID section for a full description.

12.3.19 BRANPROP – DEFINE NETWORK BRANCH PROPERTIES FOR EXTENDED NETWORK OPTION

Description

BRANPROP defines network branch properties for the extended network option for when the Extended Network Model has been activated by the NETWORK keyword in the RUNSPEC section. There are two types of network facilities in the simulator, the Standard Network model, which is defined with the GRUPNET keyword in the SCHEDULE section and the Extended Network Model defined by the BRANPROP and NODEPROP keywords, again in the SCHEDULE section.

For the Extended Network Model the group hierarchy can be different to that defined by the GRUPTREE keyword; however, the bottom most nodes in the network tree associated with wells, must be the same as that defined by the GRUPTREE keyword.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	DOWNNODE	A character string of up to eight characters in length that defines the down stream node name for this branch, that is the node closest to the wells. Thus for a production network, this will be an inlet node as the wells are importing fluid into the branch node. Whereas for an injection node, this is an outlet node as injection fluid is being exported to the wells.			None
2	UPNODE	A character string of up to eight characters in length that defines the up stream node name for this branch, that is the node furthermost from the wells. Thus for a production network, this will be an outlet node as the wells are exporting fluid from the branch node. Whereas for an injection node, this is an inlet node as the wells are importing the injection fluid.			None
3	VFPTAB	A positive integer greater than or equal to zero that defines the vertical lift performance table to be used for calculating the pressure behavior between the inlet (for production) or outlet (for injection) node (DOWNNODE) and the outlet (for production) or inlet (for injection) node (UPNODE). For a production network this must reference a table associated with the VFPPROD keyword, and for an injection network a table declared via the VFPINJ keyword. Both keywords are in the SCHEDULE section. If the pressure behavior between the two nodes is zero (no pressure loss in the network branch), then a value of 9999 should be entered for this variable. A value of zero for VFPTAB removes the branch from the extended network and the resulting flows are ignored in the network flow stream.			None

No.	Name	Description			Default
		Field	Metric	Laboratory	
4	ALQ-NODE	A real positive value that defines the artificial lift quantity to be used in conjunction with the VFPPROD assigned to the branch via the VPFTAB variable. VFPTAB vertical lift performance table and the artificial lift quantity ALQ-NODE are used with the branch fluid rates to calculate the pressure behavior through the branch. For a network this can be considered to be either a pump to pump fluid through the network or a compressor to compress gas to a higher export pressure. Basically, ALQ-NODE is used to reduce the pressure loss through the branch. Note that the units for ALQ-NODE are dependent on the associated variable on the VFPPROD keyword. Should be set to zero if ALQ-DEN is set to either DENO or DENG, or if the branch is associated with an automatic compressor.			0.0
5	ALQ-DEN	A defined character string that defines that ALQ-NODE variable represents either as a surface density or as an artificial lift quantity for a pump or a compressor, and should be set to one of the following: 1) DENO: ALQ-NODE represents the average surface oil density flowing through the branch. 2) DENG: ALQ-NODE represents the average surface gas density flowing through the branch. 3) NONE: ALQ-NODE is the artificial lift quantity as defined on the VFPPROD keyword to model a pump or a compressor. The VFPPROD keyword should be consistent with this variable, that is, if ALQ-DEN is set to to DENO then the surface oil should be used as the ALQ variable on the VFPPROD keyword.			NONE
Notes: 1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".					

Table 12.10: BRANPROP Keyword Description

See also the NETWORK keyword in the RUNSPEC section and the NODEPROP keyword in the SCHEDULE section.

Example

Given the following Extended Network model in Figure 12.3.

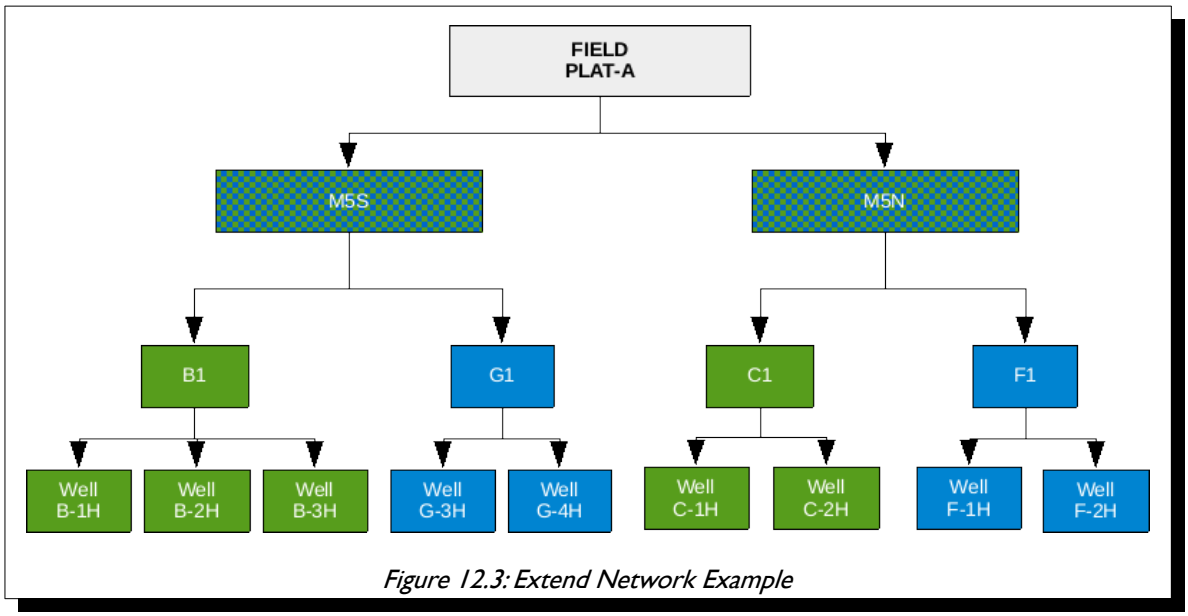


Figure 12.3: Extend Network Example

First the Extended Network model should be used invoked in the RUNSPEC section, and then the BRANPROP keyword should be used to define the branch network, and finally the NODEPROP keyword is used to describe the node properties.

```
-- ==============================================================================
--
-- RUNSPEC SECTION
--
-- ==============================================================================
RUNSPEC
--
--       ACTIVATE THE EXTENDED NETWORK OPTION AND DEFINE PARAMETERS
--
--       MAX.     MAX      NOT
--       NODE     LINK     USED
NETWORK
         3        2        1*                                                  /
…..……..
..……..…..
-- ==============================================================================
--
-- SCHEDULE SECTION
--
-- ==============================================================================
SCHEDULE
--
--       EXTENDED NETWORK BRANCH PROPERTIES
--
-- DOWN  UP       VFP      VFP
-- NODE  NODE     TABLE    ALFQ
BRANPROP
B1       PLAT-A   5        1*                                                  /
C1       PLAT-A   4        1*                                                  /
/
```

```
--
--       EXTENDED NETWORK NODE PROPERTIES
--
-- NODE  NODE   CHOKE  GAS   CHOKE  SOURCE  NETWORK
-- NAME  PRESS  OPTN   LIFT  GROUP  SINK    TYPE
NODEPROP
PLAT-A   21.0   NO     NO                                                  /
B1       1*     NO     NO                                                  /
C1       1*     NO     NO                                                  /
/
```

Here the main platform for the field, PLAT-A, has a fixed 21 barsa pressure applied as an operating constraint.

12.3.20 CALTRAC – DEFINE A GAS CALORFIC VALUE TRACER

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The CALTRAC keyword is used to assign a gas calorfic value to a tracer, for when the Tracer option has been invoked by the TRACER keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.21 CECON – DEFINE WELL CONNECTIONS ECONOMIC LIMIT CRITERIA

Description

CECON sets the economic cut-off criteria for a well's connection to the simulation grid.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.22 CECONT – DEFINE WELL CONNECTIONS TRACER ECONOMIC LIMIT CRITERIA

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, CECONT, sets the tracer economic cut-off criteria for a well's connection to the simulation grid.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.23 COLUMNS – DEFINE INPUT FILE COLUMN MARGINS

The COLUMNS keyword defines the input file column margins; characters outside the margins are ignored by the input parser.

See *COLUMNS – Define Input File Column Margins* in the GLOBAL section for a full description.

12.3.24 COMPDAT – DEFINE WELL CONNECTIONS TO THE GRID

Description

The COMPDAT keyword defines how a well is connected to the reservoir by defining or modifying existing well connections. Ideally the connections should be declared in the correct sequence, starting with the connection nearest the well head and then working along the wellbore towards the bottom or toe of the well, however this may not be possible or convenient, for example when connections are added or removed from a well during the simulation (see COMPORD in the SCHEDULE section for options regarding connection ordering).

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WELNAME	A character string of up to eight characters in length that defines the well name for which the well connection data are being defined. Note that the well name (WELNAME) must have been declared previously using the WELSPECS keyword in the SCHEDULE section, otherwise an error may occur.			None
2	I	A positive integer greater than or equal to zero and less than or equal to NX that defines the connection location in the I-direction. If set to zero or defaulted with 1* the location is taken from the wellhead location I-direction value on the WELSPECS keyword in the SCHEDULE section.			0
3	J	A positive integer greater than or equal to zero and less than or equal to NY that defines the connection location in the J-direction. If set to zero or defaulted with 1* the location is taken from the wellhead location J-direction value on the WELSPECS keyword in the SCHEDULE section.			0
4	K1	A positive integer greater than or equal to one and less than or equal to NZ that defines the UPPER connection location in the K-direction.			None
5	K2	A positive integer greater than or equal to K1 and less than or equal to NZ that defines the LOWER connection location in the K-direction.			None
6	STATUS	A defined character string of length four that defines the connections' operational status, STATUS should be set to one of the following character strings: 1) OPEN: the connections are open to flow. 2) SHUT: the connections are closed to flow (shut-in). 3) AUTO: the connection are initially closed, but may be opened automatically if an economic limit is violated. Currently this option is not supported by OPM Flow.			OPEN

No.	Name	Description			Default
		Field	Metric	Laboratory	
7	SATNUM	An integer greater than or equal to zero and less than NTSFUN as declared on the TABDIMS keyword in the RUNSPEC section, that defines the saturation table number to be used for flow between the reservoir grid block and the well connections. If SATNUM is set to zero or defaulted with 1* then: 1) The saturation table allocated to the grid block that the connections are located within are used. 2) If the hysteresis option has been activated via the HYSTER variable on the SATOPTS keyword in the RUNSPEC section, then both the imbibition and drainage saturation tables allocated to the grid block that the connections are located within are used. The imbibition table allocation can be changed by the COMPIMB keyword in the RUNSPEC section, provided it is entered after the COMPDAT keyword.			0
8	CONFACT	A real value greater than or equal to zero that defines the transmissibility connection factor between the well bore and the reservoir grid block. If set to zero or defaulted with 1* then items (9) through (13) are used to calculate CONFACT.			
		cP.rb/day/psia 0	cP.rm^3/day/bars 0	cP.rcc/hr/atm 0	Defined
9	DW	A real positive value that defines the well bore diameter of the connections for the well. DW is used in calculating a well's productivity or injectivity index; however the value will be ignored in calculating the connections CONFACT value if CONFACT has been directly entered.			
		feet	m	cm	None
10	KH	A real value that defines the effective KH (permeability x length) for the connections. If less than or equal to zero, or defaulted by 1*, then KH is calculated from the connected grid blocks. KH is ignored if CONFAC has been directly entered.			
		mD.ft	mD.m	mD.cm	Calculated from connected grid blocks
11	SKIN	A real value that defines the connections dimensionless skin factor. SKIN is used in calculating a well's productivity or injectivity index; however, the value will be ignored in calculating the connections CONFACT value if CONFAC has been directly entered.			
		dimensionless	dimensionless	dimensionless	0.0
12	DFACT	A real value that defines the non-Darcy D factor coefficient for gas wells. This value should be defaulted with 1* and the non-Darcy D factor coefficient for gas wells defined via the WDFAC keyword in the SCHEDULE section. Currently this option is not supported by OPM Flow.			
		day/Mscf	day/m^3	hour/sc	1*

No.	Name	Description			Default
		Field	Metric	Laboratory	
13	DIRECT	A one letter character string that defines the orientation of the connections and should be set to either X, Y, or Z. The direction of connections also determines the length of the connection used to calculate the connection factor if CONFACT has not been entered directly. The default value is for a vertical connection, that is DIRECT is defaulted to Z.			Z

Notes:

1) The keyword is followed by up to MXCONS records as declared on the WELLDIMS keyword in the RUNSPEC section.
2) Each record is terminated by a "/" and the keyword should be terminated by a "/".

Table 12.11: COMPDAT Keyword Description

Multiple grid block connections can be defined on one record for vertical wells by assigning different values to K1 and K2, for deviated and horizontal wells this may not be possible and therefore each grid block connection must be separately defined by using one record per connection, setting K1 equal to K2 in each record.
See also the WELSPECS keyword to define wells, the COMPIMB to reset the imbibition relative permeability table allocation, and the COMPORD to re-order the completions along the well trajectory. In addition, the COMPLUMP keyword groups well connections together to form well completions for a well. All the aforementioned keywords are described in the SCHEDULE section.

Note

The term well connection is used to describe individual connections from the wellbore to the reservoir grid, as opposed to well completions. A well completion is used to describe a set of connections, for example, a well may consist of several completions with each completion consisting of multiple connections.

Example

The following example defines two vertical oil wells using the WELSPECS keyword and their associated connection data.

```
--
--       WELL SPECIFICATION DATA
--
-- WELL  GROUP     LOCATION  BHP    PHASE  DRAIN  INFLOW  OPEN  CROSS  PRESS
-- NAME  NAME        I    J  DEPTH  FLUID  AREA   EQUANS  SHUT  FLOW   TABLE
WELSPECS
OP01     PLATFORM   14   13   1*     OIL    1*     STD    SHUT   NO     1*  /
OP02     PLATFORM   35   96   1*     OIL    1*     STD    SHUT   NO     1*  /
/
--
--       WELL CONNECTION DATA
--
-- WELL  --- LOCATION ---  OPEN   SAT   CONN   WELL   KH    SKIN   D     DIR
-- NAME   II  JJ  K1  K2   SHUT   TAB   FACT   DIA    FACT  FACT   FACT  PEN
COMPDAT
OP01      1*  1*  20  56   OPEN   1*    1*    0.708   1*    0.0    1*    'Z' /
OP01      1*  1*  75 100   SHUT   1*    1*    0.708   1*    0.0    1*    'Z' /
OP02      35  96  75 100   OPEN   1*    1*    0.708   1*    0.0    1*    'Z' /
```

Well OP01 has two sets of connections; the first one connects grid cells (14, 13, 20) to (14, 13, 56) to the well and is open to flow and the second connecting grid cells (14, 13, 75) to (14, 13, 100) is shut. Well OP02 has only one open connection from cells (35, 96, 75) to cells (35, 96, 100).

12.3.25 COMPDATL – DEFINE WELL CONNECTIONS TO A LGR GRID

Description

The COMPDATL keyword defines how a well in a Local Grid Refinement ("LGR") is connected to the reservoir by declaring the LGR and defining or modifying existing well connections. Ideally the connections should be declared in the correct sequence, starting with the connection nearest the well head and then working along the wellbore towards the bottom or toe of the well, however this may not be possible or convenient, for example when connections are added or removed from a well during the simulation (see the COMPORD keyword in the SCHEDULE section for options regarding connection ordering).

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WELNAME	A character string of up to eight characters in length that defines the well name for which the well connection data are being defined. Note that the well name (WELNAME) must have been declared previously using the WELSPECS keyword in the SCHEDULE section, otherwise an error may occur.			None
2	LGRNAME	A character string of up to eight characters in length that defines the LGR name for which the well LGR connection data are being defined. Note that the well name (LGRNAME) must have been declared previously using the WELSPECL keyword in the SCHEDULE section, otherwise an error may occur. If defaulted with 1* the LGR on the WELSPECL keyword will be utilized.			Defined
3	I	A positive integer greater than or equal to zero and less than or equal to NX that defines the connection location in the I-direction. If set to zero or defaulted with 1* the location is taken from the wellhead location I-direction value on the WELSPECS keyword in the SCHEDULE section.			0
4	J	A positive integer greater than or equal to zero and less than or equal to NY that defines the connection location in the J-direction. If set to zero or defaulted with 1* the location is taken from the wellhead location J-direction value on the WELSPECS keyword in the SCHEDULE section.			0
5	K1	A positive integer greater than or equal to one and less than or equal to NZ that defines the UPPER connection location in the K-direction.			None
6	K2	A positive integer greater than or equal to K1 and less than or equal to NZ that defines the LOWER connection location in the K-direction.			None

No.	Name	Description			Default
		Field	Metric	Laboratory	
7	STATUS	A character string of length four that defines the connections' operational status, STATUS should be set to one of the following character strings: 1) OPEN: the connections are open to flow. 2) SHUT: the connections are closed to flow (shut-in). 3) AUTO: the connection are initially closed, but may be opened automatically if an economic limit is violated. Currently this option is not supported by OPM Flow			OPEN
8	SATNUM	An integer greater than or equal to zero and less than NTSFUN as declared on the TABDIMS keyword in the RUNSPEC, that defines the saturation table number to be used for flow between the reservoir grid block and the well connections. If SATNUM is set to zero or defaulted with 1* then: 1) The saturation table allocated to the grid block that the connections are located within are used. 2) If the hysteresis option has been activated via the HYSTER variable on the SATOPTS keyword in the RUNSPEC section, then both the imbibition and drainage saturation tables allocated to the grid block that the connections are located within are used. The imbibition table allocation can be changed by the COMPIMB keyword in the RUNSPEC section, provided it is entered after the COMPDAT keyword.			0
9	CONFACT	A real value greater than or equal to zero that defines the transmissibility connection factor between the well bore and the reservoir grid block. If set to zero or defaulted with 1* then items (9) through (13) are used to calculate CONFACT.			
		cP.rb/day/psia 0	cP.rm^3/day/bars 0	cP.rcc/hr/atm 0	Defined
10	DW	A real positive value that defines the well bore diameter of the connections for the well. DW is used in calculating a well's productivity or injectivity index; however the value will be ignored in calculating the connections CONFACT value if CONFAC has been directly entered.			
		feet	m	cm	None
11	KH	A real value that defines the effective KH (permeability x length) for the connections. If less than or equal to zero or defaulted by 1* then KH is calculated from the connected grid blocks. KH is ignored if CONFAC has been directly entered.			
		mD.ft	mD.m	mD.cm	Calculated from connected grid blocks
12	SKIN	A real value that defines the connections dimensionless skin factor. SKIN is used in calculating a well's productivity or injectivity index; however, the value will be ignored in calculating the connections CONFACT value if CONFAC has been directly entered.			0.0

No.	Name	Description			Default
		Field	Metric	Laboratory	
		dimensionless	dimensionless	dimensionless	
13	DFACT	A real value that defines the non-Darcy D factor coefficient for gas wells. This value should be defaulted with 1* and the non-Darcy D factor coefficient for gas wells defined via the WDFAC keyword in the SCHEDULE section. Currently this option is not supported by OPM Flow.			
		day/Mscf	day/m^3	hour/sc	1*
14	DIRECT	A one letter character string that defines the orientation of the connections and should be set to either X, Y, or Z. The direction of connections also determines the length of the connection used to calculate the connection factor if CONFAC has not been entered directly. The default value is for a vertical connection, that is DIRECT is defaulted to Z.			Z

Notes:

1) The keyword is followed by up to MXCONS records as declared on the WELLDIMS keyword in the RUNSPEC section.
2) Each record is terminated by a "/" and the keyword should be terminated by a "/".

Table 12.12: COMPDATL Keyword Description

Multiple grid block connections can be defined on one record for vertical wells by assigning different values to K1 and K2, for deviated and horizontal wells this may not be possible and therefore each grid block connection must be separately defined by using one record per connection, setting K1 equal to K2 in each record.
See also the WELSPECS keyword to define wells, the COMPIMB to reset the imbibition relative permeability table allocation, and the COMPORD to re-order the completions along the well trajectory. In addition, the COMPLUMP keyword groups well connections together to form well completions for a well. All the aforementioned keywords are described in the SCHEDULE section.

Note

The term well connection is used to describe individual connections from the wellbore to the reservoir grid, as opposed to well completions. A well completion is used to describe a set of connections, for example, a well may consist of several completions with each completion consisting of multiple connections.

Example

The following example defines two vertical oil wells using the WELSPECS keyword and their associated connection data.

```
--
--       WELL LGR SPECIFICATION DATA
--
--       WELL GROUP  LGR    -LOCATION- BHP    PHASE DRAIN INFLOW SHUT CROSS PVT
--       NAME NAME   NAME     I    J   DEPTH  FLUID AREA  EQUA.  IN   FLOW  TABLE
WELSPECL
         OP01  PLAT OP01LGR  14   13   1*     OIL   1*    STD    SHUT  NO   1*  /
         OP02  PLAT OP02LGR  28   96   1*     OIL   1*    STD    SHUT  NO   1*  /
/
--
--       WELL LGR CONNECTION DATA
--
--       WELL  LGR   ---LOCATION---   OPEN   SAT CONN  WELL KH   SKIN  D    DIR
--       NAME  NAME   II  JJ  K1  K2  SHUT   TAB FACT  DIA  FACT FACT  FACT PEN
COMPDATL
         OP01 OP01LGR 1*  1*  20  56  OPEN   1*  1*  0.708   1*   1*   1*   Z /
         OP01 OP01LGR 1*  1*  75 100  SHUT   1*  1*  0.708   1*   1*   1*   Z /
        OP02 OP02LGR 35  96  75 100  OPEN   1*  1*  0.708   1*   1*   1*   Z /
/
```

Well OP01 has two sets of connections; the first one connects grid cells (14, 13, 20) to (14, 13, 56) to the well and is open to flow and the second connecting grid cells (14, 13, 75) to (14, 13, 100) is shut. Well OP02 has only one open connection from cells (35, 96, 75) to cells (35, 96, 100).

12.3.26 COMPDATM – DEFINE WELL CONNECTIONS TO AN AMALGAMATED LGR GRID

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The COMPDATM keyword is an alias for the COMPDATL keyword. COMPDATM defines how a well in an amalgamated Local Grid Refinement ("LGR") is connected to the reservoir by declaring the LGR and defining or modifying existing well connections. Ideally the connections should be declared in the correct sequence, starting with the connection nearest the well head and then working along the wellbore towards the bottom or toe of the well, however this may not be possible or convenient, for example when connections are added or removed from a well during the simulation (see the COMPORD keyword in the SCHEDULE section for options regarding connection ordering).

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.27 COMPFLSH – ASSIGN COMPDAT DIFFERENTIAL-FLASH LIBERATION RATIOS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

COMPFLSH is used to assign saturated PVT differential-flash liberation ratios to individual well completions for both the oil formation volume factor and the gas-oil ratio. Only the saturated oil properties are changed by this keyword, that is only data entered via the PVCO or PVTO keywords in the PROPS section will be used in the calculation. The other fluid PVT property keywords: PVTW, PVTG, DENSITY, etc., are not used.

The differential liberation expansion ("DLE") experiment is designed to approximate the depletion process of an oil reservoir, and thereby provide suitable PVT data for calculating reservoir performance. This is then coupled with surface faculties via the multi-stage separator flash data. The multi-stage separator test is performed on an oil sample primarily to provide a basis for converting differential liberation data from a residual oil to a stock-tank oil basis. Occasionally, several separator tests are run to help choose separator conditions that maximize stock-tank oil production. Usually two or three stages of separation are used, with the last stage being at atmospheric pressure and near-ambient temperature (60 to 80 °F).

Note, normally the data on the PVCO and PVTO keywords should have already taken the two datasets into account and therefore this keyword should not normally be used in practice, unless there is a specific requirement to adjust the oil PVT data on a well connection basis.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.28 COMPIMB – ASSIGN IMBIBITION SATURATION TABLES TO WELL CONNECTIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The COMPIMB keyword assigns imbibition saturation tables to well connections. The COMPDAT keyword in the SCHEDULE section also assigns imbibition saturation tables to connections, but in this case the table number is the same as for the drainage curve. If this is not the required assignment then the COMPIMB keyword can be used to reset the imbibition saturation table number. For this to be effective the COMPIMB keyword must precede the COMPDAT keyword, otherwise it will have no effect.

The COMPIMB keyword should only be used if the hysteresis option has been activated via the HYSTER variable on the SATOPTS keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WELNAME	A character string of up to eight characters in length that defines the well name for which the well connection data is being defined. Note that the well name (WELNAME) must have been declared previously using the WELSPECS keyword in the SCHEDULE section, otherwise an error may occur.			None
2	I	A positive integer greater than or equal to zero and less than or equal to NX that defines the connection location in the I-direction. If set to zero or defaulted with 1* then all connections in the I-direction that also satisfy J, K1 and K2 criteria are assigned the IMBNUM imbibition table number.			0
3	J	A positive integer greater than or equal to zero and less than or equal to NY that defines the connection location in the J-direction. If set to zero or defaulted with 1* then all connections in the J-direction that also satisfy I, K1 and K2 criteria are assigned the IMBNUM imbibition table number.			0
4	K1	A positive integer greater than or equal to one and less than or equal to NZ that defines the UPPER connection location in the K-direction. If set to zero or defaulted with 1* then the upper most connection in the well is used.			0
5	K2	A positive integer greater than or equal to K1 and less than or equal to NZ that defines the LOWER connection location in the K-direction. If set to zero or defaulted with 1* then the lowest most connection in the well is used.			0

No.	Name	Description			Default
		Field	Metric	Laboratory	
6	IMBNUM	An integer greater than or equal to zero and less than NTSFUN as declared on the TABDIMS keyword in the RUNSPEC, that defines the imbibition saturation table number to be used for flow between the reservoir grid block and the well connections. If IMBNUM is set to zero or defaulted with 1* then the inhibition saturation table allocated to the grid block that the connections are located within is used. If I, J, K1, K2 are all set to zero or defaulted to 1*, then IMBNUM is allocated to all connections in the well.			0

Notes:

1) The keyword is followed by up to MXCONS records as declared on the WELLDIMS keyword in the RUNSPEC section.

2) Each record is terminated by a "/" and the keyword should be terminated by a "/".

Table 12.13: COMPIMB Keyword Description

Multiple grid block connections can be defined on one record for vertical wells by assigning different values to K1 and K2, for deviated and horizontal wells this may not be possible and therefore each grid block connection must be separately defines by setting K1 equal to K2.

See also the COMPDAT keyword in the SCHEDULE section.

Example

The following example defines the connections for two vertical oil wells using the COMPDAT keyword and then re-sets the imbibition saturation functions using the COMPIMP keyword.

```
--
--        WELL CONNECTION DATA
--
-- WELL   --- LOCATION ---   OPEN    SAT    CONN    WELL    KH     SKIN    D      DIR
-- NAME    II   JJ   K1   K2   SHUT    TAB    FACT    DIA     FACT   FACT    FACT   PEN
COMPDAT
OP01       1*   1*   20   56   OPEN    1      1*     0.708    1*     0.0     1*     'Z' /
OP01       1*   1*   75 100    SHUT    2      1*     0.708    1*     0.0     1*     'Z' /
OP02       35   96   75 100    OPEN    1      1*     0.708    1*     0.0     1*     'Z' /
--
-- ASSIGN IMBIBITION SATURATION TABLES TO CONNECTIONS
--
-- WELL      ---LOCATION---    SAT
-- NAME      II   JJ   K1   K2   TAB
COMPIMB
OP01         1*   1*   20   56   11                    /
OP01         1*   1*   75 100    12                    /
OP02         1*   1*   1*   1*   11                    /
/
```

Well OP01 has two sets of COMPIMB records to overwrite the imbibition saturation tables, one for connections (14, 13, 20) to (14, 13, 56) resetting the imbibition saturation table number from one to 11 and one for connections (14, 13, 75) to (14, 13, 100) that resets the imbibition table number from 2 to 12. Well OP02 has only one connection from cells (35, 96, 75) to cells (35, 96, 100), so all the default values for I, J, K1, and K2 can be used to set the imbibition table numbers from 2 to 11. Note in all cases the drainage saturation table retains the value as specified by the COMPDAT keyword, that is one, two and one.

12.3.29 COMPINJK – ASSIGN INJECTION WELL RELATIVE PERMEABILITY VALUES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The COMPINJK keyword assigns injection well relative permeability values to well connections.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.30 COMPLMPL – ASSIGN WELL LGR CONNECTIONS TO COMPLETIONS

Description

The COMPLMPL keyword assigns well connections in a LGR, as defined by the COMPDATL keyword in the SCHEDULE section, to completion intervals. This "lumping" of the connections to various completion intervals allows automatic workovers and economic criteria to be applied to the completions (that is a set of connections) as opposed to the connections. This allows for a more realistic approach for workovers operations.

For example, if the water cut criteria for working over a well was set to 95%, and the average grid block connection thickness was one meter, then once a well's water cut reached 95% the worst offending one meter connection would be shut-in. If the well's actual perforation interval was 10 meters and the 10 connections were lumped as one completion, then when the water cut limit of 95% is reach, the completion would be shut-in, that is all of the 10 connections within the completion would be shut-in.

As the keyword is used to lump connections into a completions, the simulator adds together the contribution from all connections in the completion and uses the total values to test the economic limits. Note that a connection can only belong to one completion. In addition, completions can be used instead of connections in the WELOPEN and WPIMULT keywords if the completions have been defined by COMPLUMP for a well.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WELNAME	A character string of up to eight characters in length that defines the well name for which the well connection data are being defined. Note that the well name (WELNAME) must have been declared previously using the WELSPECS keyword in the SCHEDULE section, otherwise an error may occur.			None
2	LGRNAME	A character string of up to eight characters in length that defines the LGR name for which the well LGR connection data are being defined. Note that the well name (LGRNAME) must have been declared previously using the WELSPECL keyword in the SCHEDULE section, otherwise an error may occur. If defaulted with 1* the LGR on the WELSPECL keyword will be utilized.			Defined
3	I	A positive integer greater than or equal to zero and less than or equal to NX that defines the connection location in the I-direction. If set to zero or defaulted with 1* then all connections in the I-direction that also satisfy J, K1 and K2 criteria are assigned the ICOMP completion number.			0
4	J	A positive integer greater than or equal to zero and less than or equal to NY that defines the connection location in the J-direction. If set to zero or defaulted with 1* then all connections in the J-direction that also satisfy I, K1 and K2 criteria are assigned the ICOMP completion number.			0

No.	Name	Description			Default
		Field	Metric	Laboratory	
5	K1	A positive integer greater than or equal to one and less than or equal to NZ that defines the UPPER connection location in the K-direction. If set to zero or defaulted with 1* then the upper most connection in the well is used.			0
6	K2	A positive integer greater than or equal to K1 and less than or equal to NZ that defines the LOWER connection location in the K-direction. If set to zero or defaulted with 1* then the low most connection in the well is used.			0
7	ICOMP	An integer greater than or equal to one and less than or equal to MXCONS as defined on the WELLDIMS keyword in the RUNSPEC section, that defines the completion number of the currently defined set of connections. If I, J, K1, K2 are all set to zero or defaulted to 1*, then all connections in the well have the same completion number of ICOMP.			None

Notes:

1) The keyword is followed by any number of records.
2) Each record is terminated by a "/" and the keyword should be terminated by a "/".

Table 12.14: COMPLUPL Keyword Description

Multiple grid block connections can be defined on one record for vertical wells by assigning different values to K1 and K2, for deviated and horizontal wells this may not be possible and therefore each grid block connection must be separately defined by setting K1 equal to K2.

See also the COMPDATL keyword in the SCHEDULE section.

Example

The following example defines the connections for two vertical oil wells using the COMPDATL keyword and the re-allocation of the connections to completions intervals using the COMPLMPL keyword.

```
--
--       WELL CONNECTION DATA FOR LGR WELLS
--
--       WELL    LGR      --- LOCATION ---  OPEN   SAT   CONN   WELL   D     DIR
--       NAME    NAME      II  JJ  K1  K2   SHUT   TAB   FACT   DIA    FACT  PEN
COMPDATL
         OP01    OP01LGR    14  13  20  56  OPEN   1*    1*    0.708   3*     Z /
         OP01    OP01LGR    14  13  75 100  SHUT   1*    1*    0.708   3*     Z /
         OP02    OP02LGR    35  96  75 100  OPEN   1*    1*    0.708   3*     Z /
/
--
--       ASSIGN WELL CONNECTIONS TO COMPLETIONS
--
--       WELL    LGR      --- LOCATION ---  COMPL
--       NAME    NAME      II  JJ  K1  K2   NO.
COMPLMPL
             OP01    OP01LGR   1*  1*  20  56    1          / COMPLETION NO. 01
             OP01    OP01LGR   1*  1*  75 100    2          / COMPLETION NO. 02
             OP02    OP02LGR   1*  1*  75  85    1          / COMPLETION NO. 01
             OP02    OP21LGR   1*  1*  86 100    2          / COMPLETION NO. 02
/
```

Here the well OP01 connections (14, 13, 20) to (14, 13, 56) are assigned to completion number one and connections (14, 13, 75) to (14, 13, 100) are assigned to completion number two. Well OP02 has only one set of connection data from cells (35, 96, 75) to cells (35, 96, 100), but they have split into two separate completion intervals, with connections (35. 96, 75) to (35. 96, 85) assigned to completion interval number one and (35, 96, 86) to (35, 96, 100) to completion number two.

12.3.31 COMPLUMP – ASSIGN WELL CONNECTIONS TO COMPLETIONS

Description

The COMPLUMP keyword assigns connections, as defined by the COMPDAT keyword in the SCHEDULE section, to completion intervals. This "lumping" of the connections to various completion intervals allows automatic workovers and economic criteria to be applied to the completions (that is a set of connections) as opposed to the connections. This allows for a more realistic approach for workovers operations.

For example, if the water cut criteria for working over a well was set to 95%, and the average grid block connection thickness was one meter, then once a well's water cut reached 95% the worst offending one meter connection would be shut-in. If the well's actual perforation interval was 10 meters and the 10 connections were lumped as one completion, then when the water cut limit of 95% is reach, the completion would be shut-in, that is all of the 10 connections within the completion would be shut-in.

As the keyword is used to lump connections into a completions, the simulator adds together the contribution from all connections in the completion and uses the total values to test the economic limits. Note that a connection can only belong to one completion. In addition, completions can be used instead of connections in the WELOPEN and WPIMULT keywords if the completions have been defined by COMPLUMP for a well.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WELNAME	A character string of up to eight characters in length that defines the well name for which the well connection data are being defined. Note that the well name (WELNAME) must have been declared previously using the WELSPECS keyword in the SCHEDULE section, otherwise an error may occur.			None
2	I	A positive integer greater than or equal to zero and less than or equal to NX that defines the connection location in the I-direction. If set to zero or defaulted with 1* then all connections in the I-direction that also satisfy J, K1 and K2 criteria are assigned the ICOMP completion number.			0
3	J	A positive integer greater than or equal to zero and less than or equal to NY that defines the connection location in the J-direction. If set to zero or defaulted with 1* then all connections in the J-direction that also satisfy I, K1 and K2 criteria are assigned the ICOMP completion number.			0
4	K1	A positive integer greater than or equal to one and less than or equal to NZ that defines the UPPER connection location in the K-direction. If set to zero or defaulted with 1* then the uppermost connection in the well is used.			0
5	K2	A positive integer greater than or equal to K1 and less than or equal to NZ that defines the LOWER connection location in the K-direction. If set to zero or defaulted with 1* then the lowermost connection in the well is used.			0

No.	Name	Description			Default
		Field	Metric	Laboratory	
6	ICOMP	An integer greater than or equal to one and less than or equal to MXCONS as defined on the WELLDIMS keyword in the RUNSPEC section, that defines the completion number of the currently defined set of connections. If I, J, K1, K2 are all set to zero or defaulted to 1*, then all connections in the well have the same completion number of ICOMP.			None
Notes: 1) The keyword is followed by any number of records. 2) Each record is terminated by a "/" and the keyword should be terminated by a "/".					

Table 12.15: COMPLUMP Keyword Description

Multiple grid block connections can be defined on one record for vertical wells by assigning different values to K1 and K2, for deviated and horizontal wells this may not be possible and therefore each grid block connection must be separately defined by setting K1 equal to K2.
See also the COMPDAT keyword in the SCHEDULE section.

Example

The following example defines the connections for two vertical oil wells using the COMPDAT keyword and the re-allocation of the connections to completions intervals using the COMPLUMP keyword.

```
--
--       WELL CONNECTION DATA
--
-- WELL   --- LOCATION ---   OPEN    SAT    CONN    WELL    KH     SKIN    D      DIR
-- NAME    II   JJ   K1   K2   SHUT    TAB    FACT    DIA     FACT   FACT    FACT   PEN
COMPDAT
OP01       14   13   20   56   OPEN    1*     1*     0.708    1*     0.0     1*     'Z' /
OP01       14   13   75 100    SHUT    1*     1*     0.708    1*     0.0     1*     'Z' /
OP02       35   96   75 100    OPEN    1*     1*     0.708    1*     0.0     1*     'Z' /
/
--
--       ASSIGN WELL CONNECTIONS TO COMPLETIONS
--
-- WELL   --- LOCATION ---   COMPL
-- NAME    II   JJ   K1   K2   NO.
COMPLUMP
OP01       1*   1*   20   56    1                              / COMPLETION NO. 01
OP01       1*   1*   75 100    2                              / COMPLETION NO. 02
OP02       1*   1*   75   85    1                              / COMPLETION NO. 01
OP02       1*   1*   86 100    2                              / COMPLETION NO. 02
/
```

Here the well OP01 connections (14, 13, 20) to (14, 13, 56) are assigned to completion number one and connections (14, 13, 75) to (14, 13, 100) are assigned to completion number two. Well OP02 has only one set of connection data from cells (35, 96, 75) to cells (35, 96, 100), but they have been split into two separate completion intervals, with connections (35, 96, 75) to (35. 96, 85) assigned to completion interval number one and (35, 96, 86) to (35, 96, 100) assigned to completion number two.

12.3.32 COMPOFF – DEACTIVATE NETWORK AUTOMATIC COMPRESSORS

Description

The COMPOFF keyword deactivates network automatic compressors defined via the GASFCOMP keyword in the SCHEDULE section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.33 COMPORD - DEFINE WELL CONNECTION ORDERING

Description

The COMPORD keyword defines how the well connection data entered on the COMPDAT keyword in the SCHEDULE section are to be ordered for a well.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WELNAME	A character string of up to eight characters in length that defines the well name for which the well connection data are being defined. Note that the well name (WELNAME) must have been declared previously using the WELSPECS keyword in the SCHEDULE section, otherwise an error may occur.			None
2	COMPORD	A character string that defines the method for ordering the well connections given on the COMPDAT keyword, and should be set to DEPTH, INPUT, or TRACK. 1) DEPTH: The connections are ordered by a connection's true vertical depth from the shallowest to the deepest. If multiple connections are at the same depth then these connections are sub ordered by the sequence they were entered on the COMPDAT keyword. 2) INPUT: This option results in the connections being ordered in the same sequence as entered via the COMPDAT keyword. In this case the connections should be declared in the correct sequence, starting with the connection nearest the well head and then working along the wellbore towards the bottom or toe of the well. 3) TRACK: This option enables OPM Flow to trace the well connections through the grid to obtain the correct order for the connections. If the supplied COMPDAT indicates the well is vertical (via the DIRECT variable being equal to Z on the COMPDAT keyword) then the DEPTH option will be applied instead. All options are now supported by OPM Flow.			TRACK
Notes: 1) The keyword is followed by any number of records. 2) Each record is terminated by a "/" and the keyword should be terminated by a "/".					

Table 12.16: COMPORD Keyword Description

See also the COMPDAT keyword in the SCHEDULE section.

Note

If visual inspection of the well trajectories in the model indicate problematic or unrealistic well connections, the options on this keyword may be useful in correcting the issue.

Example

The following example defines the connections for two vertical oil wells using the COMPDAT keyword and the COMPORD to defined the connection ordering for the wells.

```
--
--       WELL CONNECTION DATA
--
-- WELL   --- LOCATION ---   OPEN    SAT    CONN    WELL    KH     SKIN    D      DIR
-- NAME    II   JJ   K1   K2   SHUT    TAB    FACT    DIA     FACT   FACT    FACT   PEN
COMPDAT
OP01       1*   1*   20   56   OPEN    1*     1*     0.708    1*     0.0     1*     'Z' /
OP01       1*   1*   75  100   SHUT    1*     1*     0.708    1*     0.0     1*     'Z' /
OP02       35   96   75  100   OPEN    1*     1*     0.708    1*     0.0     1*     'Z' /
--
--       DEFINE WELL CONNECTION ORDERING
--
-- WELL   COMPL
-- NAME   ORDER
COMPORD
OP01      DEPTH                                                        /
OP02      DEPTH                                                        /
/
```

The DEPTH option has been chosen because both wells are vertical. Also one could use the following format instead for the COMPORD:

```
--
--       DEFINE WELL CONNECTION ORDERING
--
-- WELL   COMPL
-- NAME   ORDER
COMPORD
*         DEPTH                                                        /
/
```

as both wells should utilize the DEPTH option. This version would set all wells in the model to DEPTH connection ordering.

12.3.34 COMPRIV – DEFINE GRID CELL CONNECTIONS TO A RIVER

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The COMPRIV keyword defines grid cell connections to a river, for when the River option has been activated via the RIVRDIMS keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.35 COMPRP – RE-SCALE FLUID SATURATIONS OF WELL CONNECTIONS

Description

The COPMPRP keyword re-scales the fluid saturations of a well's connection to the grid block.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.36 COMPRPL – RE-SCALE FLUID SATURATIONS OF WELL LGR CONNECTIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The COPMPRPL keyword re-scales the fluid saturations of a well's connection to an LGR grid block, for when the Local Grid Refinement ("LGR") option has been activated by the LGR keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.37 COMPSEGL – DEFINE WELL CONNECTIONS FOR MULTI-SEGMENT WELLS IN A LGR

Description

The COMSEGSL keyword defines how a multi-segment well is connected to the reservoir by defining or modifying existing well connections in an LGR. Note that well must have been previously define by the WELSPECL keyword in the SCHEDULE section and the well connections must have been previously defined via the COMPDATL keyword in the SCHEDULE section. The COMPSEGL keyword should be repeated for each multi-segment well in the model.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1-1	WELNAME	A character string of up to eight characters in length that defines the well name for which a multi-segment well is being defined. Note that the well name (WELNAME) must have been declared previously using the WELSPECL keyword in the SCHEDULE section, otherwise an error may occur.			None
1-2	/	Record terminated by a "/"			Not Applicable
2-1	LGRNAME	A character string of up to eight characters in length that defines the name of the local grid refinement for which the well is assigned to.			None
2-2	I	A positive integer greater than or equal to one and less than or equal to NX that defines the connection location in the I-direction.			None
2-3	J	A positive integer greater than or equal to zero and less than or equal to NY that defines the connection location in the J-direction.			None
2-4	K	A positive integer greater than or equal to zero and less than or equal to NZ that defines the connection location in the K-direction.			None
2-5	IBRANCH	A positive integer greater than or equal to one and less than or equal to MXBRAN on WSEGDIMS keyword in the RUNSPEC section that defines the branch number of the defined I, J and K connection.			None
2.6	DEPTH1	DEPTH1 is a real positive value that defines the length of the tubing from the tubing head or wellhead at the surface to the start of the connection in the I, J, K cell.			
		feet	m	cm	None
2-7	DEPTH2	DEPTH2 is a real positive value that defines the length of the tubing from the tubing head or wellhead at the surface to the end of the connection in the I, J, K cell.			
		feet	m	cm	None

No.	Name	Description			Default
		Field	Metric	Laboratory	
2-8	DIRECT	A one letter character string that defines the orientation of the connections and should be set to either X, Y, or Z. The direction of connections also determines the length of the connection The default value is for a vertical connection, that is DIRECT is defaulted to Z.			Z
2-9	IEND	IEND is positive or negative integer, that is not equal to zero that is set to one of the following: 1) a value between -NX and +NX that is not equal to zero that defines the last connection location in the I-direction, 2) a value between -NY and +NY that is not equal to zero that defines the last connection location in the J-direction, or 3) a value between -NZ and +NZ that is not equal to zero that defines the last connection location in the K-direction, that defines the end of the range of the connections depending on the value of DIRECT. For example, if DIRECT is equal to Y or J then the IEND will be associated with the J-direction. The value may be positive or negative but must be calculated to remain within the grid. For example for NY is set 100 on the DIMENS keyword in the RUNSPEC section and J on this record set to 50, then IEND most range between -49 to +50. Currently this option is not supported by OPM Flow.			None
2-10	DEPTH3	DEPTH3 is a real positive value that defines the datum depth for this set of connection, normally taken as the mid-point of the perforations associated with this set of connections. Currently this option is not supported by OPM Flow.			
		feet	m	cm	None
2-11	LENGTH	LENGTH is a real positive value that defines the length of the well for this set of completions that is used in thermal calculations.. Currently this option is not supported by OPM Flow.			
		feet	m	cm	None
2-12	ISEG	A real positive values equal to or greater than zero that defines the coordinate in the x-direction of the nodal point of this segment that is used for display purposes only. Currently this option is not supported by OPM Flow.			
		feet	m	cm	None
2-13	/	Record terminated by a "/"			Not Applicable

No.	Name	Description			Default
		Field	Metric	Laboratory	

Notes:

1) Each multi-segment wells must be defined by a separate COMPSEGL keyword that consists of two records, with entries 1-1 to 1-2 representing record one items and 2-1 to 2-13 representing record number two items in the "No." column in this table.
2) Record number two of the keyword, items 2-1 to 2-13 is followed by up to MXCONS records as declared on the WSEGDIMS keyword in the RUNSPEC section.
3) Each of the records are terminated by a "/" and is explicitly shown in the above rows and the keyword should be terminated by a "/".

Table 12.17: COMPSEGL Keyword Description

The total number of wells and completions should be defined via the WELLSDIMS keyword and the number of multi-segment wells and completions should be declared on the WSEGDIMS keyword, both keywords are in the RUNSPEC section.

See also the WELSPECL keyword to define wells in an LGR, the COMPDATL keyword to define the well connections for both ordinary wells and multi-segment wells with an LGR, and the COMPSEGS keyword to define a multi-segment connections in the global grid. All the aforementioned keywords are described in the SCHEDULE section.

Example

The following example defines the completions for two oil producing segment oil wells (OP01 and OP02) using the COMPSEGS keywords.

```
--        COMPLETION SEGMENT SPECIFICATION DATA
--
-- WELL
-- NAME
COMPSEGS
OP01                                                                    /
--       LGR   --LOCATION--   BRAN   TUBING   NODAL    DIR   LOC     MID     COMP     ISEG
--       NAME   II   JJ   K1    NO     LENGTH   DEPTH    PEN   I,J,K   PERFS   LENGTH
         LGR01 10   10    1     1     2512.5   2525.0                                  /
         LGR01 10   10    2     1     2525.0   2550.0                                  /
         LGR01 10   10    3     1     2550.0   2575.0                                  /
         LGR01 10   10    4     1     2575.0   2600.0                                  /
         LGR01 10   10    5     1     2600.0   2625.0                                  /
         LGR01 10   10    6     1     2625.0   2650.0                                  /

         LGR01  9   10    2     2     2637.5   2837.5                                  /
         LGR01  8   10    2     2     2837.5   3037.5                                  /
         LGR01  7   10    2     2     3037.5   3237.5                                  /
         LGR01  6   10    2     2     3237.5   3437.5                                  /
         LGR01  5   10    2     2     3437.5   3637.5                                  /
/
```

Note that the COMPDATL keyword in the SCHEDULE section must also be defines for this well.

12.3.38 COMPSEGS – DEFINE WELL CONNECTIONS FOR MULTI-SEGMENT WELLS

Description

The COMSEGS keyword defines how a multi-segment well is connected to the reservoir by defining or modifying existing well connections. Note that well must have been previously define by the WELSPECS keyword in the SCHEDULE section and the well connections must have been previously defined via the COMPDAT keyword in the SCHEDULE section

The COMPSEGS keyword should be repeated for each multi-segment well in the model.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1-1	WELNAME	A character string of up to eight characters in length that defines the well name for which a multi-segment well is being defined. Note that the well name (WELNAME) must have been declared previously using the WELSPECS keyword in the SCHEDULE section, otherwise an error may occur.			None
1-2	/	Record terminated by a "/"			Not Applicable
2-1	I	A positive integer greater than or equal to one and less than or equal to NX that defines the connection location in the I-direction.			None
2-2	J	A positive integer greater than or equal to zero and less than or equal to NY that defines the connection location in the J-direction.			None
2-3	K	A positive integer greater than or equal to zero and less than or equal to NZ that defines the connection location in the K-direction.			None
2-4	IBRANCH	A positive integer greater than or equal to one and less than or equal to MXBRAN on WSEGDIMS keyword in the RUNSPEC section that defines the branch number of the defined I, J and K connection.			None
2-5	DEPTH1	DEPTH1 is a real positive value that defines the length of the tubing from the tubing head or wellhead at the surface to the start of the connection in the I, J, K cell.			
		feet	m	cm	None
2-6	DEPTH2	DEPTH2 is a real positive value that defines the length of the tubing from the tubing head or wellhead at the surface to the end of the connection in the I, J, K cell.			
		feet	m	cm	None
2-7	DIRECT	A one letter character string that defines the orientation of the connections and should be set to either X, Y, or Z. The direction of connections also determines the length of the connection The default value is for a vertical connection, that is DIRECT is defaulted to Z. Currently this option is not supported by OPM Flow.			Z

No.	Name	Description			Default
		Field	Metric	Laboratory	
2-8	IEND	IEND is positive or negative integer, that is not equal to zero that is set to one of the following: 1) a value between -NX and +NX that is not equal to zero that defines the last connection location in the I-direction, 2) a value between -NY and +NY that is not equal to zero that defines the last connection location in the J-direction, or 3) a value between -NZ and +NZ that is not equal to zero that defines the last connection location in the K-direction, that defines the end of the range of the connections depending on the value of DIRECT. For example, if DIRECT is equal to Y or J then the IEND will be associated with the J-direction. The value may be positive or negative but must be calculated to remain within the grid. For example for NY is set 100 on the DIMENS keyword in the RUNSPEC section and J on this record set to 50, then IEND most range between -49 to +50. Currently this option is not supported by OPM Flow.			None
2-9	DEPTH3	DEPTH3 is a real positive value that defines the datum depth for this set of connection, normally taken as the mid-point of the perforations associated with this set of connections. Currently this option is not supported by OPM Flow.			
		feet	m	cm	None
2-10	LENGTH	LENGTH is a real positive value that defines the length of the well for this set of completions that is used in thermal calculations.. Currently this option is not supported by OPM Flow.			
		feet	m	cm	None
2-11	ISEG	A real positive values equal to or greater than zero that defines the coordinate in the x-direction of the nodal point of this segment that is used for display purposes only. Currently this option is not supported by OPM Flow.			
		feet	m	cm	None
2-12	/	Record terminated by a "/"			Not Applicable

Notes:

1) Each multi-segment wells must be defined by a separate COMPSEGS keyword that consists of two records, with entries 1-1 to 1-2 representing record one items and 2-1 to 2-12 representing record number two items in the "No." column in this table.
2) Record number two of the keyword, items 2-1 to 2-12 is followed by up to MXCONS records as declared on the WSEGDIMS keyword in the RUNSPEC section.
3) Each of the records are terminated by a "/" and is explicitly shown in the above rows and the keyword should be terminated by a "/".

Table 12.18: COMPSEGS Keyword Description

The total number of wells and completions should be defined via the WELLSDIMS keyword and the number of multi-segment wells and completions should be declared on the WSEGDIMS keyword, both keywords are in the RUNSPEC section.

See also the WELSPECS keyword to define wells, the COMPDAT keyword to define the well connections for both ordinary wells and multi-segment wells, and the COMPSEGL keyword to define a multi-segment connections in a LGR. All the aforementioned keywords are described in the SCHEDULE section.

Example

The following example defines the completions for two oil producing segment oil wells (OP01 and OP02) using the COMPSEGS keywords.

```
--        COMPLETION SEGMENT SPECIFICATION DATA
--
-- WELL
-- NAME
COMPSEGS
OP01                                                                    /
--       --LOCATION--  BRAN  TUBING  NODAL   DIR  LOC    MID    COMP    ISEG
--        II  JJ  K1   NO    LENGTH  DEPTH   PEN  I,J,K  PERFS  LENGTH
          10  10   1    1    2512.5  2525.0                              /
          10  10   2    1    2525.0  2550.0                              /
          10  10   3    1    2550.0  2575.0                              /
          10  10   4    1    2575.0  2600.0                              /
          10  10   5    1    2600.0  2625.0                              /
          10  10   6    1    2625.0  2650.0                              /

           9  10   2    2    2637.5  2837.5                              /
           8  10   2    2    2837.5  3037.5                              /
           7  10   2    2    3037.5  3237.5                              /
           6  10   2    2    3237.5  3437.5                              /
           5  10   2    2    3437.5  3637.5                              /
/
--        COMPLETION SEGMENT SPECIFICATION DATA
--
-- WELL
-- NAME
COMPSEGS
OP02                                                                    /
--       --LOCATION--  BRAN  TUBING  NODAL   DIR  LOC    MID
--        II  JJ  K1   NO    LENGTH  DEPTH   PEN  I,J,K  PERFS
           1   9   3    1    2662.5  2862.5                              /
           1   8   3    1    2862.5  3062.5                              /
           1   7   3    1    3062.5  3262.5                              /
           1   6   3    1    3262.5  3462.5                              /
           1   5   3    1    3462.5  3662.5                              /

           2  10   5    2    2712.5  2912.5                              /
           2  10   5    2    2912.5  3112.5                              /
           4  10   5    2    3112.5  3312.5                              /
           5  10   5    2    3312.5  3512.5                              /
           6  10   5    2    3512.5  3712.5                              /

           1   9   6    3    2737.5  2937.5                              /
           1   8   6    3    2937.5  3137.5                              /
           1   7   6    3    3137.5  3337.5                              /
           1   6   6    3    3337.5  3537.5                              /
           1   5   6    3    3537.5  3737.5                              /
/
```

Note that the COMPDAT keyword in the SCHEDULE section must also be defined for these two wells.

12.3.39 COMPTRAJ – DEFINE WELL TRAJECTORY CONNECTIONS TO THE GRID

Description

The COMPTRAJ keyword defines how a well that has been declared as a trajectory well, using the WELTRAJ keyword in the SCHEDULE section, is connected to the reservoir model by defining or modifying existing well perforation depths. The keyword can only be used for wells defined by the WELTRAJ keyword, and WELTRAJ defined wells must use the COMPTRAJ keyword to define the connections to the grid, that is one cannot use COMPDAT for these type of wells.

Note

This is an OPM Flow specific keyword and will therefore cause an error in the commercial simulator.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WELNAME	A character string of up to eight characters in length that defines the well name for which the well connection data are being defined. Note that the well name (WELNAME) must have been declared previously using the WELSPECS keyword in the SCHEDULE section, otherwise an error may occur.			None
2	IBRANCH	A positive integer greater than or equal to one and less than or equal to MXBRAN on WSEGDIMS keyword in the RUNSPEC section that defines the branch number of a segment. All segments on the main stem must have IBRANCH set to one and lateral branches should have values between two and MXSEGS on the WSEGDIMS keyword in the RUNSPEC section. Only the default value of one is currently supported, that is only the main branch of a multi-segment well is supported, or a single trajectory for a conventional well.			1
3	TOP	A real positive value that defines depth of top of the perforation interval, and should be less than the value entered for the BOT parameter (the bottom perforation depth).			
		feet	m	cm	None
4	BOT	A real positive value that defines the depth of the base of the perforation interval, and should be greater than the value entered for the TOP parameter (the top perforation interval).			
		feet	m	cm	None
5	REF	REF is a defined character string that defines the reference depth type for TOP and BOT, and should be set to either: 1) MD for the depths referencing Measured Depth, or 2) TVD for depths referencing True Vertical Depth. Only measured depth is currently supported, that is MD.			MD

No.	Name	Description			Default
		Field	Metric	Laboratory	
6	ICOMP	An integer greater than or equal to one and less than or equal to MXCONS as defined on the WELLDIMS keyword in the RUNSPEC section, that defines the completion number of the currently defined perforation interval (connection interval). If defaulted with 1*, then ICOMP is set equal to one.			1
7	STATUS	A defined character string of length four that defines the connections' operational status within the perforation interval, STATUS should be set to one of the following character strings: 1) OPEN: the connections are open to flow. 2) SHUT: the connections are closed to flow (shut-in). 3) AUTO: the connection are initially closed, but may be opened automatically if an economic limit is violated. Currently this option is not supported by OPM Flow.			OPEN
8	SATNUM	An integer greater than or equal to zero and less than NTSFUN as declared on the TABDIMS keyword in the RUNSPEC, that defines the saturation table number to be used for flow between the reservoir grid block and the well connections. If SATNUM is set to zero or defaulted with 1* then: 1) The saturation table allocated to the grid block that the connections are located within are used. 2) If the hysteresis option has been activated via the HYSTER variable on the SATOPTS keyword in the RUNSPEC section, then both the imbibition and drainage saturation tables allocated to the grid block that the connections are located within are used. The imbibition table allocation can be changed by the COMPIMB keyword in the RUNSPEC section, provided it is entered after the COMPTRAJ keyword.			0
9	CONFACT	A real value greater than or equal to zero that defines the transmissibility connection factor between the well bore and the reservoir grid block. If set to zero or defaulted with 1* then items (10) through (13) are used to calculate CONFACT.			
		cP.rb/day/psia 0	cP.rm^3/day/bars 0	cP.rcc/hr/atm 0	Defined
10	DW	A real positive value that defines the well bore diameter of the connections for the well. DW is used in calculating a well's productivity or injectivity index; however the value will be ignored in calculating the connections CONFACT value if CONFAC has been directly entered.			
		feet	m	cm	None
11	KH	A real value that defines the effective KH (permeability x length) for the connections within the perforation interval. If less than or equal to zero, or defaulted by 1*, then KH is calculated from the connected grid blocks. KH is ignored if CONFACT has been directly entered.			Calculated from connected grid blocks

No.	Name	Description			Default
		Field	Metric	Laboratory	
		mD.ft	mD.m	mD.cm	
12	SKIN	A real value that defines the connections dimensionless skin factor. SKIN is used in calculating a well's productivity or injectivity index; however, the value will be ignored in calculating the connections CONFACT value if CONFACT has been directly entered.			
		dimensionless	dimensionless	dimensionless	0.0
13	DFACT	A real value that defines the non-Darcy D factor coefficient for gas wells. This value should be defaulted with 1* and the non-Darcy D factor coefficient for gas wells defined via the WDFAC keyword in the SCHEDULE section. Currently this option is not supported by OPM Flow.			
		day/Mscf	day/m^3	hour/sc	1*

Notes:

1) Note unlike the COMPDAT keyword, there is no direction parameter, COMPDAT(DIRECT), for COMPTRAJ, as the wellbore direction is implicit in the well trajectory defined by the WELTRAJ keyword in the SCHEDULE section.
2) The keyword is followed by up to MXCONS records as declared on the WELLDIMS keyword in the RUNSPEC section.
3) Each record is terminated by a "/" and the keyword should be terminated by a "/".

Table 12.19: COMPTRAJ Keyword Description

Using the WELTRAJ and COMPTRAJ keywords to define wells and how they are connected to grid, offers several advantages compared to the conventional approach based on the (I, J, K) co-ordinates of the grid. The approach allows for the wells to be independent of the grid, which is particularly useful when running ensemble cases, as the well connections are no longer required to be re-calculated for each ensemble case. In addition, quality control of the model is improved by using consistent perforation data in both the static and dynamic models.

Note

The term well connection is used to describe individual connections from the wellbore to the reservoir grid, as opposed to well completions. A well completion is used to describe a set of connections, for example, a well may consist of several completions with each completion consisting of multiple connections.

For wells defined using the WELTRAJ and COMPTRAJ keywords, the WELTRAJ keyword defines the trajectory of the well within the model, and the COMPTRAJ defines the perforation intervals in the well. A perforation interval will automatically generate various well connections to the grid, and in addition multiple perforation intervals may be grouped into a completion.

Example

The following example defines two trajectory wells oil wells, OP01 and OP02, using the WELTRAJ keyword, together with their perforations using the COMPTRAJ keyword.

```
--
--       WELL SPECIFICATION DATA
--
-- WELL  GROUP     LOCATION  BHP    PHASE  DRAIN  INFLOW  OPEN  CROSS  PVT
-- NAME  NAME        I    J  DEPTH  FLUID  AREA   EQUANS  SHUT  FLOW   TABLE
WELSPECS
OP01     PLATFORM   1*   1*   1*     OIL    1*     STD    SHUT   NO     1*     /
OP02     PLATFORM   1*   1*   1*     OIL    1*     STD    SHUT   NO     1*     /
/
--
--       WELL TRAJECTORY DATA
--
-- WELL   BRAN  XCORD          YCORD          TVDSS          MD
-- NAME   NO                                 DEPTH          DEPTH
-- -----  ----  ------------  ------------  ------------  ------------
WELTRAJ
OP01      1*    2.805445e+06  3.602948e+06  -100.000000   0.0          /
OP01      1*    2.805445e+06  3.602948e+06  877.0000000   977.0        /
OP01      1*    2.805445e+06  3.602948e+06  957.9950240   1058.0       /
OP01      1*    2.805444e+06  3.602946e+06  1051.976081   1152.0       /
.....                                                                  /
OP02      1*    2.810828e+06  3.604507e+06  9371.792711   11418.0      /
OP02      1*    2.810885e+06  3.604525e+06  9443.657000   11511.0      /
OP02      1*    2.810952e+06  3.604546e+06  9531.966162   11624.0      /
OP02      1*    2.810973e+06  3.604553e+06  9560.411742   11660.0      /
/
--
--
--       WELL TRAJECTORY CONNECTION DATA
--
-- WELL  BRAN  -- PERFORATION --  COMPL OPEN  SAT CONN  WELL   KH    SKIN  D
-- NAME  NO.    TOP    BOT   REF  NO.   SHUT  TAB FACT  DIA    FACT  FACT  FACT
COMPTRAJ
OP01     1*     8230   8244  MD    1    SHUT  1*  1*   0.708  1*    0.0   1*  /
OP01     1*     8352   8380  MD    1    SHUT  1*  1*   0.708  1*    0.0   1*  /
OP01     1*     9070   9100  MD    1    SHUT  1*  1*   0.708  1*    0.0   1*  /
OP01     1*     9220   9250  MD    2    SHUT  1*  1*   0.708  1*    0.0   1*  /
OP01     1*     9266   9280  MD    2    SHUT  1*  1*   0.708  1*    0.0   1*  /
OP01     1*     9693   9703  MD    3    SHUT  1*  1*   0.708  1*    0.0   1*  /
OP01     1*     9940   9974  MD    3    SHUT  1*  1*   0.708  1*    0.0   1*  /
OP02     1*     9979   9985  TVD   1*   SHUT  1*  1*   0.708  1*    0.0   1*  /
OP02     1*    10173  10183  TVD   1*   SHUT  1*  1*   0.708  1*    0.0   1*  /
OP02     1*    10190  10204  TVD   1*   SHUT  1*  1*   0.708  1*    0.0   1*  /
OP02     1*    10327  10333  TVD   1*   SHUT  1*  1*   0.708  1*    0.0   1*  /
OP02     1*    10339  10345  TVD   1*   SHUT  1*  1*   0.708  1*    0.0   1*  /
OP02     1*    11528  11538  TVD   1*   SHUT  1*  1*   0.708  1*    0.0   1*  /
/
```

Well OP01 has eight perforation intervals, with the intervals one to three grouped into one completion, perforation intervals four to five grouped into completion number two, and finally the bottom three perforations are grouped into completion number three. In contrast, OP02 has six perforated intervals with their completion interval defaulted to one.

12.3.40 COMPVE – RE-DEFINE WELL CONNECTION DEPTHS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The COMPVE keyword is used to re-define the well connection depths in the global grid.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.41 COMPVEL – RE-DEFINE WELL LGR CONNECTION DEPTHS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The COMPVEL keyword is used to re-define the well connection depths in a Local Grid Refinement ("LGR") grid.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.42 CPIFACT – DEFINE WELL CONNECTION TRANSMISSIBILITY MULTIPLIERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The CPIFACT keyword is used to define well connection transmissibility multipliers for well connections to the global grid.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.43 CPIFACTL – DEFINE WELL CONNECTION TRANSMISSIBILITY MULTIPLIERS IN A LGR

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The CPIFACT keyword is used to define well Local Grid Refinement ("LGR") connection transmissibility multipliers for well connections to a LGR, for when the LGR option has been invoked by the LGR keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.44 CSKIN – RE-DEFINE WELL CONNECTION SKIN FACTORS

Description

This keyword, CSKIN, is used to re-define a well's connection skin factors and as such will result in the well's connection transmissibility factors being updated accordingly. The well connections must already be defined using the COMPDAT keyword in the SCHEDULE section before this keyword can be used.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WELNAME	A character string of up to eight characters in length that defines the well name for which the well connection skin is being re-defined. Note that the well name (WELNAME) must have been declared previously using the WELSPECS keyword in the SCHEDULE section, otherwise an error may occur.			None
2	I	A positive integer greater than or equal to one and less than or equal to NX that defines the connection location in the I-direction. If set to zero or defaulted with 1* then connections in any I-direction location will be modified.			0
3	J	A positive integer greater than or equal to one and less than or equal to NY that defines the connection location in the J-direction. If set to zero or defaulted with 1* then connections in any J-direction location will be modified.			0
4	K1	A positive integer greater than or equal to one and less than or equal to K2 and NZ that defines the UPPER connection location in the K-direction. If set to zero or defaulted with 1* then this will be taken from the top connection of the well.			0
5	K2	A positive integer greater than or equal to K1 and less than or equal to NZ that defines the LOWER connection location in the K-direction. If set to zero or defaulted with 1* then this will be taken from the bottom connection of the well.			0
6	SKIN	A real value that defines the connections' dimensionless skin factors.			
		dimensionless	dimensionless	dimensionless	0.0
Notes: 1) Each record is terminated by a "/" and the keyword should be terminated by a "/".					

Table 12.20: CSKIN Keyword Description

Example

The following example re-defines the skin factor for all well OP01 connections in layers 4 to 6.

```
--
--        CONNECTION SKIN DATA
--
-- WELL   --- LOCATION ---   SKIN
-- NAME    II   JJ   K1   K2   FACT
CSKIN
OP01       1*   1*    4    6   2.0    /
/
```

12.3.45 DATES – ADVANCE SIMULATION BY REPORTING DATE

Description

This keyword advances the simulation to a given report date after which additional keywords may be entered to instruct OPM Flow to perform additional functions via the SCHEDULE section keywords, or further DATES data sets or keywords may be entered to advance the simulator to the next report date.

If the DATES keyword is to be used during the simulation, then the START keyword in the RUNSPEC section must be declared to set the start date for the run.

No.	Name	Description	Default
1	DAY	A positive integer that defines the day of the month for the data set, the value should be greater than or equal to one and less than or equal to 31.	None
2	MONTH	Character string for the month for the data set and should be one of the following 'JAN', 'FEB', 'MAR', 'APR', 'MAY', 'JUN', 'JUL' (or 'JLY'), 'AUG', 'SEP', 'OCT', 'NOV', or 'DEC'	None
3	YEAR	A positive four digit integer value representing the year for the data set, which must be specified fully by four digits, that is 1986.	None
4	TIME	A numeric character string that defines the time for the data set in the form of: HH:MM:SS.SSSS The default value means in most cases this parameter can be defaulted. TIME is normally used when detailed DST matching is performed to enable the pressures and rates to be stated at specific dates and times.	00:00:00
Notes: 1) The keyword is followed by a number of data sets (or rows) representing one DATE record per row. 2) Each record (or row) is terminated by a "/" and the keyword is terminated by a "/".			

Table 12.21: DATES Keyword Description

Note that OPM Flow uses the standard Gregorian calendar and therefore leap years are accounted for in the DATES keyword. Thus, it is more accurate to use the DATES keyword to progress the simulator through time if one is matching actual production data.

See also the TIME and TSTEP keywords in the SCHEDULE section.

Whenever possible it is a good idea to always set the start date to be at the beginning of the year, as like most simulators, OPM Flow reports are always stated at the number of days from the start date (and sometimes at a given date). If the start date is at the beginning of the year, then calculating the actual date is relatively straight forward and simple.

Examples

Given a start date of January 1, 2020 set via the START keyword in the RUNSPEC section, the following example advances the simulator from the start date of January 1, 2020 to January 1, 2021, using quarterly reporting time steps.

```
-- ================================================================================
--
-- SCHEDULE SECTION
--
-- ================================================================================
SCHEDULE

-- --------------------------------------------------------------------------------
-- SCHEDULE SECTION - 2020-01-01
-- --------------------------------------------------------------------------------
RPTSCHED
          'WELLS=2'    'WELSPECS'    'CPU=2'     FIP=2'                          /

DATES
          2  JAN   2020  /
/

RPTSCHED
          'NOTHING'                                                             /

DATES
          1  APR   2020  /
          1  JLY   2020  /
          1  OCT   2020  /
/
-- --------------------------------------------------------------------------------
-- SCHEDULE SECTION - 2021-01-01
-- --------------------------------------------------------------------------------
RPTSCHED
          'WELLS=2'    'WELSPECS'    'CPU=2'     FIP=2'                          /

DATES
          1  JAN   2021  /
/
RPTSCHED
          'NOTHING'                                                             /

DATES
          1  APR   2021  /
          1  JLY   2021  /
          1  OCT   2021  /
/
```

The above example writes out a series of report at the start of the run and then advances the simulation one day to January 2, 2020 and switches off the reporting. The simulation then advances to April 1, July 1 and October 1, 2020 with no further changes to the run. After October 1, 2020 reporting is switched on again to enable a report on January 1, 2021, which is then subsequently switched off after the January 1, 2021 report time step has been completed.

Note if one wishes to terminate the run at the end of year (as opposes to the beginning of the year and get a final report for the year, then the next example demonstrates the keyword sequence to enable this.

```
-- ------------------------------------------------------------------------
-- SCHEDULE SECTION - 2021-01-01
-- ------------------------------------------------------------------------
RPTSCHED
          'WELLS=2'    'WELSPECS'    'CPU=2'     FIP=2'                          /

DATES
          2  JAN   2021  /
/

RPTSCHED
         'NOTHING'                                                             /

DATES
          1  FEB   2021  /
          1  MAR   2021  /
          1  APR   2021  /
          1  MAY   2021  /
          1  JUN   2021  /
          1  JLY   2021  /
          1  AUG   2021  /
          1  SEP   2021  /
          1  OCT   2021  /
          1  NOV   2021  /
          1  DEC   2021  /
/
--
-- FINAL REPORT AND RESTART AT YEAR END
--
RPTSCHED
          'WELLS=2'    'WELSPECS'    'CPU=2'     FIP=2'                          /

RPTRST
          'BASIC=2'                                                            /

DATES
           31 DEC   2021  /
/
```

In the above example monthly reporting time steps have been used instead of quarterly and report is requested after the December 1, 2021 time step and is therefore written out on December 31, 2021.

12.3.46 DCQDEFN – DEFINE GAS DCQ UNITS AS RATE OR ENERGY

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The DCQDEFN keyword defines the DCQ units to be rate or energy (calorific value) when using the Gas Field Operation model and the Gas Calorific Value control option. The gas DCQ rates are controlled by the GASYEAR, GASPERIO, GDCQ, GASFTARG or GASFDECR keywords in the SCHEDULE section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.47 DEBUG – DEFINE THE DEBUG DATA TO BE PRINTED TO FILE

This keyword defines the debug data to be written to the debug file (*.DBG). This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

See *DEBUG – Define the Debug Data to be Printed to File* in the GLOBAL section for a full description.

12.3.48 DELAYACT - DEFINE DELAYED ACTION KEYWORDS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The DELAYACT keyword defines a series of keywords that should be executed after an ACTION keyword has been triggered by the ACTION, ACTIONG, ACTIONR, ACTIONW, ACTIONS, or ACTIONX keywords.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.49 DIFFMMF – DEFINE DIFFUSIVITY MULTIPLIERS FOR MATRIX-FRACTURES

This keyword, DIFFMMF, defines the diffusivity multipliers for matrix-fractures for when the Dual Porosity option has be activated by either the DUALPORO or DUALPERM keywords, or the Coal Bed Methane option is selected by the COAL keyword, and the Diffusivity option has been activated by the DIFFUSE keywords; all four keywords are in the RUNSPEC section.

See *DIFFMMF – Define Diffusivity Multipliers for Matrix-Fractures* in the GRID section for a full description.

12.3.50 DIMPES – DEFINE IMPES DYNAMIC SOLUTION PARAMETERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, DIMPES, defines the Implicit in Pressure Explicit Saturation ("IMPES") dynamic solution parameters and results in the simulator switching from the current solution formulation to the IMPES formulation. OPM Flow uses a different numerical scheme which makes this keyword redundant; hence, OPM Flow ignores this keyword. It is documented here for completeness.

There is no data required for this keyword.

See section *2.2 Running OPM Flow 2023-10 From The Command Line* on how to invoke various numerical schemes via the OPM Flow command line interface.

Example

```
--
--       ACTIVATE THE IMPES SOLUTION OPTION
--
DIMPES
```

The above example switches on the IMPES solution option; however, this has no effect in OPM Flow input decks.

12.3.51 DIMPLICT – ACTIVATE FULLY IMPLICIT DYNAMIC SOLUTION FORMULATION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, DIMPLICT, activates the Fully Implicit Formulation and results in the simulator switching from the current solution formulation to the fully implicit formulation. OPM Flow uses a different numerical scheme which makes this keyword redundant; hence, OPM Flow ignores this keyword. It is documented here for completeness.

There is no data required for this keyword.

See section *2.2 Running OPM Flow 2023-10 From The Command Line* on how to invoke various numerical schemes via the OPM Flow command line interface.

Example

```
--
--       ACTIVATES THE FULLY IMPLICIT SOLUTION OPTION
--
DIMPLICT
```

The above example switches on the fully implicit solution option; however, this has no effect in OPM Flow input decks.

12.3.52 DRILPRI – DEFINE PRIORITIZED DRILLING QUEUE PRIORITY PARAMETERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, DRILPRI, defines the prioritized drilling queue priority parameters used in the priority formulae for this type of drilling queue.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.53 DRSDT – SOLUTION GAS (RS) MAXIMUM RATE OF INCREASE PARAMETERS

Description

DRSDT defines the maximum rate at which the solution gas-oil ratio (Rs) can be increased in a grid cell. The keyword is similar in functionality to the DRSDTR keyword, that defines the maximum rate at which Rs can be increased in a grid cell by region. Both keywords should only be used if the OIL, GAS, and DISGAS keywords in the RUNSPEC section have been invoked to allow oil, gas and dissolved gas to be present in the model. The keyword only affects the behavior of an increasing Rs, for example when gas is being injected into an oil reservoir, and is subject to the availability of free gas and the ability of the undersaturated oil to adsorb this gas.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	DRSDT1	DRSDT1 is a real positive number that defines the maximum rate at which the solution gas-oil ratio is allowed to increase in a grid cell, that is the maximum rate the gas can dissolve into the available undersaturated oil. A value of zero means that Rs cannot increase and free gas cannot dissolve into the unsaturated oil in a grid cell. Alternatively a very large value of DRSDT1 allows Rs to increase rapidly until there is no free gas or the oil within the grid block is fully saturated. Note if the keyword is not present in the input deck then DRSDT1 is assumed to be a very large number resulting in complete re-solution of the gas into the available undersaturated oil.			
		Mscf/stb/d	sm^3/sm^3/day	scc/scc/day	None
2	DRSDT2	DRSDT2 is a defined character string that defines whether the DRSDT1 is applied to either all grid blocks or just those grid blocks containing free gas: 1) ALL: means the DRSDT1 maximum rate at which Rs is allowed to increase in a grid cell is applied to all grid blocks. 2) FREE: means the DRSDT1 maximum rate at which Rs is allowed to increase in a grid cell is applied to grid blocks only containing free gas. Note if the keyword is not present in the input deck then DRSDT2 is set to the default value of ALL.			ALL
Notes: 1) The keyword is terminated by a "/".					

Table 12.22: DRSDT Keyword Description

Note this keyword can be used in history matching field performance to control the availability of the movable gas phase.

See also the VAPPAR keyword in the SOLUTION section and the DRSDTR, DRVDT and DRVDTR keywords in the SCHEDULE section that controls how vaporized oil is treated and the rate at which the dissolved phase ratio increases within a grid block.

Examples

The first example prevents the solution gas-oil ratio from increasing and applies this to all grid cells.

```
--
--       SOLUTION GAS (RS) MAXIMUM RATE OF INCREASE FOR MODEL
--
DRSDT
--       MAX RS    ALL/FREE
--       DRSDT1    DRSDT2
--       -------   --------
         0.000     ALL                                    /
```

And the second example below applies 0.005 Mscf/stb/d as the maximum rate at which the solution gas-oil ratio is allowed to increase in a grid cell, and applies this to only cells containing free gas.

```
--
--       SOLUTION GAS (RS) MAXIMUM RATE OF INCREASE FOR MODEL
--
DRSDT
--       MAX RS    ALL/FREE
--       DRSDT1    DRSDT2
--       -------   --------
         0.0005    FREE                                   /
```

Again, the keyword parameters when applied are subject to the availability of free gas and the ability of the undersaturated oil to adsorb this gas.

12.3.54 DRSDTCON – CO2 CONVECTIVE DISSOLUTION PARAMETER (χ)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

DRSDTCON defines a dimensionless parameter (χ) that controls convective dissolution of carbon dioxide (CO2) into in situ brine within a grid cell, as described by Sandve et al.[303], based on an assumption of vertical equilibrium[304]. The keyword internally causes the simulator to calculate the solution gas-oil ratio (Rs), as normally defined by the DRSDT1 parameter on the DRSDT keyword in the SCHEDULE section, making the DRSDT keyword redundant.

The keyword should only be used if the CO2STORE keyword in the RUNSPEC section has been activated to model CO2 storage via OPM Flow's CO2-Brine PVT model. Both of these keywords are OPM Flow specific keywords. The DRSDTCON keyword must be used together with the GAS and OIL keywords in the RUNSPEC section to declare that the gas and oil phases are present in the model. Internally the simulator refers to the oil phase as the brine phase and the gas phase to the CO2. Input / output keywords need to be consistent with this assumption. i.e., SGOF (gas-oil relative permeability) is used for the CO2-Brine relative permeability, FOIP (Field Oil-In-Place) shows the total amount of brine in the reservoir, etc.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	DRSDTCON	DRSDTCON is a real positive number that defines a dimensionless parameter (χ) that controls convective dissolution of CO_2 into in situ brine within a grid cell. A value of zero means that convective dissolution of CO2 into in situ brine does not occur and free CO2 cannot dissolve into the brine in a grid cell. Alternatively a none zero value of DRSDTCON allows convective dissolution of CO2. Note if the CO2STORE keyword is present but the DRSDTCON is absent from the input deck, then this results in instantaneous dissolution of CO2 into the available undersaturated in situ brine.			
		dimensionless	dimensionless	dimensionless	0.0

Notes:

1) The keyword is terminated by a "/".

Table 12.23: DRSDTCON Keyword Description

The dissolution rate[305], F, in kg /(m2s) is defined as:

$$F = \chi\left(\frac{c_{max}\, K_z \Delta\rho_c\, g}{\mu}\right) \quad (12.13)$$

Where:

[303] *Tor Harald Sandve1, Sarah E. Gasda, Atgeirr Rasmussen, and Alf Birger Rustad. Convective dissolution in field scale CO_2 storage simulation using the OPM Flow simulator. Submitted to TCCS 11 – Trondheim Conference on CO_2 Capture, Transport and Storage Trondheim, Norway – June 21-23, 2021.*

[304] *Since the oil phase represents the brine phase phase and the gas phase is the CO_2 phase, then effectively the Rs (DRSDT) models the maximum rate at which CO_2 can dissolve into the brine phase. Thus, there is no need to enter data for the DRSDT keyword as this is computed internally.*

[305] *Elenius, M. T., Nordbotten, J. M., & Kalisch, H. (2014). Convective mixing influenced by the capillary transition zone. Computational Geosciences, 18(3-4), 417-431*

χ	=	a non-dimensional parameter controlling the dissolution,
Cmax	=	the maximum concentration at the solubility limit,
Kz	=	the vertical permeability,
$\Delta\rho c$	=	the difference of the brine density at maximum amount of dissolved CO2 and the density without dissolved CO2.
g	=	the gravity constant, and
μ	=	the viscosity.

Now in the standard black-oil model the solution gas oil ratio (Rs) is given by:

$$Rs = \frac{Volume\,of\ Gas_{ref}}{Volume\,of\ Oil_{ref}} \tag{12.14}$$

and relating Rs to mass we have:

$$Rs = \left(\frac{\chi_o^g}{1-\chi_o^g}\right)\left(\frac{\rho_{o,ref}}{\rho_{g,ref}}\right) \tag{12.15}$$

Where:

χgo	=	the mass fraction of gas in the oil phase,
ρo, ref	=	density of oil at reference conditions,
$\rho_{g,\,ref}$	=	density of gas at reference conditions,

In the black-oil model, the dissolution rate (as defined by the DRSDT keyword) is defined as the maximum rate at which the solution gas-oil ratio (Rs) can be increased in a grid cell per time. Thus, in order to convert equation (12.13) into a black-oil formulation we first need to replace the maximum concentration at the solubility limit used in the equation (12.13)) with its black-oil equivalent, Rssat. where Rssat is defined as:

$$\begin{aligned} Rs_{sat} &= \frac{V_{CO_2,max,ref}}{V_{brine,ref}} \\ &= \frac{m_{CO_2,max}}{V_{brine}\ B_{brine}\ \rho_{CO_2,ref}} \end{aligned} \tag{12.16}$$

Where:

$m_{CO2,\,max}$	=	the mass of CO2 at the maximum solubility limit, and
B_{brine}	=	the brine formation volume factor

The next step is to convert F in equation (12.13) from kg/(m^{2}s) to change in Rs (gas-oil ratio at reference conditions) per day. This is done by multiplying the cell top face area (A) by a conversion factor τ=86,400 (s/day), then dividing by the volume of the brine and the density of the CO_2 both at reference condition, that is:

$$\frac{\tau F\ A}{V_{brine,ref}\ \rho_{CO_2,ref}} = \frac{\tau F\ A}{V\ S_{brine,ref}\ \phi\ \rho_{CO_2,ref}\ B_{brine}} \tag{12.17}$$

Where:

S_{brine}	=	the brine saturation within a cell,
V	=	the cell volume, estimate by $V \approx A\ D_z$ in which D_z is the cell thickness and,
Φ	=	the cell's porosity.

Combining equations (12.16) and (12.17) to obtain the maximum dissolution rate, as per the DRSDT keyword in the SCHEDULE section in units of Sm3/(Sm3/day), we obtain:

$$DRSDT = \chi\left(\tau \frac{c_{max} K_z \Delta\rho_c g}{\mu S_{brine} D_z \phi}\right) \quad (12.18)$$

The value of χ can either be estimated from historical data, from fine-scale simulation, or from laboratory tests[306]. It typically varies from 0.01 - 0.1 depending on reservoir properties. Elenius et al.[307] states that 0.04 is a reasonable value for χ for the Utsira formation.

Note

In the commercial simulator a constant or regional value for DRSDT can be given as an input parameter. The DRSDT can be used to include the effect of convective mixing as shown by Thibeau and Dutin[308].

OPM Flow's approach differs from this in that the DRSDT value is computed internally and is dependent on both the static and dynamic cell properties.

Example

The example below is identical to the one shown under the CO2STORE keyword in the RUNSPEC section, and is repeated here for convenience. In the RUNSPEC section one declares that the carbon dioxide storage model is active for the run to account for both carbon dioxide and water phase solubility using OPM Flow's CO2-Brine PVT model.

```
-- ==========================================================================
--
-- RUNSPEC SECTION
--
-- ==========================================================================
RUNSPEC
-- --------------------------------------------------------------------------
-- FLUID TYPES AND TRACER OPTIONS
-- --------------------------------------------------------------------------
--
--       OIL PHASE IS PRESENT IN THE RUN BUT IS THE BRINE PHASE FOR CO2STORE
--
OIL
--
--       GAS PHASE IS PRESENT IN THE RUN BUT IS THE CO2 PHASE FOR CO2STORE
--
GAS
--
--       ACTIVATE CO2 STORAGE IN THE MODEL (OPM FLOW CO2 STORAGE KEYWORD)
--
CO2STORE
```

The second part of the example covers the data required for the PROPS section, in which the input keywords need to be consistent with the OIL phase referring to the Brine and the GAS to CO2; that is SGOF (gas-oil relative permeability) is used for the CO2-Brine relative permeability,.

```
-- ==========================================================================
--
```

[306] *Taheri, A., Torsæter, O., Lindeberg, E., Hadia, N. J., & Wessel-Berg, D. (2018). Qualitative and quantitative experimental study of convective mixing process during storage of CO2 in heterogeneous saline aquifers. International Journal of Greenhouse Gas Control, 71, 212-226.*

[307] *Elenius, M. T., Nordbotten, J. M., & Kalisch, H. (2014). Convective mixing influenced by the capillary transition zone. Computational Geosciences, 18(3-4), 417-431*

[308] *Thibeau, S., & Dutin, A. (2011). Large scale CO2 storage in unstructured aquifers: Modeling study of the ultimate CO2 migration distance. Energy Procedia, 4, 4230-4237.*

```
-- PROPS SECTION
--
-- ================================================================================
PROPS
--
--       ROCK COMPRESSIBILITY
--
--       REFERENCE PRESSURE IS TAKEN FROM THE HCPV WEIGHTED RESERVOIR PRESSURE
--
--       REF PRES  CF
--       BARSA     1/BARSA
--       --------  --------
ROCK
         1.0000    1.0E-06                                  / ROCK COMPRESSIBILITY
--
--       GAS-OIL RELATIVE PERMEABILITY TABLES (SGOF) - CO2STORE PHASES
SGOF
--       SG          KRG       KROG      PCOG
--       FRAC                            PSIA
--       -------   --------  -------   -------
         0.00000   0.000000  1.00000    0.0000
         1.00000   1.000000  0.00000    0.0000             / TABLE No. 01
--
--       SET SALINITY FOR ALL CELLS (OPM FLOW KEYWORD)
--
SALINITY
0.7                                                       /

-- ================================================================================
```

No other data is required to define the fluid and rock properties in the PROPS section as the data is generated from internal analytic correlations and models by the simulator. Finally, note that units for salinity are to the 10-3, thus for metric units we have 10-3 x kg-M/kg.

The third and final part of the example sets the maximum dissolution rate for convective CO_2 mixing via the DRSDTCON keyword in the SCHEDULE section.

```
-- ================================================================================
--
-- SCHEDULE SECTION
--
-- ================================================================================
SCHEDULE
--
--       CO2 CONVECTIVE DISSOLUTION PARAMETER
--
DRSDTCON
--       CO2 CONV
--       DISSOLN
--       --------
         0.04                                          /
```

See also the CO2STORE keyword in the RUNSPEC section for further information on OPM Flow's CO2 storage facility.

12.3.55 DRSDTR – SOLUTION GAS (RS) MAXIMUM RATE OF INCREASE PARAMETERS BY REGION

Description

DRSDTR defines the maximum rate at which the solution gas-oil ratio (Rs) can be increased in a grid cell for various regions in the model. The keyword is similar in functionality to the DRSDT keyword, that defines the maximum rate at which Rs can be increased in a grid cell for all cells in the model. The number of DRSDTR vector data sets is defined by the NTPVT parameter on the TABDIMS keyword in the RUNSPEC section and the allocation of the DRSDTR records to different grid blocks in the model is done via the PVTNUM keyword in the REGION section. One data set consists of one record or line which is terminated by a "/".

DRSDTR should only be used if the OIL, GAS, and DISGAS keywords in the RUNSPEC section have been invoked to allow oil, gas and dissolved gas to be present in the model. The keyword only affects the behavior of an increasing Rs, for example when gas is being injected into an oil reservoir, and is subject to the availability of free gas and the ability of the undersaturated oil to adsorb this gas.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	DRSDT1	DRSDT1 is a real positive number that defines the maximum rate at which the solution gas-oil ratio is allowed to increase in a grid cell, that is the maximum rate the gas can dissolve into the available undersaturated oil. A value of zero means that Rs cannot increase and free gas cannot dissolve into the unsaturated oil in a grid cell. Alternatively a very large value of DRSDT1 allows Rs to increase rapidly until there is no free gas or the oil within the grid block is fully saturated. Note if the keyword is not present in the input deck then DRSDT1 is assumed to be a very large number resulting in complete re-solution of the gas into the available undersaturated oil.			
		Mscf/stb/d	sm^3/sm^3/day	scc/scc/day	None
2	DRSDT2	DRSDT2 is a defined character string that defines whether the DRSDT1 is applied to either all grid blocks or just those grid blocks containing free gas: 1) ALL: means the DRSDT1 maximum rate at which Rs is allowed to increase in a grid cell is applied to all grid blocks. 2) FREE: means the DRSDT1 maximum rate at which Rs is allowed to increase in a grid cell is applied to grid blocks only containing free gas. Note if the keyword is not present in the input deck then DRSDT2 is set to the default value of ALL.			ALL

Notes:

1) The keyword is followed by NTPVT records as declared on the TABDIMS keyword in the RUNSPEC section.
1) Each record is terminated by a "/" and there is no "/" terminator for the keyword.

Table 12.24: DRSDTR Keyword Description

Note this keyword can be used in history matching field performance to control the availability of the movable gas phase.

See also the VAPPAR keyword in the SOLUTION section and the DRSDT, DRVDT and DRVDTR keywords in the SCHEDULE section that controls how vaporized oil is treated and the rate at which the dissolved phase ratio increases within a grid block.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored but is documented here for completeness, as it is expected to be available in the next release of OPM Flow.

Examples

The first example prevents the solution gas-oil ratio from increasing and applies this to all regions for when NTPVT on the TABDIMS keyword in the RUNSPEC section is set to three.

```
--
--       SOLUTION GAS (RS) MAXIMUM RATE OF INCREASE BY REGION
--
DRSDTR
--       MAX RS    ALL/FREE
--       DRSDT1    DRSDT2
--       -------   --------
         0.0000    ALL                                  /
         0.0000    ALL                                  /
         0.0000    ALL                                  /
```

The second example below prevents the solution gas-oil ratio from increasing and applies this to all grid cells in PVTNUM region one. For PVTNUM regions one and two the keyword applies 0.005 Mscf/stb/d as the maximum rate at which the solution gas-oil ratio is allowed to increase in a grid cell, and applies this to only cells containing free gas.

```
--
--       SOLUTION GAS (RS) MAXIMUM RATE OF INCREASE BY REGION
--
DRSDTR
--       MAX RS    ALL/FREE
--       DRSDT1    DRSDT2
--       -------   --------
         0.0000    ALL                                  /
         0.0005    FREE                                 /
         0.0005    FREE                                 /
```

Again, the keyword parameters when applied are subject to the availability of free gas and the ability of the undersaturated oil to adsorb this gas.

12.3.56 DRVDT – SOLUTION OIL (RV) MAXIMUM RATE OF INCREASE PARAMETERS

Description

DRVDT defines the maximum rate at which the solution oil-gas ratio or condensate-gas ratio (Rv) can be increased in a grid cell. The keyword is similar in functionality to the DRVDTR keyword, that defines the maximum rate at which Rv can be increased in a grid cell by region. Both keywords should only be used if the OIL, GAS, and VAPOIL (condensate) keywords in the RUNSPEC section have been invoked to allow oil, gas and condensate to be present in the model. The keyword only affects the behavior of an increasing Rv, for example when gas is being injected into a gas condensate reservoir as part of as gas re-cycling scheme, and is subject to the availability of free oil (condensate) and the ability of the undersaturated gas to adsorb this condensate.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	DRVDT1	DRVDT1 is a real positive number that defines the maximum rate at which the solution oil-gas ratio is allowed to increase in a grid cell, that is the maximum rate at which the oil can dissolve into the available undersaturated gas. A value of zero means that Rv cannot increase and free oil cannot dissolve into the unsaturated gas in a grid cell. Alternatively a very large value of DRVDT1 allows Rv to increase rapidly until there is no free oil or the gas within the grid block is fully saturated. Note if the keyword is not present in the input deck then DRVDT1 is assumed to be a very large number resulting in complete re-solution of the oil into the available undersaturated gas.			
		stb/Mscf/d	sm^3/sm^3/day	scc/scc/day	None
Notes: 1) The keyword is terminated by a "/".					

Table 12.25: DRVDT Keyword Description

Note this keyword can be used in history matching field performance to control the availability of the movable gas phase. See also the VAPPARS keyword in the SOLUTION section and the DRVDTR, DRSDT and DRSDTR keywords in the SCHEDULE section that controls how vaporized oil is treated and the rate at which the dissolved phase ratio increases within a grid block.

Example

The example prevents the solution oil-gas ratio from increasing.

```
--
--        SOLUTION OIL (RV) MAXIMUM RATE OF INCREASE FOR MODEL
--
DRVDT
--        MAX RV
--        DRVDT1
--        -------
          0.000                                                  /
```

Again, the keyword parameters when applied are subject to the availability of free oil and the ability of the undersaturated gas to adsorb this oil.

12.3.57 DRVDTR – SOLUTION OIL (RV) MAXIMUM RATE OF INCREASE PARAMETERS BY REGION

Description

DRVDTR defines the maximum rate at which the solution oil-gas ratio or condensate-gas ratio (Rv) can be increased in a grid cell for various regions in the model. The keyword is similar in functionality to the DRVDT keyword, that defines the maximum rate at which Rv can be increased in a grid cell for all cells in the model. The number of DRVDTR vector data sets is defined by the NTPVT parameter on the TABDIMS keyword in the RUNSPEC section and the allocation of the DRVDTR records to different grid blocks in the model is done via the PVTNUM keyword in the REGION section. One data set consists of one record or line which is terminated by a "/".

This keyword should only be used if the OIL, GAS, and VAPOIL (condensate) keywords in the RUNSPEC section have been invoked to allow oil, gas and condensate to be present in the model. The keyword only affects the behavior of an increasing Rv, for example when gas is being injected into a gas condensate reservoir as part of as gas re-cycling scheme, and is subject to the availability of free oil (condensate) and the ability of the undersaturated gas to adsorb this condensate.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	DRVDT1	DRVDT1 is a real positive number that defines the maximum rate at which the solution oil-gas ratio is allowed to increase in a grid cell, that is the maximum rate at which the oil can dissolve into the available undersaturated gas. A value of zero means that Rv cannot increase and free oil cannot dissolve into the unsaturated gas in a grid cell. Alternatively a very large value of DRVDT1 allows Rv to increase rapidly until there is no free oil or the gas within the grid block is fully saturated. Note if the keyword is not present in the input deck then DRVDT1 is assumed to be a very large number resulting in complete re-solution of the oil into the available undersaturated gas.			
		stb/Mscf/d	sm^3/sm^3/day	scc/scc/day	None

Notes:

1) The keyword is followed by NTPVT records as declared on the TABDIMS keyword in the RUNSPEC section.
2) Each record is terminated by a "/" and there is no "/" terminator for the keyword.

Table 12.26: DRVDTR Keyword Description

Note this keyword can be used in history matching field performance to control the availability of the movable gas phase.

See also the VAPPARS keyword in the SOLUTION section and DRVD, DRSDT, and DRSDTR keywords in the SCHEDULE section that controls how vaporized oil is treated and the rate at which the dissolved phase ratio increases within a grid block.

Examples

The first example prevents the solution oil-gas ratio from increasing and applies this to all regions for when NTPVT on the TABDIMS keyword in the RUNSPEC section is set to three.

```
--
--       SOLUTION OIL (RV) MAXIMUM RATE OF INCREASE PARAMETERS BY REGION
--
DRVDTR
--       MAX RV
--       DRVDT1
--       -------          -
         0.000                                          /
         0.000                                          /
         0.000                                          /
```

The second example below prevents the solution oil-gas ratio from increasing and applies this to all grid cells in PVTNUM region one. For PVTNUM regions one and two the keyword applies 0.005 stb//Mscf/d as the maximum rate at which the solution oil-gas ratio is allowed to increase in a grid cell,

```
--
--       SOLUTION OIL (RV) MAXIMUM RATE OF INCREASE PARAMETERS BY REGION
--
DRVDTR
--       MAX RV
--       DRVDT1
--       -------
         0.0000                                         /
         0.0005                                         /
         0.0005                                         /
```

Again, the keyword parameters when applied are subject to the availability of free oil and the ability of the undersaturated gas to adsorb this oil.

12.3.58 DUMPCUPL – ACTIVATE OUTPUT TO THE RESERVOIR COUPLING FILE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, DUMPCUPL, activates output to the Reservoir Coupling file from the reservoir coupling file in the master run for when reservoir coupling is invoked by the GRUPMAST and SLAVES keywords in the SCHEDULE section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.59 DYNAMICR – DEFINE DYNAMIC REGION PARAMETERS

The DYNAMICR keyword marks the start of a Dynamic Region section and defines the parameters used for Dynamic Regions that allows for property and reporting regions to vary as the run progresses, based on the parameters and logic defined by this keyword and section. A Dynamic Region section is terminated by the ENDDYN keyword in the SOLUTION or SCHEDULE sections.

See *DYNAMICR – Start of Dynamic Region Parameter Definition* in the SOLUTION section for a full description.

12.3.60 ECHO – ACTIVATE ECHOING OF USER INPUT FILES TO THE PRINT FILE

Turns on echoing of all the input files to the print file; note that this keyword is activated by default and can subsequently be switched off by the NOECHO activation keyword.

See *ECHO – Activate Echoing of User Input Files to the Print File* in the GLOBAL section for a full description.

12.3.61 END – DEFINE THE END OF THE INPUT FILE

This keyword marks the end of the input file and can occur in any section. Any keywords and data after the END keyword are ignored.

See *END – Define the End of the Input File* in the GLOBAL section for a full description.

12.3.62 ENDACTIO – END THE DEFINITION OF ACTION COMMANDS

Description

The ENDACTIO keyword defines the end of a series of conditions that invoke run time processing of the ACTION series of keywords, namely: ACTION, ACTIONG, ACTIONR, ACTIONS, ACTIONW and ACTIONX. Only the ACTIONX keyword is implemented in OPM Flow as this keyword implements the ACTION, ACTIONG, ACTIONR, ACTIONS, ACTIONW functionality with greater flexibility. See the ACTIONX keyword in the SCHEDULE section for a full description of the ACTION facility.

There is no data required for this keyword and there is no terminating "/" for this keyword.

Example

The example shows the use of the ACTIONX and ENDACTIO keywords to test if the field's gas production rate is less than 600 MMscf/d after 2020 and to open up additional wells if this occurs.

```
--
-- START OF ACTIONX FIELD PHASE-2 DEVELOPMENT DEFINITION
--
ACTIONX
     PHASE2       1               /
     GGPR  'FIELD' < 600E3 AND    /
     YEAR > 2020                  /
/
-- WELL PRODUCTION STATUS
--
--  WELL    WELL    --LOCATION--  COMPLETION
--  NAME    STAT      I   J    K  FIRST LAST
WELOPEN
GP10        OPEN                           /
GP11        OPEN                           /
/
--
-- END OF ACTIONX FIELD PHASE-2 DEVELOPMENT DEFINITION
--
ENDACTIO
```

12.3.63 ENDBOX – DEFINE THE END OF THE BOX DEFINED GRID

This keyword marks the end of a previously defined BOX sub-grid as defined by a previously entered BOX keyword. The keyword resets the input grid to be the full grid as defined by the NX, NY, and NZ variables on the DIMENS keyword in the RUNSPEC section.

See *ENDBOX – Define the End of the BOX Defined Grid* in the GRID section for a full description.

12.3.64 ENDDYN– END OF DYNAMIC REGION PARAMETER DEFINITION

The ENDDYN keyword marks the end of a Dynamic Region section that was started with the DYNAMICR keyword in the SOLUTION or SCHEDULE sections. Dynamic Regions allow for property and reporting regions to vary as the run progresses, based on the parameters and logic defined within the section.

See *ENDDYN– End of Dynamic Region Parameter Definition* in the SOLUTION for a full description.

12.3.65 ENDFIN – END THE DEFINITION OF A LOCAL GRID REFINEMENT

ENDFIN defines the end of a Cartesian or radial local grid refinement ("LGR") definition and a LGR property definition data set.

See *ENDFIN – End the Definition of a Local Grid Refinement* in the GRID section for a full description.

12.3.66 ENDINC – DEFINE THE END OF AN INCLUDE FILE

This keyword marks the end of an include file specified on the INCLUDE keyword. When the ENDINC keyword is encountered in the INCLUDE file, input data is read from the next keyword in the current file. Any keywords and data after the ENDINC keyword in the INCLUDE file are ignored.

See *ENDINC – Define the End of an Include File* in the GLOBAL section for a full description.

12.3.67 ENDSKIP – DEACTIVATE SKIPPING OF KEYWORDS AND INPUT DATA

Turns off skipping of keywords that was activated by the SKIP, SKIP100, or SKIP300 keywords. Note that for each SKIP keyword activated there must be a corresponding ENDSKIP keyword.

See *ENDSKIP – DeActivate Skipping of Keywords and Input Data* in the GLOBAL section for a full description.

12.3.68 EPSDBGS – WRITE END-POINT DEBUG DATA TO THE DEBUG FILE (MULTIPLE)

This keyword, EPSDBGS, defines the end-point debug data for multiple grid blocks that should be written to the DEBUG file (*.DBG) for when the End-Point Scaling option has been activated by the ENDSCALE keyword in the RONSPEC section.

See *EPSDBGS - Write End-Point Debug Data to the DEBUG File (Multiple)* in the PROPS section for afull description.

12.3.69 EPSDEBUG - WRITE END-POINT DEBUG DATA TO THE DEBUG FILE (INDIVIDUAL)

This keyword, EPSDEBUG, defines the end-point debug data for individual grid blocks that should be written to the DEBUG file (*.DBG) for when the End-Point Scaling option has been activated by the ENDSCALE keyword in the RONSPEC section.

See *EPSDEBUG - Write End-Point Debug Data to the DEBUG File (Individual)*n the PROPS section for afull description.

12.3.70 EXCAVATE - SET THE STATUS OF A GRID BLOCK TO ACTIVE OR EXCAVATE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, EXCAVATE, sets the status of global and LGR grid blocks to active or excavate. Excavated grid blocks have all the transmissibilities set to zero thus disabling flow between the surrounding grid blocks.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.71 EXIT – EXIT SIMULATION FROM WITHIN AN ACTION SECTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The EXIT keyword is part of OPM Flow's ACTION facility that allows for terminating the simulation for when a condition within an ACTIONX definition is satisfied. Invoking the keyword within an ACTIONX definition will result in the simulation terminating with an exit status code. The ACTION facility allows the user to enter computational logic to the simulation run based on the how the simulation run is proceeding – see the ACTIONX keyword in the SCHEDULE section.

Note

This is an OPM Flow specific keyword for the simulator's ACTION facility and will therefore cause an error if used in the commercial simulator.

No.	Name	Description	Default
1	EXITCODE	An optional integer that sets the exit code printed to the *.PRT file, if not not defined the default value of zero will be used.	0

Notes:

1) The keyword is terminated by a "/".

Table 12.27: EXIT Keyword Description

The EXIT keyword should only be used as part of an ACTIONX block, if found elsewhere in the input deck it will be ignored.

Examples

The first example uses the ACTIONX keyword to define a condition for when the Field Oil Production Rate ("FOPR) falls below 1,000 stb/d (or 1,000 m^3) using the default value for the EXITCODE.

```
--
--       DEFINE START OF ACTIONX SECTION
--
ACTIONX
         'CHECK_FOPR' 100000            /
         FOPR < 1000                    /
/
--
--       TERMINATE AND EXIT SIMULATION
--
EXIT
                                        /

ENDACTIO
```

The next example terminates the simulation with EXITCODE one when the Field Pressure ("FPR") falls below 200 psia (or 200 barsa).

```
--
--       DEFINE START OF ACTIONX SECTION
--
ACTIONX
         'CHECK_FPR'  100000             /
          FPR < 200                      /
/
--
--       TERMINATE AND EXIT SIMULATION
--
EXIT
         1                              /

ENDACTIO
```

Note is is probably good practice to always set the EXITCODE to be able to identify the reason for the simulation stopping.

12.3.72 EXTRAPMS – ACTIVATE EXTRAPOLATION WARNING MESSAGES

The EXTRAPMS keyword activates extrapolation warning messages for when OPM Flow extrapolates the PVT or VFP tables. Frequent extrapolation warning messages should be investigated and resolved as this would indicate possible incorrect data and may result in the simulator extrapolating to unrealistic values.

See *EXTRAPMS – Activate Extrapolation Warning Messages* in the GLOBAL section for a full description.

12.3.73 FBHPDEF – DEFINE WELL DEFAULT BHP TARGET AND CONSTRAINTS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, FBHPDEF, defines the default well BHP target for production wells and the default BHP constraint for injection wells.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

12.3.74 FILEUNIT – ACTIVATE UNIT CONSISTENCY VERIFICATION

The FILEUNIT keyword defines the units of the of the data set, and is used to verify that the units in the input deck and any associated include files are consistent grid data. The keyword does not provide for the conversion between different sets of units.

See *FILEUNIT – Activate Unit Consistency Checking* in the GRID section for a full description.

12.3.75 FORMFEED – DEFINED THE PRINT FILE FORM-FEED CHARACTER

The FORMFEED keyword defines the form-feed character, or carriage control character, for the output print (*.PRT) run summary (*.RSM) files. The keyword should be place at the very top of the input file.

See *FORMFEED – Defined the Print File Form-Feed Character* in the GLOBAL section for a dull description.

12.3.76 GASBEGIN – DEFINE START OF ANNUAL SCHEDULING SECTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, GASBEGIN, defines the start of an Annual Scheduling section set of keywords used when the Gas Field Operations option has been activated by the GASFIELD keyword in the RUNSPEC section. An Annual Scheduling section starts with the GASBEGIN keyword and is terminated by the GASEND keyword, with keywords in between used to control and write reports at selected times between the start and end of a contract period. Only one Annual Scheduling section is activate at a time, that is, a subsequent Annual Scheduling section overwrites the previous set of entries. To clear the current Annual Schedule section enter the GASBEGIN keyword followed by the GASEND keyword word with no other keywords in between.

There is no data required for this keyword and there is no terminating "/" for this keyword.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.77 GASEND – DEFINE END OF ANNUAL SCHEDULING SECTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, GASEND, defines the end of an Annual Scheduling section set of keywords used when the Gas Field Operations option has been activated by the GASFIELD keyword in the RUNSPEC section. An Annual Scheduling section starts with the GASBEGIN keyword and is terminated by the GASEND keyword, with keywords in between used to control and write reports at selected times between the start and end of a contract period. Only one Annual Scheduling section is activate at a time, that is, a subsequent Annual Scheduling section overwrites the previous set of entries. To clear the current Annual Schedule section enter the GASBEGIN keyword followed by the GASEND keyword word with no other keywords in between.

There is no data required for this keyword and there is no terminating "/" for this keyword.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.78 GASFCOMP – DEFINE AUTOMATIC GAS COMPRESSORS

Description

This keyword, GASFCOMP, defines automatic gas compressors for when the Gas Field Operations option has been activated by the GASFIELD keyword in the RUNSPEC section and the Standard Network option has been specified by the GRUPTREE, GRUPNET and GNETINJE series of keywords in the SCHEDULE section. Automatic gas compressors are automatically switch on for a group if a group's gas production target cannot be satisfied. In addition, if a group's gas target is reduced then the automatic compressors are initially switch off to test that the reduced gas rate target can be met without compression, if not, compression is switched back on. Note that all automatic compressors are "switch on" when calculating a field's gas deliverability.

There is no data required for this keyword and there is no terminating "/" for this keyword.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.79 GASFDECR – DEFINE FIELD GAS SALES CONTRACT MONTHLY REDUCTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, GASFDECR, defines the field's monthly reduction in the gas sales contract quantity for when the Gas Field Operations option has been activated by the GASFIELD keyword in the RUNSPEC section. Gas contracts are commonly based on a Daily Contract Quantity ("DCQ") that determines the gas rate that the field should be produced at, which is normally expressed as a multiple of the DCQ, for example 1.33, and is often referred to as the "swing factor". Some gas contracts also define a maximum DCQ ("Max DCQ") and/or a minimum take or pay DCQ ("Min DCQ"), as well as seasonal demand characteristics. For example, gas rates may be set higher in the winter months in order to meet heating demand compared with summer months in colder climates, and the opposite in warmer climates where air conditioning demand is high.

Thus, the DCQ must be calculated first over a contract period, where the contract period is commonly contract years, or in some instances contract quarters. This is performed by the simulator using the current DCQ and checking to see if the (DCQ x Swing Factor) can be satisfied throughout the current contract period, if not the DCQ is re-calculated so that the (DCQ x Swing Factor) condition is satisfied. Once this condition is met, the second and final pass uses the calculated DCQ in conjunction with the monthly scaling profile data to set the monthly gas rate for the field:

$$Q_{month} = DCQ \times SWINGFAC_{month} \tag{12.19}$$

Where:

Qmonth	=	the monthly gas production target
DCQ	=	Daily Contract Quantity
$SWINGFAC_{month}$	=	monthly rate scaling factor that takes into account seasonal demand, etc.

Here the GASFDECR keyword allows for a reduction in the calculated final pass monthly gas rates and thus equation (12.19) become:

$$Q_{month} = \left(DCQ \times SWINGFAC_{month} \right) - GASFDECR_{month} \tag{12.20}$$

Where:

Qmonth	=	the monthly gas production target
DCQ	=	Daily Contract Quantity
$SWINGFAC_{month}$	=	monthly rate scaling factor that takes into account seasonal demand, etc.
$GASFDECR_{month}$	=	monthly gas rate reduction.

Since the simulator must make two passes to calculate the final rates this will naturally decrease computational efficiency.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.80 GASFDELC – DEFINE GAS DELIVERABILITY CALCULATION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The GASFDELC keyword defines how the field's gas deliverability calculation should be performed for when the Gas Field Operations option has been activated by the GASFIELD keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.81 GASFTARG – DEFINE FIELD GAS SALES CONTRACT MONTHLY TARGET

Description

This keyword, GASFTARG, defines the field's monthly gas sales contract quantity for when the Gas Field Operations option has been activated by the GASFIELD keyword in the RUNSPEC section. Gas contracts are commonly based on a Daily Contract Quantity ("DCQ") that determines the gas rate that the field should be produced at, which is normally expressed as a multiple of the DCQ, for example 1.33, and is often referred to as the "swing factor". Some gas contracts also define a maximum DCQ ("Max DCQ") and/or a minimum take or pay DCQ ("Min DCQ"), as well as seasonal demand characteristics. For example, gas rates may be set higher in the winter months in order to meet heating demand compared with summer months in colder climates, and the opposite in warmer climates where air conditioning demand is high.

Thus, the DCQ must be calculated first over a contract period, where the contract period is commonly contract years, or in some instances contract quarters. This is performed by the simulator using the current DCQ and checking to see if the (DCQ x Swing Factor) can be satisfied throughout the current contract period, if not the DCQ is re-calculated so that the (DCQ x Swing Factor) condition is satisfied. Once this condition is met, the second and final pass uses the calculated DCQ in conjunction with the monthly scaling profile data to set the monthly gas rate for the field:

$$Q_{month} = DCQ \times SWINGFAC_{month} \tag{12.21}$$

Where:

Qmonth = the monthly gas production target
DCQ = Daily Contract Quantity
$SWINGFAC_{month}$ = monthly rate scaling factor that takes into account seasonal demand, etc.

Here the GASFTARG keyword sets a minimum target rate in the calculated final pass monthly gas rates and thus equation (12.21) become:

$$Q_{month} = Minimum\left(\left(DCQ \times SWINGFAC_{month}\right), GASFTARG_{month}\right) \tag{12.22}$$

Where:

Qmonth = the monthly gas production target
DCQ = Daily Contract Quantity
$SWINGFAC_{month}$ = monthly rate scaling factor that takes into account seasonal demand, etc.
$GASFTARG_{month}$ = minimum monthly gas rate target.

Since the simulator must make two passes to calculate the final rates this will naturally decrease computational efficiency.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.82 GASMONTH – DEFINE START OF ANNUAL SCHEDULING EVENT

Description

This keyword, GASMONTH, states the month for which subsequent scheduling events take place within an Annual Schedule section for when the Gas Field Operations option has been activated by the GASFIELD keyword in the RUNSPEC section. The keyword must lie in between the GASBEGIN, that defines the start of an Annual Scheduling section and the GASEND keyword that ends the section. Optionally, the keyword can be used to write a report to the print file (*.PRT) at the requested month.

See also the GASBEGIN and GASEND keywords in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.83 GASPERIO – ADVANCE SIMULATION BY GAS CONTRACT PERIOD

Description

This keyword advances the simulation over one or more gas contract periods for when the Gas Field Operations option has been activated by the GASFIELD keyword in the RUNSPEC section. A contract period in this case is the period over which the Daily Contract Quantity is fixed, this can a be year or one or more months. If the contract period is a year then the GASYEAR keyword in the SCHEDULE section can be used instead of GASPERIOD.

GASPERIO is an alternative to the DATES, TIME and TSTEP keywords in the SCHEDULE section that advances the simulation to a given report time after which additional keywords may be entered to instruct OPM Flow to perform additional functions via the SCHEDULE section keywords.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.84 GASYEAR – ADVANCE SIMULATION BY GAS CONTRACT YEAR

Description

This keyword advances the simulation over one or more gas contract years for when the Gas Field Operations option has been activated by the GASFIELD keyword in the RUNSPEC section. A contract year in this case is the period over which the Daily Contract Quantity is fixed, this can a be year, this keyword or the GASPERIO keyword in the SCHEDULE section, or one or more months. If the contract period is over one or more months then the GASPERIO keyword in the SCHEDULE section can be used instead of GASYEAR.

GASYEAR is an alternative to the DATES, TIME and TSTEP keywords in the SCHEDULE section that advances the simulation to a given report time after which additional keywords may be entered to instruct OPM Flow to perform additional functions via the SCHEDULE section keywords.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.85 GCALECON – GROUP ECONOMIC CRITERIA FOR PRODUCTION CALORIFIC GROUPS

Description

The GCALECON keyword defines economic criteria for production groups, including the field level group FIELD, that have previously been defined by the GCONPROD keyword in the SCHEDULE section and have had their rate targets and constraints set by calorific value via the GCONVAL keyword in the SCHEDULE section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

Note that wells are allocated to a group when they are specified by the WELSPECS keyword and wells can also have economic controls. Wells under group control are therefore subject to the economic criteria set via the GCONPROD and CECON keywords in the SCHEDULE section and the controls specified by the WECON keyword.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	GRPNAME	A character string of up to eight characters in length that defines the group name for which the group target and constraints are being defined. The group named FIELD is the top most group and should be used to set targets and constraints for the field. Note that the group hierarchy should be defined by the GRUPTREE keyword when there is more than one level of groups, otherwise all the groups will sit directly under the FIELD group in the group tree hierarchy.			None
2	ENEVAL	A real positive value that defines the minimum economic surface energy production rate, below which an economic action of shutting in or stopping all the wells in the group, as requested by item (9) of the WELSPECS keyword. A value less than or equal to zero switches of this criteria.			
		BTU/day	kJ/day	J/hour	0.0
3	CALVAL	A real positive value that defines the minimum economic surface calorific value, below which an economic action of shutting in or stopping all the wells in the group, as requested by item (9) of the WELSPECS keyword. A value less than or equal to zero switches of this criteria,			
		Btu/Mscf	kJ/sm^3	J/hour	0.0
8	END	A defined character string that defines if the simulation should terminate if all the producing wells in the group, including the FIELD group, are shut or stopped. END should be set to one of the following character strings: 1) NO: no action is taken and the run continues. 2) YES: terminate the run at the next report time step.			NO

No.	Name	Description			Default
		Field	Metric	Laboratory	
Notes: 1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".					

Table 12.28: GCALECON Keyword Description

See also the WELSPECS keyword to define a wells shut-in or stop options, GCONPROD, GCONCAL, GCONENG for group controls, and WECON for setting a well's economic criteria. All the aforementioned keywords are described in the SCHEDULE section.

Example

The following example defines the economic criteria for the field with a minimum economic surface energy production rate of 5 x 109 BTU/day and a minimum economic surface calorific value of900 Btu/Mscf

```
--
--       GROUP ECONOMIC CRITERIA FOR PRODUCTION GROUPS UNDER CALORIFIC CONTROL
--
-- GRUP  ENERGY  CALORIFIC  END
-- NAME  RATE    VALUE      RUN
GCALECON
FIELD    5E9     900.0     'YES'                                              /
/
```

If the economic limits are violated then the run will stop at the next report time step.

12.3.86 GCONCAL – GROUP PRODUCTION CALORIFIC TARGETS

Description

The GCONCAL keyword defines calorific production targets and constraints for groups, including the top most group in the group hierarchy known as the FIELD group. Wells are allocated to groups when the wells are specified by the WELSPECS keyword in the SCHEDULE section. Wells defined to be under group control will have their production rates controlled by the group to which they belong, in addition to any well constraints defined for the wells.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	GRPNAME	A character string of up to eight characters in length that defines the group name for which the group's target calorific value is being defined. The group named FIELD is the top most group and should be used to set targets and constraints for the field. Note that the group hierarchy should be defined by the GRUPTREE keyword when there is more than one level of groups, otherwise all the groups will sit directly under the FIELD group in the group tree hierarchy.			None
2	CALVAL	A real positive value that defines the target surface calorific value of the produced gas. The default value of 1 x 1020 switches off calorific control for the group.			
		Btu/Mscf	kJ/sm^3	J/hour	1.0 x 10^{20}
3	ACTION	A defined character string that defines the action to be taken if the CALVAL is violated. ACTION should be set to one of the following character strings: 1) NONE: no action is taken and the group's produced calorific value will therefore no longer meet CALVAL. 2) RATE: scale back the gas rate of various wells under the group by the value of FACTOR until the calorific target (CALVAL) is satisfied. The corrective action takes places at the end of the time step in which the constraint is violated.			None
4	FACTOR	A real positive value that is less than or equal to one that defines the amount wells can be scaled back in order to satisfy CALVAL. Note this assumes that there are wells within the group that are producing with higher and lower calorific values, and the simulator is thus able to fine a combination of wells that satisfy the group's CALVAL target.			
		dimensionless	dimensionless	dimensionless	None

Notes:

1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".

Table 12.29: GCONCAL Keyword Description

See also the GRUPTREE keyword to define the hierarchy of the groups below the FIELD level, the

GCONPROD and GCONINJE keywords to define a group's production and injection rate targets and constraints, the WCONPROD keyword to define a production well's targets and constraints, and the WCONINJE keyword to define an injection well's targets and constraints. All the aforementioned keywords are described in the SCHEDULE section.

Example

The following example defines the calorific production target for the field.

```
---
--       GROUP CALORIFIC PRODUCTION CONTROLS
--
-- GRUP  CALORIFIC  ACTION  CUT
-- NAME  VALUE              BACK
GCONCAL
FIELD    1010E3     RATE    0.95                                    /
/
```

Here the calorific production target has been set to 1,010 x 103 Btu/Mscf for the field and if the target cannot be met then the well rates are reduced by 0.95 at each iteration until the target is satisfied.

12.3.87 GCONENG – GROUP PRODUCTION ENERGY TARGETS

Description

The GCONCAL keyword defines energy production targets and constraints for groups, including the top most group in the group hierarchy known as the FIELD group. Wells are allocated to groups when the wells are specified by the WELSPECS keyword in the SCHEDULE section. Wells defined to be under group control will have their production rates controlled by the group to which they belong, in addition to any well constraints defined for the wells.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	GRPNAME	A character string of up to eight characters in length that defines the group name for which the group's target calorific value is being defined. The group named FIELD is the top most group and should be used to set targets and constraints for the field. Note that the group hierarchy should be defined by the GRUPTREE keyword when there is more than one level of groups, otherwise all the groups will sit directly under the FIELD group in the group tree hierarchy.			None
2	ENGVAL	A real positive value that defines the surface energy target for the group. The default value of 1 x 1020 switches off the energy target for the group.			
		BTU/day	kJ/day	J/hour	1.0×10^{20}
Notes: 1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".					

Table 12.30: GCONENG Keyword Description

See also the GRUPTREE keyword to define the hierarchy of the groups below the FIELD level, the GCONPROD and GCONINJE keywords to define a group's production and injection rate targets and constraints, the WCONPROD keyword to define a production well's targets and constraints, and the WCONINJE keyword to define an injection well's targets and constraints. All the aforementioned keywords are described in the SCHEDULE section.

Example

The following example defines the energy production target for the field.

```
--
--       GROUP ENERGY PRODUCTION CONTROLS
--
-- GRUP  ENERGY
-- NAME  VALUE
GCONENG
FIELD    1010E9                                                    /
/
```

Here the energy production target has been set to 1,010 x 109 Btu/day for the field.

12.3.88 GCONINJE – GROUP INJECTION TARGETS AND CONSTRAINTS

Description

The GCONINJE keyword defines injection targets and constraints for groups, including the top most group in the group hierarchy known as the FIELD group. Wells are allocated to groups when the wells are specified by the WELSPECS keyword in the SCHEDULE section. Wells defined to be under group control will have their injection rates controlled by the group to which they belong, in addition to any well constraints defined for the wells.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	GRPNAME	A character string of up to eight characters in length that defines the group name for which the group target and constraints are being defined. The group named FIELD is the top most group and should be used to set targets and constraints for the whole field. Note that the group hierarchy should be defined by the GRUPTREE keyword in the SCHEDULE, when there is more than one level of groups, otherwise all the groups will sit directly under the FIELD group in the group tree hierarchy.			None
2	TYPE	A defined character string that defines the type of injection fluid. TYPE should be set to one of the following character strings: 1) GAS: for a gas injection well. 2) OIL: for a water injection well. 3) WAT: for a water injection well.			None
3	TARGET	A defined character string that sets the target injection control for the group, all the other phases will therefore act as constraints. The simulator will attempt to meet the TARGET based on the phase rate stated in items (4) to (7) on this keyword. TARGET should be set to one of the following character strings: 1) NONE: the group has no target phase, but if entered, constraints are still defined and active. 2) FLD: this group is controlled from a higher level group, including the FIELD group. 3) RATE: the injection phase will be control by the surface fluid rate for the phase defined by the TYPE variable. For example, if TYPE has been set to WAT then this would mean the water injection rate as defined by item (4). 4) RESV: the target is set to the in situ reservoir volume rate as defined by item (5). 5) REIN: the target is set to the group's production of the phase defined by TYPE multiplied by the value on item (6). For example, if TYPE has been set to WAT then this would mean the group's water production multiplied by item (6). 6) VREP: the target is set to the group's voidage replacement ratio as defined by item (7).			None

No.	Name	Description			Default
		Field	Metric	Laboratory	
4	RATE	A real positive value that defines the maximum surface injection rate target or constraint for the phase declared by the TYPE variable.			
		Liquid stb/d Gas Mscf/d	Liquid sm^3/day Gas sm3/day	Liquid scc/hour Gas scc/hour	None
5	RESV	A real positive value that defines the maximum reservoir volume injection rate target or constraint. Note setting a value here other than the default means that TYPE, item (2) will be the supplement or "make up" phase.			
		rtb/d	rm^3/day	rcc/hour	None
6	REIN	A real positive value that defines the target or constraint re-injection fraction for the produced phase defined by the TYPE variable. For example, if TYPE is equal to GAS and REIN is equal to 0.85, then 85% of the produced gas will be re-injected.			
		dimensionless	dimensionless	dimensionless	None
7	VREP	A real positive value that defines the target or constraint of the voidage replacement ratio based on all the produced fluids. For example, if TYPE is equal to WAT and VREP is equal to 1.00, then 100% of the produced reservoir volume will be re-inject as an equivalent water volume. Note setting a value here other than the default means that TYPE, item (2) will be the supplement phase.			
		dimensionless	dimensionless	dimensionless	None
8	GRPCNTL	A defined character string that determines if this group is subject to higher level group control. 1) YES: then this group is subject to a higher level group's control and the flow rates for this group will be adjusted accordingly. 2) NO: then this group is NOT subject to a higher level group's control and the flow rates for this group will only be controlled by the parameters for this group. This variable is ignored if GRPNAME is equal to FIELD.			YES
9	GRPGUIDE	A real positive value that defines a group's injection guide rate expressed as a dimensionless number. A group requires a value for GRPGUIDE only if it is required to produce a specified proportion of a higher level group's rate. Defaulting GRPGUIDE results in the subordinate groups and wells under guide control having their rates dictated by any higher level groups under guide rate control. In other words the GRPNAME is masked out. Setting GRPGUIDE to a real positive value and GUIPHASE to either RATE or RESV will result in a constant injection guide rate.			
		dimensionless	dimensionless	dimensionless	None

No.	Name	Description			Default
		Field	Metric	Laboratory	
10	GUIPHASE	A defined character string that sets the guide phase to which the guide rate in item (9) applies. GUIPHASE should be set to one of the following character strings: 1) NETV: the guide phase is set to production rate minus any injected reservoir rate from the other phases. The net volume is calculated at the beginning of a time step and the option should only be used for groups under voidage replacement control. Note that using this option means that TYPE, item (2) will be the supplement or "make up" phase and the value entered for the group's guide rate (GRPGUIDE) will be ignored with this option. 2) RATE: the guide phase is set to the surface injection rate. Setting GUIPHASE to RATE and GRPGUIDE to a real positive value will result in a constant injection guide rate. 3) RESV: the guide phase is set to the in situ reservoir volume rate. Setting GUIPHASE to RESV and GRPGUIDE to a real positive value will result in a constant injection guide rate. 4) VOID: the guide rate is calculated at the beginning of each time step based on the group's net voidage rate. OPM Flow now supports all guide rate options. The default value of 1* means that the group has no injection guide rate for this phase.			None
11	GRPREIN	A character string of up to eight characters in length that defines the group name whose production rate should be used for applying the REIN quantity to be injected into GRPNAME. This variable is used to re-inject the REIN production faction from another group (GRPREIN) via this group (GRPNAME). If GRPREIN is defaulted then the re-injection quantity for GRPNAME will be based on the production from GRPNAME itself.			GRPNAME
12	GRPVREP	A character string of up to eight characters in length that defines the group name whose production rate should be used for applying the VREP quantity to be injected into GRPNAME. This variable is used to re-inject the VREP production faction from another group (GRPVREP) via this group (GRPNAME). If GRPVREP is defaulted then the voidage quantity for GRPNAME will be based on the production from GRPNAME itself.			GRPNAME
13	WGASRATE	Wet gas injection rate used in the commercial compositional simulator. Not used and should be defaulted with 1*.			1*

Notes:

1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".

Table 12.31: GCONINJE Keyword Description

See also the GRUPTREE keyword to define the hierarchy of the groups below the FIELD level, the GCONPROD keyword to define a group's production targets and constraints, the WCONPROD keyword to define a production well's targets and constraints, and the WCONINJE keyword to define an injection well's targets and constraints. All the aforementioned keywords are described in the SCHEDULE section.

Example

The following example defines the injection targets and constraints for the field and two groups that are one level below the field group, since the GRUPTREE keyword has not been entered to define the group hierarchy.

```
--
--        GROUP INJECTION TARGETS AND CONSTRAINTS
--
-- GRUP   FLUID CNTL    SURF    RESV    REINJ   VOID   GRUP   GUIDE   GUIDE GRUP   GRUP
-- NAME   TYPE  MODE    RATE    RATE    FRAC    FRAC   CNTL   RATE    DEF   REINJ RESV
GCONINJE
FIELD     WAT   VREP    35E3    1*      1*      1*      NO    1*      1*    1*    1*    /
GRP01     WAT   VREP    1*      1*      1*      1.0     YES   1*      1*    1*    1*    /
GRP02     WAT   VREP    1*      1*      1*      1.0     YES   1*      1*    1*    1*    /
/
```

In this example, group GRP01 and GRP02 are injecting water via voidage replacement with a voidage replacement of one and are under the control on the field group, that imposes a 35,000 m3/day total water injection limit.

12.3.89 GCONPRI – GROUP PRODUCTION PRIORITY TARGETS AND CONSTRAINTS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The GCONPRI keyword defines production targets and constraints for groups, including the top most group in the group hierarchy known as the FIELD group, for when groups and their associated wells are operating under priority control as oppose to guide rate control. Priority control is activated by the PRIORITY keyword in the SCHEDULE section. Wells are allocated to groups when the wells are specified by the WELSPECS keyword in the SCHEDULE section. Wells defined to be under group control will have their production rates controlled by the group to which they belong, in addition to any well constraints defined for the wells.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	GRPNAME	A character string of up to eight characters in length that defines the group name for which the group target and constraints are being defined. The group named FIELD is the top most group and should be used to set targets and constraints for the field. Note that the group hierarchy should be defined by the GRUPTREE keyword when there is more than one level of groups, otherwise all the groups will sit directly under the FIELD group in the group tree hierarchy.			None
2	ORAT	A real positive value that defines the maximum surface oil production rate constraint.			
		stb/d	sm^3/day	scc/hour	None

No.	Name	Description			Default
		Field	Metric	Laboratory	
3	OILACT	A defined character string that defines the action to be taken if ORAT is exceeded. OILACT should be set to one of the following character strings: 1) NONE: no action is taken. 2) CON: close the worst offending connection in the worst offending well. If connections have been grouped as completions then the worst offending completion in the worst offending well will be closed. 3) +CON: close the worst offending connection and all below it in the worst offending well. If connections have been grouped as completions then the worst offending completion and all below it in the worst offending well will be closed. 4) WELL: close the worst offending well. 5) PLUG: plug back the worst offending well based on the plug back length and options defined on the WPLUG keyword in the SCHEDULE. 6) PRI: control the group production rate using the first priority formulae defined by the PRIORITY keyword in the SCHEDULE section. 7) PR2: control the group production rate using the second priority formulae defined by the PRIORITY keyword in the SCHEDULE section.			"NONE"
4	WRAT	A real positive value that defines the maximum surface water production rate constraint.			
		stb/d	sm^3/day	scc/hour	None
5	WATACT	A defined character string that defines the action to be taken if WRAT is exceeded. WATACT should be set to a character string described by the OILACT parameter on this record.			None
6	GRAT	A real positive value that defines the maximum surface gas production rate constraint			
		Mscf/d	sm^3/day	scc/hour	None
7	GASACT	A defined character string that defines the action to be taken if GRAT is exceeded. GASACT should be set to a character string described by the OILACT parameter on this record.			None
8	LRAT	A real positive value that defines the maximum surface liquid (oil plus water) production rate constraint.			
		stb/d	sm^3/day	scc/hour	None
9	LIQRAT	A defined character string that defines the action to be taken if LRAT is exceeded. LIQACT should be set to a character string described by the OILACT parameter on this record.			None
10	RESV	A real positive value that defines the maximum reservoir volume production rate constraint.			
		rb/d	rm^3/day	rcc/hour	None

No.	Name	Description			Default
		Field	Metric	Laboratory	
11	RESVFRAC	A real positive value that defines a group's maximum production balancing fraction constraint.			
		dimensionless	dimensionless	dimensionless	None
12		Not used should be defaulted with 1*.			1*
13		Not used should be defaulted with 1*.			1*
14		Not used should be defaulted with 1*.			1*
15		Not used should be defaulted with 1*.			1*
16	LINCOMB	A real positive value that defines the linearly combined maximum surface target rate or constraint, as per the LINCOM keyword in the SCHEDULE section.			1*
17	LINACT	A defined character string that defines the action to be taken if LINCOMB is exceeded. LINACT should be set to a character string described by the OILACT parameter on this record.			None

Notes:

1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".

Table 12.32: GCONPRI Keyword Description

See also the GRUPTREE keyword to define the hierarchy of the groups below the FIELD level, the GCONPROD keyword to define a group's production targets and constraints, the GCONINJE keyword to define a group's injection targets and constraints, the WCONPROD keyword to define a production well's targets and constraints, and the WCONINJE keyword to define an injection well's targets and constraints. All the aforementioned keywords are described in the SCHEDULE section.

Example

The following example defines the production targets and constraints for the field and two groups that are one level below the field group, since the GRUPTREE keyword has not been entered to define the group hierarchy.

```
--
--        GROUP PRIORITY PRODUCTION CONTROLS
--
-- GRUP   ORAT   ORAT   WRAT   WRAT   GRAT   GRAT   LRAT   LRAT   RVOL   RVOL
-- NAME   LIMT   ACTN   LIMT   ACTN   LIMT   ACTN   LIMT   ACTN   LIMT   ACTN
GCONPRI
FIELD     40E3   PRI    30E3   +CON   125E3  PRI    1*     1*     1*     1*     /
GRP01     25E3   PRI    1*     1*     1*     1*     1*     1*     1*     1*     /
GRP02     25E3   PRI    1*     1*     1*     1*     1*     1*     1*     1*     /
/
```

All groups are controlled by oil constraints, but only the field level has water and gas constraints to reflect the actual production facility constraints. The oil production constraint of 40,000, 25,000 and 25,000 stb/d are defined for the field, GRP01 and GRP02 groups, respectively. If the oil rate constraint is exceeded then the wells will be controlled using the priority formulae one, as defined on the PRIORITY keyword in the SCHEDULE section. Similarly for the field, for when the gas constraint is exceeded. Finally, if the field water constraint is surpassed then the worst offending connection and below in the worst offending well are shut.

12.3.90 GCONPROD – GROUP PRODUCTION TARGETS AND CONSTRAINTS

Description

The GCONPROD keyword defines production targets and constraints for groups, including the top most group in the group hierarchy known as the FIELD group. Wells are allocated to groups when the wells are specified by the WELSPECS keyword in the SCHEDULE section. Wells defined to be under group control will have their production rates controlled by the group to which they belong, in addition to any well constraints defined for the wells.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	GRPNAME	A character string of up to eight characters in length that defines the group name for which the group target and constraints are being defined. The group named FIELD is the top most group and should be used to set targets and constraints for the field. Note that the group hierarchy should be defined by the GRUPTREE keyword in the SCHEDULE section, when there is more than one level of groups, otherwise all the groups will sit directly under the FIELD group in the group tree hierarchy.			None
2	TARGET	A defined character string that sets the target production phase for the group, all the other phases will therefore act as constraints. The simulator will attempt to meet the TARGET based on the phase rate stated in items (3) to (6) on this keyword. TARGET should be set to one of the following character strings: 1) NONE: the group has no target phase, but if entered, constraints are still defined and active. 2) FLD: this group is controlled from a higher level group, including the FIELD group. 3) ORAT: the target is set to the surface oil production rate as defined by item (3). 4) WRAT: the target is set to the surface water production rate as defined by item (4). 5) GRAT: the target is set to the surface gas production rate as defined by item (5). 6) LRAT: the target is set to the surface liquid (oil plus water) production rate as defined by item (6). 7) RESV: the target is set to the in situ reservoir volume rate as defined by item (14). All other options are not supported by OPM Flow.			None
3	ORAT	A real positive value that defines the maximum surface oil production rate target or constraint.			
		stb/d	sm^3/day	scc/hour	None
4	WRAT	A real positive value that defines the maximum surface water production rate target or constraint.			
		stb/d	sm^3/day	scc/hour	None

No.	Name	Description			Default
		Field	Metric	Laboratory	
5	GRAT	A real positive value that defines the maximum surface gas production rate target or constraint.			
		Mscf/d	sm^3/day	scc/hour	None
6	LRAT	A real positive value that defines the maximum surface liquid (oil plus water) production rate target or constraint.			
		stb/d	sm^3/day	scc/hour	None
7	ACTION	A defined character string that defines the action to be taken if the constraints in (3) to (6) are violated. ACTION should be set to one of the following character strings: 1) NONE: no action is taken. 2) CON: close the worst offending connection in the worst offending well. If connections have been grouped as completions then the worst offending completion in the worst offending well will be closed. 3) +CON: close the worst offending connection and all below it in the worst offending well. If connections have been grouped as completions then the worst offending completion and all below it in the worst offending well will be closed. 4) WELL: close the worst offending well. 5) PLUG: plug back the worst offending well. This option is not implemented in OPM Flow. 6) RATE: control the group production rate to equal the upper limit. This effectively changes the TARGET to be the violated phase constraint. The corrective action takes places at the end of the time step in which the constraint is violated. Note that only the NONE and RATE options are currently supported by the simulator.			None
8	GRPCNTL	A defined character string that determines if this group is subject to higher level group control. 1) YES: then this group is subject to a higher level group's control and the flow rates for this group will be adjusted accordingly. 2) NO: then this group is NOT subject to a higher level group's control and the flow rates for this group will only be controlled by the parameters for this group. GRPCNTL will be ignored for the FIELD group.			None
9	GRPGUIDE	A real positive value that defines a group's production guide rate expressed as a dimensionless number. A group requires a value for GRPGUIDE only if it is required to produce a specified proportion of a higher level group's rate			
		dimensionless	dimensionless	dimensionless	None

No.	Name	Description			Default
		Field	Metric	Laboratory	
10	GUIPHASE	A defined character string that sets the guide phase to which the guide rate in item (9) applies. GUIPHASE should be set to one of the following character strings: 1) OIL: the guide phase is set to oil. 2) WAT: the guide phase is set to water. 3) GAS: the guide phase is set to gas. 4) LIQ: the guide phase is set to liquid (oil plus water). 5) COMB: the guide phase is set to a linear combination of phases with the coefficients specified by the LINCOM keyword in the SCHEDULE section. 6) INJV: the guide rate is set to the group's reservoir volume injection rate at the start of each timestep. 7) POTN: the guide rate is set to the group's production potential at the start of each timestep. 8) FORM: the guide rate will be based on the formulae defined via the GUIDERAT keyword in the SCHEDULE section. The formulae enables subordinate groups and wells to decrease their contribution from wells producing too much gas or too much water. 9) ' ' or 1* : the group is not under guide rate control, and therefore superordinate group production targets will be pro-rated directly down to its subordinate groups and wells. Note that only the OIL, WAT, GAS, LIQ and default options are currently supported by the simulator.			1*
11	ACTWAT	A defined character string that defines the action to be taken if the WRAT constraint, item (4), is violated. ACTWAT should be set to one of the following character strings: 1) NONE: no action is taken. 2) CON: close the worst offending connection in the worst offending well. If connections have been grouped as completions then the worst offending completion in the worst offending well will be closed. 3) +CON: close the worst offending connection and all below it in the worst offending well. If connections have been grouped as completions then the worst offending completion and all below it in the worst offending well will be closed. 4) WELL: close the worst offending well. 5) PLUG: plug back the worst offending well. This option is not implemented in OPM Flow. 6) RATE: control the group production rate to equal the upper limit. This effectively changes the TARGET to be the violated phase constraint. If defaulted then procedure defined by ACTION, item (7), is applied. The corrective action takes places at the end of the time step in which the constraint is violated. Note that only the NONE and RATE options are currently supported by the simulator.			1*

No.	Name	Description			Default
		Field	Metric	Laboratory	
12	ACTGAS	A defined character string that defines the action to be taken if the GRAT constraint, item (5), is violated. ACTGAS should be set to one of the following character strings: 1) NONE: no action is taken. 2) CON: close the worst offending connection in the worst offending well. If connections have been grouped as completions then the worst offending completion in the worst offending well will be closed. 3) +CON: close the worst offending connection and all below it in the worst offending well. If connections have been grouped as completions then the worst offending completion and all below it in the worst offending well will be closed. 4) WELL: close the worst offending well. 5) PLUG: plug back the worst offending well. This option is not implemented in OPM Flow. 6) RATE: control the group production rate to equal the upper limit. This effectively changes the TARGET to be the violated phase constraint. If defaulted then procedure defined by ACTION, item (7), is applied. The corrective action takes places at the end of the time step in which the constraint is violated Note that only the NONE and RATE options are currently supported by the simulator.			1*
13	ACTLIQ	A defined character string that defines the action to be taken if the LRAT constraint, item (6), is violated. ACLIQT should be set to one of the following character strings: 1) NONE: no action is taken. 2) CON: close the worst offending connection in the worst offending well. If connections have been grouped as completions then the worst offending completion in the worst offending well will be closed. 3) +CON: close the worst offending connection and all below it in the worst offending well. If connections have been grouped as completions then the worst offending completion and all below it in the worst offending well will be closed. 4) WELL: close the worst offending well. 5) PLUG: plug back the worst offending well. This option is not implemented in OPM Flow. 6) RATE: control the group production rate to equal the upper limit. This effectively changes the TARGET to be the violated phase constraint. If defaulted then procedure defined by ACTION, item (7), is applied. The corrective action takes places at the end of the time step in which the constraint is violated. Note that only the NONE and RATE options are currently supported by the simulator.			1*
14	RESV	A real positive value that defines the maximum reservoir volume production rate target or constraint.			
		rb/d	rm^3/day	rcc/hour	None

No.	Name	Description			Default
		Field	Metric	Laboratory	
15	RESVFRAC	A real positive value that defines the maximum reservoir volume production balancing fraction. Not used and should be defaulted with 1*.			1*
16	WGASRATE	Wet gas production rate used in the commercial compositional simulator. Not used and should be defaulted with 1*.			1*
17	CALRATE	Calorific production rate used in the commercial compositional simulator. Not used and should be defaulted with 1*.			1*
18	GASFRAC	Surface gas production fraction used in the commercial compositional simulator Not used and should be defaulted with 1*.			1*
19	WATFRAC	Surface water production fraction used in the commercial compositional simulator. Not used and should be defaulted with 1*.			1*
20	COMBRATE	Linearly combined production rate used in the commercial compositional simulator. Not used and should be defaulted with 1*.			1*
21	COMBPROC	Linearly combined procure for when exceeding COMBRATE, used in the commercial black-oil simulator. Not used and should be defaulted with 1*.			1*

Notes:

1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".

Table 12.33: GCONPROD Keyword Description

See also the GRUPTREE keyword to define the hierarchy of the groups below the FIELD level, the GCONINJE keyword to define a group's injection targets and constraints, the WCONPROD keyword to define a production well's targets and constraints, and the WCONINJE keyword to define an injection well's targets and constraints. All the aforementioned keywords are described in the SCHEDULE section.

Example

The following example defines the production targets and constraints for the field and two groups that are one level below the field group, since the GRUPTREE keyword has not been entered to define the group hierarchy.

```
--
--       GROUP PRODUCTION CONTROLS
--
-- GRUP  CNTL  OIL    WAT    GAS    LIQ    CNTL  GRUP  GUIDE  GUIDE  CNTL
-- NAME  MODE  RATE   RATE   RATE   RATE   OPT   CNTL  RATE   DEF    WAT
GCONPROD
FIELD    ORAT  40E3   60E3   300E3  60E3   1*     1*    1*     1*     1*      /
GRP01    FLD   25E3   1*     1*     1*     1*     1*    1*     1*     1*      /
GRP02    FLD   25E3   1*     1*     1*     1*     1*    1*     1*     1*      /
/
```

All groups are controlled by oil rate targets or constraints, but only the field level has water, gas and liquid constraints to reflect the actual production facility constraints. The wells under group control will be produced based on oil potential of each of the wells under group control, such that the field oil production target of 40,000 stb/d is honored and subject to the other phase fluid constraints. In addition, GRP01 and GRP02 oil rate values of 25,000 stb/d are constraints as these two groups are subject to the FIELD level targets and constraints.

12.3.91 GCONSALE – DEFINE GROUP SALES GAS PRODUCTION TARGETS AND CONSTRAINTS

Description

GCONSALE defines group sales gas production targets and constraints for when the gas production from an oil field group is exported under a Gas Sales Agreement ("GSA") and the oil field group also has oil production targets and constraints.

Note that the keyword should not be used to control sales gas for a gas field group, as the gas injection rate is used to control the sales gas production with this keyword, that is:

$$\begin{aligned}\text{Gas Sales Rate} = {} & \text{Total Group Gas Production Rate} \\ & - \text{Group Gas Injection Rate} \\ & + \text{Total Group Gas Import Rate} \\ & - \text{Total Group Gas Consumption}\end{aligned} \tag{12.23}$$

Thus, surplus gas that cannot be sold is re-injected, which requires that there are active gas injectors in the model that are subordinate to groups' with gas sales targets. Note that the surplus gas re-injection rates are automatically calculated by OPM Flow at each time step.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	GRPNAME	A character string of up to eight characters in length that defines the group name for which the group gas sales target and constraints are being defined. The group named FIELD is the top most group and should be used to set the gas sales targets and constraints for the field. Note that the group hierarchy should be defined by the GRUPTREE keyword when there is more than one level of groups, otherwise all the groups will sit directly under the FIELD group in the group tree hierarchy.			None
2	GSALE	GSALE should either be set to: 1) A real positive value that defines the gas sales rate for the group, or, 2) a real negative value that switches off both gas sales and gas re-injection for the group. Note that if GSALE has been set to switch off both gas sales and gas re-injection, then the GCONINJE keyword in the SCHEDULE section may be used to re-enable gas re-injection again.			
		Mscf/d	sm^3/day	scc/hour	None
3	GSALEMAX	A real positive value that must be greater than GSALE that defines the maximum allowed gas sales rate. If GSALE exceeds GSALEMAX then the action defined by the ACTION variable on this keyword is implemented at the end of the current time step.			
		Mscf/d	sm^3/day	scc/hour	1×10^{20}

No.	Name	Description			Default
		Field	Metric	Laboratory	
4	GSALEMIN	A real positive value that must be less than GSALE that defines the minimum allowed gas sales rate. If GSALE is less than GSALEMAX then one of the following actions will be implemented at the end of the current time step: 1) If the group's maximum gas production rate constraint is constraining the gas rate, then reset the constraint to satisfy the group's minimum gas sales rate (GSALEMIN), else: 2) If the group has active subordinate dedicated gas producers, as defined by the WGASPROD keyword in the SCHEDULE section, then reset their gas target rates to satisfy GSALEMIN, else: 3) If there are subordinate dedicated gas producers for the group in a drilling queue, open the next dedicated well and set the well's gas rate to satisfy GSALEMIN. Note that only wells that are subordinate to the group and are not under gas rate control or group prioritization are considered for opening. 4) If there are no appropriate subordinate dedicated gas producers for the group in a drilling queue, open the next non-dedicated well and set the well's gas rate to satisfy GSALEMIN. Again, note that only wells that are subordinate to the group and are not under gas rate control or group prioritization are considered for opening. If none of the above actions can be implemented then the minimum gas sales rate will not be satisfied.			-1×10^{20}
5	ACTION	A defined character string that defines the action to be taken if the maximum gas rate, GSALEMAX, is violated. ACTION should be set to one of the following character strings: 1) NONE: no action is taken. 2) CON: close the worst offending connection in the worst offending well. If connections have been grouped as completions then the worst offending completion in the worst offending well will be closed. 3) +CON: close the worst offending connection and all below it in the worst offending well. If connections have been grouped as completions then the worst offending completion and all below it in the worst offending well will be closed. 4) WELL: close the worst offending well. 5) PLUG: plug back the worst offending well. This option is not implemented in OPM Flow. 6) RATE: the group's production rate target is reduced to equal GSALEMAX, after accounting for fuel gas (GCONSUMP keyword) and the current rate of re-injection. This will also place the group on gas production control. 7) END: stop the simulation at the end of the report step. The corrective action takes places at the end of the time step in which the constraint is violated.			None

Notes:

1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".

Table 12.34: GCONSALE Keyword Description

GCONSALE and also be used with the GCONINJE keyword in the SCHEDULE section in order to apply additional limits, for example by applying a maximum group injection rate. In this scenario, the TARGET variable on the GCONINJE keyword must be set to "REIN" together, and if desired, a re-injection fraction (REIN on the GCONINJE keyword), or any other constraint.

See also the GCONSUMP in the SCHEDULE section that defines the fuel gas requirements for groups.

Example

The following examples sets the field gas sales target rate:

```
--
--       GROUP GAS SALES FOR OIL FIELDS
--
-- GRUP  GAS    MAX    MIN    CNTL
-- NAME  SALES  RATE   RATE   ACTN
GCONSALE
FIELD    40E3   50E3   20E3   RATE                                          /
/
```

Here the field has a gas sales target of 40 MMscf/d, with a maximum rate of 50 MMscf/d and a minimum of 20 MMscf/d. If the maximum gas sales rate is exceeded then the group's gas production rate target is reduced to equal GSALEMAX, after accounting for fuel gas and the current rate of re-injection. This will also place the group on gas production control.

12.3.92 GCONSUMP – DEFINE GROUP GAS CONSUMPTION AND GAS IMPORT TARGETS

Description

GCONSUMP defines the group gas consumption rate either as an actual rate or as a percentage of the group's production. In both oil and gas fields produced gas is commonly used as fuel to support the processing and utility facilities needed to run the plant.

In addition to defining gas consumption, the keyword can also be used to define the group's gas import rate, if required. This is used to import gas into the model from other sources (fields, reservoirs etc.) that are not included in the current run. For example, if a several fields are supplying gas to a power plant (field A, B and C), but only one is being modeled in the current import deck (A), then production from the other two fields (B and C) can be incorporated into model in order to meet the plant demand. Note in this case the import gas rates from fields B and C are fixed, and therefore field A acts like a "swing" producer to match the gas demand target.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	GRPNAME	A character string of up to eight characters in length that defines the group name for which the group gas consumption is being defined. The group named FIELD is the top most group and should be used to set the fuel consumption for the field. Note that the group hierarchy should be defined by the GRUPTREE keyword when there is more than one level of groups, otherwise all the groups will sit directly under the FIELD group in the group tree hierarchy.			None
2	GASFUEL	A real value that defines the gas consumption, that is the fuel gas consumed by the group, either defined as a volumetric rate or as a fraction of the group's gas production. The two options are implemented by: 1) Volumetric Rate Option: Setting GASFUEL to a value greater than zero will be interpreted as the actual fuel gas rate consumed by the group. 2) Fraction of Produced Gas: Setting GASFUEL to a negative value between minus one and zero will be interpreted as a fraction of the groups production rate. For example, if GASFUEL is entered as -0.05, would mean 5% of the group's produced gas will be consumed as fuel.			
		Mscf/d	sm^3/day	scc/hour	0.0
3	GASIMP	A real positive value greater than zero that defines the amount of gas to be imported into the group. This option is currently not supported by OPM Flow			
		stb/d	sm^3/day	scc/hour	0.0
4	GASNODE	A character string of up to eight characters in length that defines the network node in the Extended Network Model, for which the fuel gas should be removed (GASFUEL) or the imported gas (GASIMP) assigned. This option is currently not supported by OPM Flow			None

No.	Name	Description			Default
		Field	Metric	Laboratory	
Notes: 1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".					

Table 12.35: GCONSUMP Keyword Description

If the group is acting under Group Gas Sales control via the GCONSALE keyword in the SCHEDULE section, then the sales gas is calculated by:

$$\begin{aligned}\text{Gas Sales Rate} = {} & \text{Total Group Gas Production Rate} \\ & - \text{Group Gas Injection Rate} \\ & + \text{Total Group Gas Import Rate} \\ & - \text{Total Group Gas Consumption}\end{aligned} \tag{12.24}$$

If the group is acting under Group Gas Re-Injection control via the GCONINJE keyword in the SCHEDULE section, then the group gas injection rate calculated by:

$$\begin{aligned}\text{Group Gas Injection Rate} = {} & \text{Group Gas Injection Rate} \times \text{Group Re-Injection Fraction} \\ & + \text{Total Group Gas Import Rate} \\ & - \text{Total Group Gas Consumption}\end{aligned} \tag{12.25}$$

Note

In oil fields with no gas compression typical values of fuel gas range from three to five percent.

Example

The first example sets the fuel gas consumption to 3.0 MMscf/d for the field.

```
--
--       GROUP GAS CONSUMPTION (FUEL) AND IMPORT
--
-- GRUP   GAS      GAS
-- NAME   FUEL    IMPORT
--        ------   -------
GCONSUMP
FIELD    3.0E3                                                    /
/
```

The second example sets group PLAT-EST's fuel consumption to be 5% of the platform's produced gas and group PLAT-WST's to a constant 1.0 MMscf/d.

```
--
--       GROUP GAS CONSUMPTION (FUEL) AND IMPORT
--
-- GRUP   GAS      GAS
-- NAME   FUEL    IMPORT
--        ------   -------
GCONSUMP
PLAT-WST -0.050                                                   /
PLAT-EST  1.0E3                                                   /
/
```

12.3.93 GCONTOL – DEFINE GROUP CONSTRAINT TOLERANCE

Description

The GCONTOL keyword defines the tolerance and parameters used to control the accuracy of group targets and constraints, including the field's targets and constraints. The keyword sets the tolerance and number of Newton iterations for each time step so that the wells under group control can match the desired group targets and constraints.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	GRPCNV	GRPCNV is a real positive value less than one that sets the group tolerance criteria used to define if convergence has been satisfied for group's under rate control. Here, GRPCNV is an acceptable fraction of the group's rate target. Thus, the a numerical quantity 0.01 means that values must be with 0.01 (or 1.0%) of the groups' production target. For groups under priority control, as per GCONPRI keyword in the SCHEDULE section, GRPCNV is ignored. Note that this criteria may not be satisfied if the number of Newton iterations used in updating the well targets, as set by the NUPCOL keyword in the RUNSPEC section, or the NUPCOL parameter on this keyword, is exceeded. In this case, and only if the well potentials allow, the well rates are re-calculated ignoring the NUPCOL limit. Group tolerance criteria can also be set by the GRPCNV parameter on the NETBALAN keyword in the SCHEDULE section. If both values of GRPCNV have been entered, then the minimum of the two is used. The default value means there is no convergence criteria.			
		dimensionless	dimensionless	dimensionless	1.0 x 1020
2	NUPCOL	A positive integer that defines the maximum number of Newton iterations used to update well targets within a time step. Wells under group control may suffer from some dependency with other wells in the same group that are under group control. This may cause some oscillation in the production and injection well rates within the group. In order to avoid this, after the number Newton iterations within a time step surpasses NUPCOL, the group well rates are frozen until the time step has converged. Reducing the potential of well rate oscillations within the time step may result in the group targets and limits not being exactly being met in this case. Increasing the value of NUPCOL will improve the accuracy of the group targets and limits at the expense of computational efficiency. NUPCOL can also be set by the NUPCOL keyword in the RUNSPEC section. A value entered on this keyword overwrites any previously entered values of NUPCOL, including the value set by the NUPCOL keyword. The default value of 1* invokes the last previously entered value of either by the NUPCOL keyword or this keyword.			1*

No.	Name	Description			Default
		Field	Metric	Laboratory	
		dimensionless	dimensionless	dimensionless	
3	GASCNV	In the commercial compositional simulator GASCNV is a real positive value less than one that sets the tolerance criteria used to define if convergence has been satisfied for well gas injection rates. GASCNV is also used in conjunction with well availability gas fraction quantities in the commercial compositional simulator The value is ignored by OPM Flow.			
		dimensionless	dimensionless	dimensionless	1.0 x 10-3
4	GASMXITE	In the commercial compositional simulator GASMXITE is a positive integer value that defines the maximum number of iterations for the gas injection computation. The value is ignored by OPM Flow.			
		dimensionless	dimensionless	dimensionless	5

Notes:

1) The keyword is terminated by a "/".

Table 12.36: GCONTOL Keyword Description

See also the NUPCOL keyword in the RUNSPEC section, and the NETBALAN and WLIMTOL keywords in the SCHEDULE section that control the network balancing convergence criteria and the tolerance parameters for wells, respectively.

Examples

The example sets the group target convergence criteria to 1% (0.01) with a maximum of four Newton iterations (NUPCOL).

```
--
--       GROUP CONSTRAINT TOLERANCE
--
--       GROUP  NEWTON  INJEC  NEWTON
--       TOL    ITERS   TOL    ITERS
--       -----  -----   -----  ------
GCONTOL
         0.01    4       1*     1*                    /
```

12.3.94 GCUTBACK – DEFINE GROUP CUTBACK LIMITS AND PARAMETERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, GCUTBACK, defines a production group's cutback limits and parameters. See also the WCUTBACK keyword in the SCHEDULE section that provides similar functionality for wells.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.95 GCUTBACT – DEFINE GROUP TRACER CUTBACK LIMITS AND PARAMETERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, GCUTBACT, defines a production group's cutback limits and parameters based on the named produced tracer from the group. See also the WCUTBACT keyword in the SCHEDULE section that provides similar functionality for groups.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.96 GDCQ – DEFINE GROUP MULTIPLE DAILY CONTRACT QUANTITIES

Description

The GDCQ keyword defines the Daily Contract Quantities ("DCQ") for when multiple group contracts are required when the Gas Field Operations model has been activated by the GASFIELD keyword in the RUNSPEC section, or the GWSINGF has been invoked to define multiple group contracts in the SCHEDULE section. The group contracts must first be defined by the GSWINGF keyword, followed by the GCDQ keyword, and then the GASYEAR or GASPERIO keywords. GCDQ may be repeated in the SCHEDULE section to reset group DCQs.

See also the SWINGFAC keyword that set a single group DCQ at the field level, as opposed to having multiple DCQ group contracts using the GDCQ keyword.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.97 GDCQECON – GROUP ECONOMIC CRITERIA FOR DCQ PRODUCTION GROUPS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The GDCQECON keyword defines economic criteria for DCQ production groups, including the field level group FIELD, that have previously been defined by the GCONPROD keywords in the SCHEDULE section. Note that wells are allocated to a group when they are specified by the WELSPECS keyword and wells can also have economic controls. Wells under group control are therefore subject to the economic criteria set via the GCONPROD and CECON keywords in the SCHEDULE section and the controls specified by the WECON keyword.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	GRPNAME	A character string of up to eight characters in length that defines the group name for which the group target and constraints are being defined. The group named FIELD is the top most group and should be used to set targets and constraints for the field. Note that the group hierarchy should be defined by the GRUPTREE keyword when there is more than one level of groups, otherwise all the groups will sit directly under the FIELD group in the group tree hierarchy.			None
2	DCQ	A real positive value that defines the minimum economic DCQ gas production rate, below which an economic action of shutting in or stopping all the wells in the group, as requested by item (9) of the WELSPECS keyword. Note if GRPNAME is equal to FIELD then the run will be terminated. A value less than or equal to zero switches of this criteria.			
		Mscf/d	sm^3/day	scc/hour	0.0

Notes:

1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".

Table 12.37: GDCQECON Keyword Description

See also the WELSPECS keyword to define a wells shut-in or stop options, GCONPROD for group controls, and WECON for setting a well's economic criteria. All the aforementioned keywords are described in the SCHEDULE section.

Example

The following example defines the minimum DCQ for the field to be 10 MMscf/d.

```
--
--       GROUP ECONOMIC CRITERIA FOR DCQ PRODUCTION GROUPS
--
-- GRUP  GAS
-- NAME  DCQ
GDCQECON
FIELD    10E3                                                              /
/
```

12.3.98 GDRILPOT – DEFINE GROUP POTENTIAL RATES FOR AUTOMATIC DRILLING

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, GDRILPOT, defines the minimum group potential rate that will result in a well from the one of the automatic drilling queues, as defined by either the QDRILL or WDRILPRI keywords in the SCHEDULE section, to be drilled and placed on production. The advantage of using a group's potential, as oppose to a minimum rate limit, is that setting the potential greater than the group's minimum flow rate, will result in well being drilled in time to support the desired production rate.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.99 GECON – GROUP ECONOMIC CRITERIA FOR PRODUCTION GROUPS

Description

The GECON keyword defines economic criteria for production groups, including the top level FIELD group, that have been created by having wells assigned to them via the WELSPECS keyword in the SCHEDULE section.

Note that wells are allocated to a group when they are specified by the WELSPECS keyword and wells can also have well level economic controls. Wells under group control are therefore subject to the economic criteria set via the GECON and WECON keywords and the controls specified by the GCONPROD keyword in the SCHEDULE section.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	GRPNAME	A character string of up to eight characters in length that defines the group name for which the group target and constraints are being defined. The group named FIELD is the top most group and should be used to set targets and constraints for the field. Note that the group hierarchy should be defined by the GRUPTREE keyword when there is more than one level of groups, otherwise all the groups will sit directly under the FIELD group in the group tree hierarchy.			None
2	ORAT	A real positive value that defines the minimum economic surface oil production rate, below which an economic action of shutting in or stopping all the wells in the group, as requested by the AUTO variable item (9) of the WELSPECS keyword. A value less than or equal to zero switches off this criteria.			
		stb/d	sm3/day	scc/hour	0.0
3	GAS	A real positive value that defines the minimum economic surface gas production rate, below which an economic action of shutting in or stopping all the wells in the group, as requested by the AUTO variable item (9) of the WELSPECS keyword. A value less than or equal to zero switches off this criteria,			
		Mscf/d	sm3/day	scc/hour	0.0
4	WCUT	A real positive value that defines the maximum economic surface water cut, above which an economic action will take place. Water cut is defined as: $f_w = \frac{q_w}{q_w + q_o}$, and the various actions that are available if the water cut limit is exceeded are described in item (7). A value less than or equal to zero switches off this criteria.			
		dimensionless	dimensionless	dimensionless	0.0
5	GOR	A real positive value that defines the maximum economic surface gas-oil ratio, above which an economic action will take place, as defined by item (7). A value less than or equal to zero switches off this criteria.			0.0

No.	Name	Description			Default
		Field	Metric	Laboratory	
		Mscf/stb	sm3/sm3	scc/scc	
6	WGR	A real positive value that defines the maximum economic surface water-gas ratio, above which an economic action will take place, as defined by item (7). A value less than or equal to zero switches off this criteria.			
		stb/Mscf	sm3/sm3	scc/scc	0.0
7	ACTION	A defined character string that defines the action to be taken if the economic WCUT, GOR, or WGR limits are violated. ACTION should be set to one of the following character strings: 1) NONE: no action is taken. 2) CON: close the worst offending connection in the worst offending well. If connections have been grouped as completions then the worst offending completion will be closed. 3) +CON: close the worst offending connection and all below it in the worst offending well. If connections have been grouped as completions then the worst offending completion and all below it will be closed. 4) WELL: shut or stop the worst offending well as per the AUTO variable on the WELSPECS keyword. 5) PLUG: plug back the worst offending well based on the plug back length and options defined on the WPLUG keyword in the SCHEDULE. The corrective action takes places at the end of the time step in which the constraint is violated. Only the NONE option is currently supported by the simulator.			None
8	END	A defined character string that defines if the simulation should terminate if all the producing wells in the group, including the FIELD group, are shut or stopped. END should be set to one of the following character strings: 1) NO: no action is taken and the run continues. 2) YES: terminate the run at the next report time step. Only the NO option is currently supported by the simulator.			NO
9	MXWELS	A positive integer defining the maximum number of producing and injecting wells for this this group and any subordinate groups. The default value of zero implies that there is no limit to the number of wells. This is not currently supported by the simulator and must be defaulted.			0
Notes: 1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".					

Table 12.38: GECON Keyword Description

See also the WELSPECS keyword to define a well's shut-in or stop options, GCONPROD for group controls, and WECON for setting a well's economic criteria. All the aforementioned keywords are described in the SCHEDULE section.

Example

The following example defines the economic criteria for the field with a minimum oil rate of 2,000 m3/day and a maximum water cut of 95%.

```
--
--        GROUP ECONOMIC CRITERIA FOR PRODUCTION GROUPS
--
-- GRUP   OIL    GAS     WCT     GOR     WGR     WORK    END    MAX
-- NAME   MIN    MIN     MAX     MAX     MAX     OVER    RUN    WELLS
GECON
FIELD     2E3    1*      0.95      1*     1*      CON    'YES'   1*                    /
/
```

If the economic limits are violated then the run will stop at the next report time step.

12.3.100 GECONT – GROUP TRACER ECONOMIC CRITERIA FOR PRODUCTION GROUPS

Description

The GECONT keyword defines tracer economic criteria for production groups, including the field level group FIELD, that have previously been defined by the GCONPROD keywords in the SCHEDULE section, for tracers define by the TRACER keyword in the PROPS section.

Note that wells are allocated to a group when they are specified by the WELSPECS keyword and wells can also have economic controls. Wells under group control are therefore subject to the economic criteria set via the GCONPROD and CECON keywords in the SCHEDULE section and the controls specified by the WECON keyword.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1-1	GRPNAME	A character string of up to eight characters in length that defines the group name for which the group target and constraints are being defined. The group named FIELD is the top most group and should be used to set targets and constraints for the field. Note that the group hierarchy should be defined by the GRUPTREE keyword when there is more than one level of groups, otherwise all the groups will sit directly under the FIELD group in the group tree hierarchy.			None
1-2	ACTION	A defined character string that defines the action to be taken if the economic WCUT, GOR, or WGR limits are violated. ACTION should be set to one of the following character strings: 1) NONE: no action is taken. 2) CON: close the worst offending connection in the worst offending. If connections have been grouped as completions then the worst offending completion will be closed. 3) +CON: close the worst offending connection and all below it in the worst offending well. If connections have been grouped as completions then the worst offending completion and all below it in the worst offending well will be closed. 4) WELL: shut or stop the well as per the AUTO variable on the WELSPECS keyword. 5) PLUG: plug back the worst offending well based on the plug back length and options defined on the WPLUG keyword in the SCHEDULE. The corrective action takes places at the end of the time step in which the constraint is violated.			None
1-3	END	A defined character string that defines if the simulation should terminate if all the producing wells in the group, including the FIELD group, are shut or stopped. END should be set to one of the following character strings: 1) NO: no action is taken and the run continues. 2) YES: terminate the run at the next report time step.			NO

No.	Name	Description			Default
		Field	Metric	Laboratory	
1-4	MXWELS	A positive integer defining the maximum number of producing and injecting wells for this group and any subordinate groups. The default value of zero implies that there is no limit to the number of wells.			0
1-5	/	Record one terminated by a "/"			Not Applicable
2-1	NAME	A three letter character string defining the tracer's name. Note it is best to void names beginning with the letters F, S, and T as these names may create naming issues in post-processing software.			None
2-2	MXTOTAL	A real positive value that defines the maximum total (free plus solution) tracer rate. Tracer units are per those defined by the carrying fluid, oil, gas, water, etc.			None
2-3	MXFREET	A real positive value that defines the maximum total (free plus solution) tracer concentration. Tracer units are per those defined by the carrying fluid, oil, gas, water, etc.			None
2-4	MXFREEQ	A real positive value that defines the maximum free tracer rate. Tracer units are per those defined by the carrying fluid, oil, gas, water, etc.			None
2-5	MXCONC	A real positive value that defines the maximum free tracer concentration. Tracer units are per those defined by the carrying fluid, oil, gas, water, etc.			None
2-6	MXSOLNQ	A real positive value that defines the maximum solution rate. Tracer units are per those defined by the carrying fluid, oil, gas, water, etc.			None
2-7	MXSOLNC	A real positive value that defines the maximum solution concentration. Tracer units are per those defined by the carrying fluid, oil, gas, water, etc.			None
2-8	/	Record two terminated by a "/"			Not Applicable
3-1	/	Group terminated by a "/"			Not Applicable

Notes:

1) GECONT keyword consists of two records, with entries 1-1 to 1-5 representing record one items and 2-1 to 2-8 representing record number two items, in the "No." column in this table. A maximum of three type two records can be entered following a type one record.
2) Each type one and type two records are terminate by a "/" as indicated in the table, and a group data set is terminated by a further "/", after which additional group data sets can be entered stating with a record of type one followed by type two.
3) The keyword the keyword should be terminated by an additional "/" after the group data set termination "/" character.

Table 12.39: GECONT Keyword Description

See also the WELSPECS keyword to define a wells shut-in or stop options, GCONPROD for group controls, and WECON for setting a well's economic criteria. All the aforementioned keywords are described in the SCHEDULE section.

Example

The following example defines the tracer economic criteria for the field and two groups, FLTBLK1 and FLTBLK,.

```
--
--       GROUP TRACER ECONOMIC CRITERIA FOR PRODUCTION GROUPS
--
-- GRUP  WORK    END    MAX
-- NAME  OVER    RUN    WELLS
GECONT
FIELD    +CON   'YES'   1*                                    / START OF GROUP
--
--       TRACER  TRACER  TRACER  TRACER  TRACER  TRACER  TRACER
--       NAME    TOTAL   TOTAL   FREE    FREE    SOLN    SOLN
--               RATE    CONCEN  RATE    CONCEN  RATE    CONCEN
         PLY     1000.0                                       /
         BRI     1000.0                                       /
         TR1     1*      0.7500                               /
                                                              /
FLTBLK1  +CON   'YES'   1*                                    / START OF GROUP
--
--       TRACER  TRACER  TRACER  TRACER  TRACER  TRACER  TRACER
--       NAME    TOTAL   TOTAL   FREE    FREE    SOLN    SOLN
--               RATE    CONCEN  RATE    CONCEN  RATE    CONCEN
         PLY     800.0                                        /
         BRI     800.0                                        /
                                                              /
FLTBLK2  +CON   'YES'   1*                                    / START OF GROUP
--
--       TRACER  TRACER  TRACER  TRACER  TRACER  TRACER  TRACER
--       NAME    TOTAL   TOTAL   FREE    FREE    SOLN    SOLN
--               RATE    CONCEN  RATE    CONCEN  RATE    CONCEN
         PLY     800.0                                        /
         BRI     800.0                                        / END OF GROUP
                                                              / END OF KEYWORD
```

If the economic limits are violated then the worst offending connection and all below it in the worst offending well will be closed, If connections have been grouped as completions then the worst offending completion and all below it in the worst offending well will be closed.

12.3.101 GEFAC – DEFINE GROUP EFFICIENCY

Description

Defines a group's efficiency or up-time as opposed to setting the efficiency factors for individual wells.

Note that wells are allocated to a group when they are specified by the WELSPECS keyword and wells can also have efficiency factors.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	GRPNAME	A character string of up to eight characters in length that defines the group name for which the group efficiency factor is being defined. The group named FIELD is the top most group and cannot have an efficiency factor set. Note that the group hierarchy should be defined by the GRUPTREE keyword when there is more than one level of groups, otherwise all the groups will sit directly under the FIELD group in the group tree hierarchy.			None
2	FACTOR	A real positive value that is less than or equal to one that defines the efficiency factor for the group. If a group's down time is 5% then FACTOR should be set to 0.95 (1.0 – 0.05).			
		dimensionless	dimensionless	dimensionless	1.0
3	GRPNETWK	A defined character string that determines if the GRPNAME efficiency factor should be transferred to the equivalent Extended Network Model node, and should be set to either: 1) NO: The group's equivalent Extended Network Model node flow rates are not reduced by the efficiency factor FACTOR. 2) YES: The group's equivalent Extended Network Model node flow rates are reduced by the efficiency factor FACTOR This option is only applicable for the Extended Network Model, as in the Standard Network Model groups flow rates are always used in the calculation of pressure drops.			YES

Notes:

1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".

Table 12.40: GEFAC Keyword Description

See also the WEFAC keyword in the SCHEDULE section to define a well's' efficiency factor.

Example

```
--
--        GROUP EFFICIENCY FACTORS
--
-- GRUP   EFF       NETWK
-- NAME   FACT      OPTN
--       ------    ------
GEFAC
PLATFORM 0.950                                                          /
SUBSEA1  0.860                                                          /
/
```

In the above example the group PLATFORM has it's efficiency factor (up time) set to 0.95 and the subsea group SUBSEA1 has an up time of 0.860.

12.3.102 GLIFTLIM – GROUP ARTIFICIAL LIFT CONSTRAINTS

Description

The GLIFTLIM keyword defines the maximum number of wells on artificial lift and the maximum amount of the artificial lift that is available for a group, including the top most group in the group hierarchy known as the FIELD group. Wells are allocated to groups when the wells are specified by the WELSPECS keyword in the SCHEDULE section. Wells defined to be under group control will have their production rates controlled by the group to which they belong, in addition to any well constraints defined for the wells.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	GRPNAME	A character string of up to eight characters in length that defines the group name for which the group's artificial lift constraints are being defined. The group named FIELD is the top most group and should be used to set targets and constraints for the field. Note that the group hierarchy should be defined by the GRUPTREE keyword when there is more than one level of groups, otherwise all the groups will sit directly under the FIELD group in the group tree hierarchy.			None
2	MXLIFT	A real positive value that defines the total amount of artificial lift available for this group and any subordinate groups. The units for MXLIFT are the same as that defined by the ALQ parameter on the VFPPROD keyword in the SCHEDULE section. For example, if ALQ has been set to GRAT on the VFPPROD keyword, then MXLIFT would be the maximum amount of gas lift gas available for this group and any subordinate groups, and the units would Mscf, assuming FIELD units had been activated in the RUNSPEC section. The default value of zero implies that there is no limit applied to the group and its subordinate groups.			
		See VFPPROD (ALQ)	See VFPPROD (ALQ)	See VFPPROD (ALQ)	0
3	MXWELS	A positive integer defining the maximum number of producing wells on artificial lift for this group and any subordinate groups. The default value of zero implies that there is no limit to the number of wells.			
		dimensionless	dimensionless	dimensionless	0

Notes:

1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".

Table 12.41: GLIFTLIM Keyword Description

See also the GRUPTREE keyword to define the hierarchy of the groups below the FIELD level, the GCONPROD and GCONINJE keywords to define a group's production and injection rate targets and constraints, the WCONPROD keyword to define a production well's targets and constraints, and the WCONINJE keyword to define an injection well's targets and constraints. All the aforementioned keywords are described in the SCHEDULE section.

Example

The following example defines the artificial lift constraints for the field, assuming all the wells are on gas lift.

```
---
--       GROUP ARTIFICIAL LIFT CONSTRAINTS
--
-- GRUP  MAX   MAX
-- NAME  ALQ   WELLS
GLIFTLIM
FIELD    20E3  20                                                    /
/
```

Here the maximum amount of gas lift gas for the field is set to 20.0 MMscf/f and a maximum of 20 wells can utilize gas lift at a time.

12.3.103 GLIFTOPT – DEFINE GROUP GAS OPTIMIZATION LIMITS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The GLIFTOPT keyword defines the maximum amount of gas lift gas available and the maximum amount of gas the group can produce, including the top most group in the group hierarchy known as the FIELD group, for when gas lift optimization has been activated via the LIFTOPT keyword in the SCHEDULE section. Note that the LIFTOPT keyword should precede the GLIFTOPT keyword in the SCHEDULE section in order to activate the gas lift optimization facility.

Wells are allocated to groups when the wells are specified by the WELSPECS keyword in the SCHEDULE section. Wells defined to be under group control will have their production rates controlled by the group to which they belong, in addition to any well constraints defined for the wells, including any well gas lift optimization parameters on the WLIFTOPT keyword.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	GRPNAME	A character string of up to eight characters in length that defines the group name for which the group's gas lift optimization parameters are being defined. The group named FIELD is the top most group and should be used to set targets and constraints for the field. Note that the group hierarchy should be defined by the GRUPTREE keyword when there is more than one level of groups, otherwise all the groups will sit directly under the FIELD group in the group tree hierarchy.			None
2	MXLIFT	A real value that defines the total amount of gas lift gas available for this group and any subordinate groups, multiplied by their respective efficiency factors. The units for MXLIFT are the same as that defined by the ALQ parameter on the VFPPROD keyword in the SCHEDULE section. In this case ALQ should be GRAT on the VFPPROD keyword, as MXLIFT applies to the maximum amount of gas lift gas available. The default value of zero, or a negative value implies that there is no limit applied to the group and its subordinate groups.			
		Mscf/d	sm^3/day	scc/hour	0
3	MXGAS	A real value that defines the total amount of gas the group can process. This is the sum of the gas lift gas plus the produced gas for this group and any subordinate groups, multiplied by their respective efficiency factors. The default value of zero, or a negative value implies that there is no limit applied to the group and its subordinate groups			
		Mscf/d	sm^3/day	scc/hour	0

Notes:

1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".

Table 12.42: GLIFTOPT Keyword Description

See also the LIFTOPT keyword to activate gas lift optimization, the WLIFTOPT keyword to define the wells under gas lift optimization control, the GRUPTREE keyword to define the hierarchy of the groups below the FIELD level, the GCONPROD and GCONINJE keywords to define a group's production and injection rate targets and constraints. All the aforementioned keywords are described in the SCHEDULE section.

Example

The following example fist switches on gas lift optimization via the LIFTOPT keyword and then defines the artificial lift constraints for the field, assuming all the well are on gas lift, using the GLIFTOPT keyword.

```
--       ACTIVATE GAS LIFT OPTIMIZATION AND PARAMETERS
--
-- INCR   INCR   TSTEP    NEWTON
-- GAS    OIL    INTVAL   OPTN
LIFTOPT
12.5E3    5E-3   0.0      YES                                                  /
/
--
--       GROUP GAS LIFT OPTIMIZATION CONSTRAINTS
--
-- GRUP  MAX       MAX
-- NAME  GAS ALQ   TOTAL GAS
GLIFTOPT
FIELD    200E3     1*                                                         /
/
```

Here the LIFTOPT keyword defines the maximum incremental gas lift gas quantity to be 12.5 x 103 m3, the minimum incremental oil gain per m3 of gas lift gas is set to 5.0 x 10-3 m3, the time step interval is set to zero to perform the gas optimization every time step, and finally the gas lift optimization will be performed NUPCOL Newton iterations for the time step.

The GLIFTOPT sets the maximum amount of gas lift gas for the field to 200,000 m3 and there is no maximum limit for the total maximum amount of gas that the group can process.

12.3.104 GNETDP – GROUP NETWORK PRESSURE AND RATE CONTROLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The GNETDP keyword sets a group's minimum and maximum network pressure and rate controls for when the either the Standard Network or the Extended Network options have been activated, and the group is part of a network. The keywords allows for the pressure of the group to vary in order to satisfy the rate conditions declared by this keyword. The Standard Network option is invoked if the GRUPTREE, GRUPNET, GNETINJE, GNETPUMP, etc. series of keywords have been used in the SCHEDULE section. Whereas, the Extended Network option is activated by the NETWORK keyword in the RUNSPEC section. Several keywords, including, GNETDP, can be used by both network options.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.105 GNETINJE – DEFINE GROUP INJECTION NETWORK CONFIGURATION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The GNETINJE keyword defines the configuration of a group injection network for when the either the Standard Network or the Extended Network options have been activated. The Standard Network option is invoked if the GRUPTREE, GRUPNET, GNETINJE, GNETPUMP, etc. series of keywords have been used in the SCHEDULE section. Whereas, the Extended Network option is activated by the NETWORK keyword in the RUNSPEC section. Several keywords, including, GNETINJE, can be used by both network options.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.106 GNETPUMP – STANDARD NETWORK AUTOMATIC COMPRESSOR AND PUMPS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The GNETPUMP keyword defines the configuration of automatic compressors and pumps in a production Standard Network, for when the Standard Network option is invoked by the GRUPTREE, GRUPNET, GNETINJE, GNETPUMP, etc., series of keywords in the SCHEDULE section. Although several keywords can be used by both the Standard and Extended Network options, GNETPUMP can only be used with the Standard Network option. The equivalent keyword for the Extended Network option is the NETCOMPA keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.107 GPMAINT – DEFINE GROUP PRESSURE MAINTENANCE TARGETS AND CONTROLS

Description

The GPMAINT keyword defines the groups under pressure maintenance control, the associated flow rate and pressure targets, and fluid in-place regions associated with pressure maintenance, as well as various pressure maintenance controls. GPMAINT allows for various regions, as defined by the FIPNUM or FIP keywords in the REGIONS section, to have their average reservoir pressure maintained at a specified value.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	GRPNAME	A character string of up to eight characters in length that defines the group name for which the group's associated FIPNUM region will have a targeted average reservoir pressure maintained. The group named FIELD is the top most group and can also be used to set pressure target for the whole field. Note that the group hierarchy should be defined by the GRUPTREE keyword when there is more than one level of groups, otherwise all the groups will sit directly under the FIELD group in the group tree hierarchy.			None
2	GRPCNTL	A defined character string of length four, that sets the production or injection control for the group, used to maintain the average pressure for the FIPNUM region. GRPCNTL should be set to one of the following character strings: 1) NONE: the group will no longer maintain the average reservoir pressure for the FIPNUM region by adjusting production and injection rates. 2) GINJ: GRPNAME's gas in situ reservoir volume injection rate (RESV) will be adjusted in order to maintain FIPNUM's average reservoir pressure. 3) GINS: GRPNAME's gas surface rate (GRAT) will be used to maintain FIPNUM's average reservoir pressure. 4) OINJ: GRPNAME's oil in situ reservoir volume injection rate (RESV) will be adjusted in order to maintain FIPNUM's average reservoir pressure. 5) OINS: GRPNAME's oil surface rate (ORAT) will be used to maintain FIPNUM's average reservoir pressure. 6) PROD: GRPNAME's total in situ reservoir volume production rate (RESV) will be adjusted in order to maintain FIPNUM's average reservoir pressure. Note that group cannot be under prioritization group control, as per the GCONPRI keyword in the SCHEDULE section, with this option. 7) WINJ: GRPNAME's water in situ reservoir volume injection rate (RESV) will be adjusted in order to maintain FIPNUM's average reservoir pressure 8) WINS: GRPNAME's water surface rate (WRAT) will be used to maintain FIPNUM's average reservoir pressure.			None

No.	Name	Description			Default
		Field	Metric	Laboratory	
3	FIPNUM	A positive integer value that defines the region for which pressure maintenance is to be applied by GRPNAME, where FIPNUM may be define as per the: 1) FIPNUM keyword in the REGIONS section. 2) The FIP keyword in the REGIONS section, used to define the fluid in-place name and the associated region numbers for each grid block. This option is used in conjunction with FIPNAME (item (5)) to define the name of fluid in-place region and FIPNUM, which in this case is the region number associated with FIPNAME. 3) Alternatively, a value of zero may be entered, in which case pressure maintenance is applied to the whole field. The maximum number of FIPNUM and FIP regions is set by the REGDIMS(NMFIPR) or the TABDIMS(NTFIP) keywords(variables) in the RUNSPEC section. If both REGDIMS(NMFIPR) and TABDIMS(NTFIP) have been defined then the maximum of the two is used.			None
4	FIPNAME	A character string of up to five characters in length, defining the fluid in-place's name, as defined by the FIP keyword in the REGIONS section. For example, if the FIP keyword has been used to define a FIP group or family, named FIPBLK-A, then FIPNAME would be set to BLK-A. The default value of 1* means that FIPNUM on this keyword applies to the standard FIPNUM array and not to the FIP defined group regions.			1*
5	TARGET	A real positive value that defines the average hydrocarbon pore volume weighted reservoir pressure target for the region, for which GRPNAME should attempt to satisfy. Note that the average pressure report in the RPTSCHED series of reports is a pore volume weight average reservoir pressure, not the average hydrocarbon pore volume weighted reservoir pressure specified by TARGET on this keyword. Thus, there will be small differences between the two numbers.			
		psia	barsa	atma	None
6	ALPHA	A real positive value that defines the proportionality constant used to control the flow rates in order to maintain/reach the TARGET average hydrocarbon pore volume pressure for the region. See equation (12.26). Larger values of ALPHA accelerate the time for the region's pressure to satisfy the required target pressure (TARGET). However, ALPHA values that are too large may cause the region's pressure to oscillate. Thus, the value of ALPHA should be a value that gives the expected steady state flow rate (TARGET) of the group divided by a reasonable transient pressure error.			
		Liquid stb/d/psia Gas Mscf/d/psia RESV: rb	Liquid sm^3/day/bars Gas sm3/day/bars RESV rm^3	Liquid scc/hour/atm Gas scc/hour/atm RESV: rcc	None

No.	Name	Description			Default
		Field	Metric	Laboratory	
7	BETA	A real positive value that defines the time interval used to control the flow rates in order to maintain the TARGET average hydrocarbon pore volume pressure for the region. See equation (12.26). Larger values of BETA reduce the tendency for the regional pressure to oscillate, but consequently decrease the rate at which the pressure reaches its target value. This is because the pressure error is measured at the end of the previous time step, resulting in a "delay" in the response. Also, the larger the time steps the greater the propensity for the pressure to oscillate for given ALPHA and BETA. As a guide BETA should be set to a value that is at least as large as the maximum time step size (see the TUNING keyword in the SCHEDULE section and section *2.2 Running OPM Flow 2023-10 From The Command Line* for setting the maximum time step size).			
		day	day	day	1*

Notes:

1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".

Table 12.43: GPMAINT Keyword Description

The GPMAINT keyword utilizes control theory[309], [310], [311] and [312] to calculate the required injection and production rates to ensure some stability in the behavior of the average hydrocarbon pore volume reservoir pressure for a given region and the injection rates over time. For example, if the group's flow control (GRPCNT) is set to one of the in situ reservoir volume injection rates (GINJ, OILJ or WATJ), say Q, then the group's flow injection rates for each time step is calculated by:

$$Q = Q_i + \alpha \left(\left(P_{TARGET} - P_{i-1} \right) + \frac{\sum_{i=1}^{i-1} \left(P_{TARGET} - P_{i-1} \right) \times \Delta t_i}{\beta} \right) \quad (12.26)$$

Where:

P_{TARGET} = the regions average hydrocarbon pore volume weighted reservoir pressure target, the TARGET parameter in Table 12.43.

P_{i-1} = the resulting average hydrocarbon pore volume weighted reservoir pressure for the region from the preceding completed time step.

Q = the new in situ reservoir volume rate.

Q_i = the initial in situ reservoir volume rate at the time the keyword was activated in the input. Subsequent GPMAINT keywords will reset Q_i.

α = the proportionality constant used to control the flow rates, the ALPHA parameter in Table 12.43.

β = the time interval parameter used to control the flow rates, the BETA parameter in Table 12.43.

[309] *John Doyle, Bruce Francis, Allen Tannenbaum, Feedback Control Theory, Macmillan Publishing Co., 1990.*

[310] *Karl J. Åström; Richard M. Murray (2008). Feedback Systems: An Introduction for Scientists and Engineers. Princeton University Press. ISBN 978-0-691-13576-2.*

[311] *Andrei, Neculai (2005)., Modern Control Theory – A Hstorical Perspective, Research Institute for Informatics, Center for Advanced Modeling and Optimization, 8-10, Averescu Avenue, Bucharest 1, Romania.*

[312] *Sontag, Eduardo (1998). Mathematical Control Theory: Deterministic Finite Dimensional Systems. Second Edition, Springer. ISBN 978-0-387-98489-6.*

The $\sum_{i=1}^{i-1}\left(P_{TARGET} - P_{i-1}\right)\times\Delta t_i$ term in equation (12.26) is the cumulative sum of the average hydrocarbon pore volume weighted reservoir pressure error for the region, times the time step length, from when the last GPMAINT keyword was entered, up to the preceding completed time step.

The objective of equation (12.26) is to stabilize the injection/production rates for the group for when there are changes to the average hydrocarbon pore volume weighted reservoir pressure target set by the initial and subsequent GPMAINT keywords, or by any group constraints applied afterwards. In addition, if the pressure error is small $\left(P_{TARGET} - P_{i-1}\right)$ then the equation should result in approximately the same rates as the previous time step, if the GPMAINT has been previously entered.

However, if the first time the keyword is entered coincides with the start of injection, then Qi in equation (12.26) by definition will be zero, and the injection rate will start from zero and increase as calculated by the equation. The ALPHA and BETA parameters in Table 12.43 can be used to control how the rate is built up to meet the desired average hydrocarbon pore volume weighted reservoir pressure target.

Examples

The first example uses the gas surface rate to maintain the fields' average hydrocarbon pore volume pressure at 225 barsa, with the ALPHA and BETA parameters set to 40.0 and 70.0 respectively.

```
--
--        GROUP PRESSURE MAINTENANCE TARGETS AND CONTROLS
--
-- GRUP   CNTL    FIPNUM   FIP      PRESS    ALPHA   BETA
-- NAME   MODE    REGION   FIPNAME  TARGET   CONST   CONST
GPMAINT
FIELD     GINS    0        1*       225      40.0    70.0        /
/
```

The second example uses group's BLK-A gas surface rate to maintain the average hydrocarbon pore volume pressure at 200 barsa for FIPNUM regions one to four, with the ALPHA and BETA parameters set to 40.0 and 70.0 respectively.

```
--
--        GROUP PRESSURE MAINTENANCE TARGETS AND CONTROLS
--
-- GRUP   CNTL    FIPNUM   FIP      PRESS    ALPHA   BETA
-- NAME   MODE    REGION   FIPNAME  TARGET   CONST   CONST
GPMAINT
BLK-A     GINS    1        1*       200      40.0    70.0        /
BLK-A     GINS    2        1*       200      40.0    70.0        /
BLK-A     GINS    3        1*       200      40.0    70.0        /
BLK-A     GINS    4        1*       200      40.0    70.0        /

BLK-B     WINJ    1        FLT-B    215      30.0    65.0        /
BLK-B     WINJ    2        FLT-B    215      30.0    65.0        /
BLK-B     WINJ    3        FLT-B    215      30.0    65.0        /
BLK-B     WINJ    4        FLT-B    215      30.0    65.0        /
/
```

For group BLK-B, water in situ reservoir volume injection rate (RESV) is used to maintain FIP group/family FLT-B's average reservoir pressure at 215 barsa for FIPFLT-B regions one to four. Here the ALPHA and BETA parameters set to 30.0 and 65.0 respectively.

12.3.108 GRADGRUP – DEFINE GROUP HISTORY MATCH GRADIENT FILE OUTPUT

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The GRADGRUP keyword defines the SUMMARY field and group vectors that should be written to the History Match Gradient output file, for when the History Match Gradient option has been activated by the HMDIMS keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.109 GRADRESV – DEFINE SOLUTION DERIVATIVE HISTORY MATCH GRADIENT OUTPUT

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The GRADRESV keyword defines the SOLUTION derivative arrays that should be written to the History Match Gradient output file, for when the History Match Gradient option has been activated by the HMDIMS keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.110 GRADRFT – DEFINE RFT DERIVATIVE HISTORY MATCH GRADIENT OUTPUT

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The GRADRFT keyword defines the derivative well RFT data, the SOLUTION pressure and saturations at a well's connected grid block, that should be written to the History Match Gradient output file, for when the History Match Gradient option has been activated by the HMDIMS keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.111 GRADWELL – DEFINE WELL HISTORY MATCH GRADIENT FILE OUTPUT

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The GRADWELL keyword defines the SUMMARY well vectors that should be written to the History Match Gradient output file, for when the History Match Gradient option has been activated by the HMDIMS keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.112 GRDREACH – DEFINE RIVER AND GRID BLOCK CONNECTIONS

Description

The GRDREACH keyword defines the location of grid blocks connecting to a previously defined river, for when the River option has been activated via the RIVRDIMS keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.113 GRUPMAST – DEFINE MASTER AND SLAVE GROUPS

Description

The GRUPMAST keyword defines master groups and their associated slave groups for when the Reservoir Coupling option as been activated by the GRUPMAST and SLAVES keywords in the SCHEDULE section. Reservoir coupling allows for independent reservoir simulation decks (SLAVES) to be controlled by a separate master run file. For example, if there are five separate reservoir models each representing one field, one of the four would be used as the master and the other four would be the subordinate SLAVES.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.114 GRUPNET – DEFINE GROUP STANDARD NETWORK PARAMETERS

Description

The GRUPNET keyword defines the standard group network parameters used to model the flow and pressures behavior within the network. The group hierarchy is defined by the GRUPTREE keyword and wells are assigned to groups using the WELSPECS keyword, both keywords are in the SCHEDULE section.

Group pressure values are optionally entered for each group in the network together with a vertical lift performance ("VFP") table that the determines the pipeline pressure behavior from the LOWER to the HIGHER group given the current flowing conditions; the group relationship is defined by the GRUPTREE keyword. The VFP table is entered via the VFPPROD keyword for production pipelines and VFPINJ for injection pipelines. These are the same keywords as used for well modeling, however, the manner in which they are generated by an external program is completely different when used for pipeline modeling.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	GRPNAME	A character string of up to eight characters in length that defines the group name for which the network parameters are being defined. The group named FIELD is the top most group and may be used as a GRUPNAME.			None
2	PRES	A real value that defines the fixed pressure for this group when the group is a terminating group. If the group is not a terminating group then PRES should be defaulted with 1* or set to a negative number.			
		psia	barsa	atma	1*
3	VFPTAB	A positive integer greater than or equal to zero that defines the VFPPROD or VFPINJ vertical lift performance table to be used for calculating the pipeline pressures connecting the LOWER and HIGHER groups in the network. Note that: 1) The default value of zero implies that there is no pipeline connecting the LOWER and HIGHER groups. 2) If PRES is set to a real positive number then VFPTAB should be set to zero as this implies that GRPNAME is a terminating group and therefore there is no pipeline connecting GRPNAME to a HIGHER group. 3) If PRES and VFPTAB are defaulted with 1* or zero, then GRPNAME is not part of the network. 4) IF VFPTAB is set equal to 9999 then this implies that there is no pressure change between the LOWER and HIGHER group. If a non-zero value is entered then the vertical lift performance tables must be entered via the VFPPROD or VFPINJ keyword in the SCHEDULE section.			0

No.	Name	Description			Default
		Field	Metric	Laboratory	
4	ALQ-PIPE	A real positive value that defines the artificial lift quantity to be used in conjunction with the vertical lift performance table (VFPPROD) assigned to the group via VPFTAB variable. The vertical lift performance table and the artificial lift quantity ALQ-WELL are used with the pipeline fluid rates to calculate the pipeline change pressure between the LOWER and HIGHER groups. Note that the units for ALQ-PIPE are dependent on the associated variable on the VFPPROD keyword and may represent a pump or a compressor depending how the VFPPROD table was generated by an external program.			0.0
5	OPTION1	A defined character string that defines if a group's production target should be achieved by adjusting the tubing pressure of the wells within the group or by the adjusting the well rates by their guide rate. OPTION1 should be set to one of the following character strings: 1) YES: the group production target is achieved by adjusting the tubing pressure of the wells within the group, so that all wells flow at the same tubing head pressure. This is normally used for wells that flow into a common manifold, for example a sub-sea completion manifold. If a group is using this option and has a higher group with production targets or constraints, then this group should have it's guide rate set via the GCONPROD keyword in the SCHEDULE section, to ensure that the well's within this group operate at the same tubing head pressure. 2) NO: the group production target is achieved by adjusting the guide rates of the wells within the group. This is the standard method in matching group targets and may result with the wells within the having different tubing head pressures. Only groups containing wells can use OPTION1 equal to YES or NO, a group without wells should set OPTION1 to NO. Numerical convergence controls and iteration limits for wells using OPTION1 set equal to YES are defined via the NETBALAN keyword in the SCHEDULE section. Only the NO option is currently supported by the simulator.			NO

No.	Name	Description			Default
		Field	Metric	Laboratory	
6	OPTION2	A defined character string that defines if well gas lift gas flows through the group's pipeline. OPTION2 should be set to one of the following character strings: 1) NO: no well gas lift gas is allowed to flow through the pipeline only produced reservoir gas is allowed to flow through the pipeline. 2) FLO: the well gas lift gas is added to the gas flow along the pipeline. The well gas lift gas is the sum of the calculated ALQ-WELL values for all the subordinate wells multiplied by their efficiency factors. 3) ALQ: the pipeline gas lift gas value (ALQ-PIPE) is the sum of the calculated ALQ-WELL values for all the subordinate wells multiplied by their efficiency factors. This means that the ALQ-PIPE gas lift gas value declared on item (4) is ignored. If either FLO or ALQ have been selected then artificial lift quantity for the pipeline (ALQ-PIPE) and the wells (ALQ-WELL) must be defined as gas lift gas on the VFPPROD tables. A well's specific gas lift gas quantity is set via the ALQ-WELL variable on the WCONPROD keyword in the SCHEDULE section. Only the NO and FLO options are currently supported by the simulator.			NO
7	OPTION3	A defined character string that defines if the ALQ-PIPE variable should be reset to an equivalent surface oil or gas density flowing along the pipeline. OPTION3 should be set to one of the following character strings: 1) DENO: set ALQ-PIPE to the average surface density of the oil flowing along the pipeline. 2) DENG: set ALQ-PIPE to the average surface density of the gas flowing along the pipeline. 3) NONE: no change to ALQ-PIPE. If either DENO or DENG have been selected then artificial lift quantity on the VFPPROD tables must be based on the same density parameter. These options are normally used when a mixture of oil or gas with different surface densities flows into the network. Only the NONE option is currently supported by the simulator.			NONE
Notes: 1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".					

Table 12.44: GRUPNET Keyword Description

See also the WELSPECS keyword to define wells, the VFPPROD and VFPINJ keywords that the define vertical lift performance table to be used for calculating the pipeline pressures connecting the LOWER and HIGHER group in the network. All the aforementioned keywords are described in the SCHEDULE section.

Examples

The following example defines a network based on two groups

```
--
--       DEFINE GROUP STANDARD NETWORK PARAMETERS
--
-- GRUP  CNTL   VFP    PUMP   MANIFOLD  INCLUDE   ALQ
```

```
-- NAME  PRES   TABLE  POWER  GROUP     LIFT GAS  DENS
GRUPNET
PROD-A   1200.  1*                                    /
PROD-B   1*     1      1*    'YES'     1*        1*   /
/
```

The next example is more complex and is taken form the Norne model.

```
--
--       DEFINE GROUP STANDARD NETWORK PARAMETERS
--
-- GRUP  CNTL   VFP    PUMP   MANIFOLD  INCLUDE   ALQ
-- NAME  PRES   TABLE  POWER  GROUP     LIFT GAS  DENS
GRUPNET
FIELD    20.0   5*                                    /
PROD     20.0   5*                                    /
MANI-B2  1*     8      1*      NO       2*            /
MANI-B1  1*     8      1*      NO       2*            /
MANI-K1  1*    9999    4*                             /
B1-DUMMY 1*    9999    4*                             /
MANI-D1  1*     8      1*      NO       2*            /
MANI-D2  1*     8      1*      NO       2*            /
MANI-K2  1*    9999    4*                             /
D2-DUMMY 1*    9999    4*                             /
MANI-E1  1*     9      1*      NO       2*            /
MANI-E2  1*     9      4*                             /
/
```

Here the FIELD controlling pressure is set at 20 barsa and the same limit is used for group PROD which sits directly under the FIELD group (see Error: Reference source not found).

12.3.115 GRUPRIG – GROUP DRILLING AND WORKOVER RIG SPECIFICATIONS

Description

Defines a groups drilling and workover specifications.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.116 GRUPSLAV – DEFINE SLAVE GROUPS IN SLAVE RESERVOIRS

Description

The GRUPSLAV keyword defines slave groups in a slave input deck and their associated master groups in the master run, for when the Reservoir Coupling option as been activated by the GRUPMAST and SLAVES keywords in the SCHEDULE section. This keyword is required for every slave input deck. Reservoir coupling allows for independent reservoir simulation decks (SLAVES) to be controlled by a separate master run file. For example, if there are five separate reservoir models each representing one field, one of the four would be used as the master and the other four would be the subordinate SLAVES.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.117 GRUPTARG – MODIFY GROUP TARGETS AND CONSTRAINTS VALUES

Description

The GRUPTARG keyword modifies the target and constraints values of both rates and pressures for previously defined groups without having to define all the variables on the group control keywords: GCONPROD or GCONPRI keywords. Variables not changed by the GRUPTARG keyword remain the same as those previously entered via the group control keywords or previously entered GRUPTARG keywords. Note that the group must still be initially be fully defined using the GCONPROD or GCONPRI keywords. All the aforementioned keywords are described in the SCHEDULE section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

Note that wells are allocated to groups when the wells are specified by the WELSPECS keyword in the SCHEDULE section. Wells defined to be under group control will have their production rates controlled by the group to which they belong, in addition to any well constraints defined for the wells.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	GRPNAME	A character string of up to eight characters in length that defines the group name for which the group target and constraints are being defined. The group named FIELD is the top most group and should be used to set targets and constraints for the field. Note that the group hierarchy should be defined by the GRUPTREE keyword when there is more than one level of groups, otherwise all the groups will sit directly under the FIELD group in the group tree hierarchy.			None
2	TARGET	A defined character string that sets the item to be changed for the group the value of the item is set by item (3). 1) ORAT: reset the surface oil production rate value as defined by item (3). 2) WRAT: reset the surface water production rate value as defined by item (3). 3) GRAT: reset the surface gas production rate value as defined by item (3). 4) LRAT: reset the surface liquid (oil plus water) production rate value as defined by (3). 5) RESV: reset the in situ reservoir volume rate value as defined by (3). 6) GUID: reset the guide rate value for wells operating under group control. Note TARGET only defines the variable to be changed, it does not change how a group is controlled. For example, if a group is operating on ORAT control, as defined by the previously entered GCONPROD keyword, entering TARGET equal to LRAT with a value, changes the liquid constraint but the group still remains on ORAT control. Use the GCONPROD or GCONPRI keywords in the SCHEDULE section to change the control mode of a well.			None

No.	Name	Description			Default
		Field	Metric	Laboratory	
3	VALUE	A real positive value that defines the value of the variable declared by TARGET			
	Liquid	stb/d	sm^3/day	scc/hour	
	Gas	Mscf/d	sm3/day	scc/hour	
	Res Vol	rb/d	rm^3/day	rcc/hour	
	Pressure	psia	barsa	atma	None

Notes:

1) The keyword is followed by any number records with each record terminated by a "/" and the keyword should be terminated by a "/".

Table 12.45: GRUPTARG Keyword Description

See also the WELTARG and WELCNTL keyword, in the SCHEDULE section that can be used to reset a well's control mode, as well as a well's target and constraints of both rates and pressures.

Example

The following example below shows the oil rates for the field at the start of the schedule section (January 1, 2000).

```
-- ------------------------------------------------------------------------
-- 01 JAN 2000 START OF SCHEDULE SECTION
-- ------------------------------------------------------------------------
--
--       GROUP PRODUCTION CONTROLS
--
-- GRUP  CNTL  OIL    WAT    GAS    LIQ    CNTL  GRUP  GUIDE  GUIDE  CNTL
-- NAME  MODE  RATE   RATE   RATE   RATE   OPT   CNTL  RATE   DEF    WAT
GCONPROD
FIELD    ORAT  40E3   60E3   30E3   65E3   1*     1*    1*     1*     1*      /
/
DATES
01 FEB 2000 /
/
--
--       GROUP PRODUCTION AND INJECTION TARGETS
--
--       GROUP    GROUP  TARGET
--       NAME     TARG   VALUE
GRUPTARG
         FIELD    ORAT   45E3                                              /
         FIELD    LIQ    75E3                                              /
/
```

From January 1, 2000 to February 1, 2000 the field is on oil rate control and has a target oil rate of 40,000 stb/d, a maximum water handling capacity of 60,000 stb/d, a maximum liquid capacity of 65,000 stb/d, and a maximum gas constraint of 30 MMscf/d. After February 1, 2000 the field's target oil rate is increased to 45,000 stb/d and the maximum liquid constraint is increased to 75,000 stb/s; all the other parameters remain unchanged.

12.3.118 GRUPTREE – DEFINE GROUP TREE HIERARCHY

Description

GRUPTREE defines the group hierarchy of groups that have been created by having wells assigned to them via the WELSPECS keyword in the SCHEDULE section, By default three group levels are defined that sets the wells as level three, reporting directly to defined groups at level two, and the level two groups reporting to the FIELD group at level one. If a different configuration is required then the GRUPTREE keyword should be used to define the group hierarchy by defining a lower level group that reports directly to a higher level group.

No.	Name	Description	Default
1	LOWER	A character string of up to eight characters in length that defines the group name which belongs to the HIGHER group. The group named FIELD is the top most group and should NOT be used as as a group name for the LOWER group name. Undefined group relationships are automatically assigned to the FIELD group.	None
2	HIGHER	A character string of up to eight characters in length that defines the HIGHER group name that the LOWER group belongs to. The group named FIELD is the top most group and can be used as as the HIGHER group name. Undefined group relationships are automatically assigned to the FIELD group.	None
Notes: 1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".			

Table 12.46: GRUPTREE Keyword Description

A group hierarchy can have any number of levels but groups that have other groups as LOWER groups cannot also have wells for the HIGHER group. Thus, a group either contains wells or has LOWER groups

See also the GCONPROD and GCONINJE for defining group production and injection volumes, and the WELSPECS keywords to allocate wells to groups. All the aforementioned keywords are described in the SCHEDULE section.

Examples

The first example defines PLAT01 and PLAT03 reporting to the FIELD level (default if these records are omitted) and PLAT02 reporting to PLAT01.

```
--
--        DEFINE GROUP TREE HIERARCHY
--
--        LOWER     HIGHER
--        GROUP     GROUP
GRUPTREE
          PLAT01    FIELD                                                 /
          PLAT02    PLAT01                                                /
          PLAT03    FIELD                                                 /
/
```

The next example is more complex and is taken form the Norne model.

```
--
--        DEFINE GROUP TREE HIERARCHY
--
--        LOWER       HIGHER
--        GROUP       GROUP
GRUPTREE
        'INJE'      'FIELD'                                                 /
        'PROD'      'FIELD'                                                 /
        'MANI-B2'   'PROD'                                                  /
        'MANI-B1'   'PROD'                                                  /
        'MANI-D1'   'PROD'                                                  /
        'MANI-D2'   'PROD'                                                  /
        'MANI-E1'   'PROD'                                                  /
        'MANI-E2'   'PROD'                                                  /
        'MANI-K1'   'MANI-B1'                                               /
        'MANI-K2'   'MANI-D2'                                               /
        'MANI-C'    'INJE'                                                  /
        'MANI-F'    'INJE'                                                  /
        'WI-GSEG'   'INJE'                                                  /
        'B1-DUMMY'  'MANI-B1'                                               /
        'D2-DUMMY'  'MANI-D2'                                               /
/
```

The group hierarchy for this example is shown below.

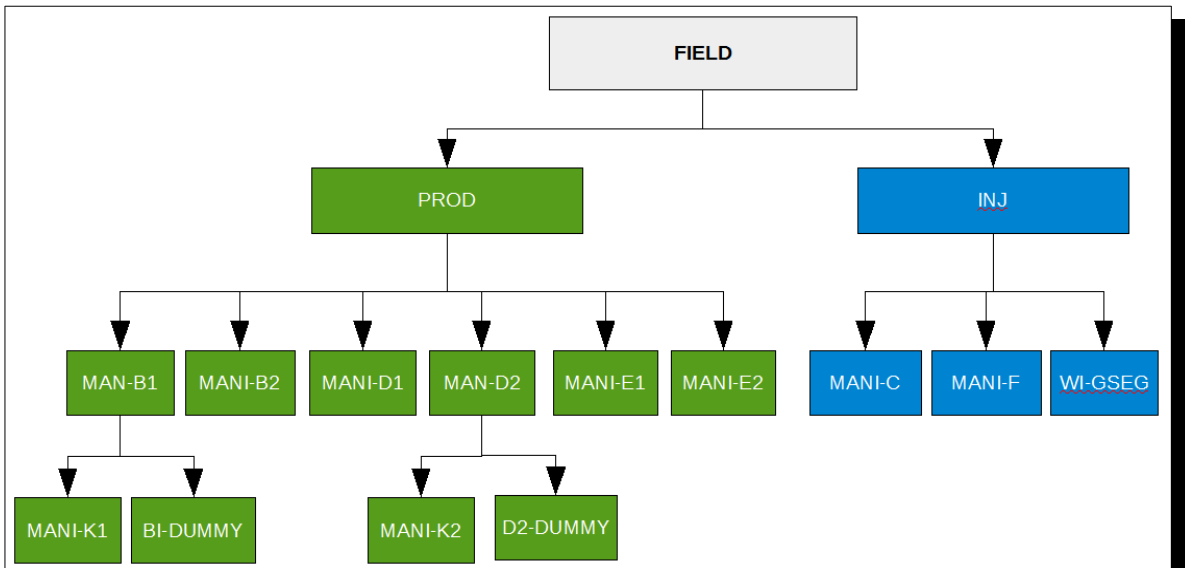


Figure 12.4: Norne Group Tree Hierarchy Example

Here groups PROD, INJ, MAN-B1, and MAN-D2 report to higher level groups and the other remaining groups all have individual wells allocated to them instead.

12.3.119 GSATINJE – DEFINE GROUP SATELLITE INJECTION RATES

Description

The GSATINJE keyword defines a satellite group's oil, gas and water injection rates in the model. Satellite groups are not connected to the reservoir model and therefore have no wells or subordinate groups associated with them, they are nevertheless connected to other higher level groups and higher level groups within a network model (if activated). They thus provide a means to "add-in" outside injection and production to the model without modeling the "add-in" reservoir model.

The keyword is used to define injection rates into the model from other sources (fields, reservoirs etc.) that are not defined in the current run. For example, if a several fields are producing through a common plant (field A, B and C), but only one is being modeled in the current input deck (A), then injection and production from the other two fields (B and C) can be incorporated into model in order to meet the plant injection rates for field A. Note in this case the import rates from fields B and C are fixed, and therefore field A acts like a "swing" injector to match the overall injection requirement.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	GRPNAME	A character string of up to eight characters in length that defines the satellite group name for which the group injection rates are being defined. The group named FIELD is the top most group and should not be used to set targets and constraints for the whole field. Note that the group hierarchy should be defined by the GRUPTREE keyword in the SCHEDULE, when there is more than one level of groups, otherwise all the groups will sit directly under the FIELD group in the group tree hierarchy. Note that a satellite group cannot have subordinate groups or wells.			None
2	TYPE	A defined character string that defines the type of injection fluid. TYPE should be set to one of the following character strings: 1) GAS: for a gas injection well. 2) OIL: for a water injection well. 3) WAT: for a water injection well.			0.0
3	RATE	A real positive value that defines the surface injection rate for the phase declared by the TYPE variable.			
		Liquid stb/d Gas Mscf/d	Liquid sm^3/day Gas sm3/day	Liquid scc/hour Gas scc/hour	0.0
4	RESV	A real positive value that defines the reservoir volume injection rate for the phase declared by the TYPE variable for the satellite group. Generally, RESV should be set to zero, unless one wishes to have the satellite reservoir volume rate added to the group's superordinate groups. If RESV is set to zero for all satellite groups and if the superordinate group has a reservoir volume injection target, a voidage replacement target, or a production balancing fraction target, then only the non-satellite group volumes will be used.			0.0

No.	Name	Description			Default
		Field	Metric	Laboratory	
		rtb/d	rm^3/day	rcc/hour	
5	CALRATE	Calorific injection rate used in the commercial compositional simulator. Not used and should be defaulted with 1* or zero.			0.0
Notes: 1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".					

Table 12.47: GSATINJE Keyword Description

See also the GSATPROD and GRUPTREE keywords to define satellite production rates and the group hierarchy, respectively. For non-satellite groups see the GCONINJE and GCONPROD keywords. All the aforementioned keywords are in the SCHEDULE section.

Note

Once a group has been defined to be a satellite group, via the GSATINJE and GSATPROD keywords, then the equivalent modeled group keywords, GCONINJE and GCONPROD in the SCHEDULE section, cannot be used to set the operating conditions for satellite groups, only the GSATINJE and GSATPROD keywords may be used.

Example

The example below is based on an oil field, with the main field FLD-A being modeled in the input deck with two groups (FLD-A1 and FLD-A2), combined with one satellite field supplementing oil production and water injection (FLD-B).

```
-- ================================================================================
--
-- SCHEDULE SECTION
--
-- ================================================================================
SCHEDULE
--
--        DEFINE GROUP TREE HIERARCHY
--        LOWER      HIGHER
--        GROUP      GROUP
GRUPTREE
         FLD-A      FIELD                                                        /
         FLD-A1     FLD-A                                                        /
         FLD-A2     FLD-A                                                        /
         FLD-B      FIELD                                                        /
/
```

```
--
--       GROUP PRODUCTION CONTROLS
--
-- GRUP  CNTL  OIL    WAT    GAS    LIQ    CNTL  GRUP  GUIDE  GUIDE  CNTL
-- NAME  MODE  RATE   RATE   RATE   RATE   OPT   CNTL  RATE   DEF    WAT
GCONPROD
FIELD    ORAT  20E3   30E3    50E3  1*     1*    1*                          /
FLD-A    FLD   20E3   30E3    50E3  1*     1*    1*                          /
/
--
--       LOAD INCLUDE FILE - LOAD ALL WELLS AND VFP TABLES
--
INCLUDE
         'FLD-A-P50-WELLS.INC'  /
--
--       GROUP INJECTION TARGETS AND CONSTRAINTS
--
-- GRUP  FLUID CNTL   SURF   RESV   REINJ  VOID  GRUP  GUIDE  GUIDE GRUP  GRUP
-- NAME  TYPE  MODE   RATE   RATE   FRAC   FRAC  CNTL  RATE   DEF   REINJ RESV
GCONINJE
FIELD    WAT   REIN   35E3   1*     1*     1.0    NO   1*     1*    1*    1*   /
/
--
--       SATELLITE GROUP PRODUCTION CONTROLS
--
-- GRUP  OIL    WAT    GAS    RESV   GAS   CALORIFIC
-- NAME  RATE   RATE   RATE   RATE   LIFT  RATE
GCONPROD
FLD-B    5E3    0.00   2E3    1*     1*                  /
/
--
--       SATELLITE GROUP INJECTION CONTROLS
--
-- GRUP  FLUID  SURF   RESV   CALORIFIC
-- NAME  TYPE   RATE   RATE   RATEV
GSATINJE
FLD-B    WAT    10E3   1*     1*                        /
/
--
--       ADVANCE SIMULATION BY REPORTING DATE
--
DATES
         1  FEB   2021  /
/
--
--       SATELLITE GROUP PRODUCTION CONTROLS
--
-- GRUP  OIL    WAT    GAS    RESV   GAS   CALORIFIC
-- NAME  RATE   RATE   RATE   RATE   LIFT  RATE
GCONPROD
FLD-B    5E3    0.00   2E3    0.0    1*                  /
/
--
--       SATELLITE GROUP INJECTION CONTROLS
--
-- GRUP  FLUID  SURF   RESV   CALORIFIC
-- NAME  TYPE   RATE   RATE   RATEV
GSATINJE
FLD-B    WAT    10E3   1*     1*                        /
/
```

Here FLD-B will supplement FLD-A's water re-injection rated by 10,000 stb/d of water, subject to a maximum field water injection rate of 35,000 stb/d of water on the GCONINJE keyword. This means that

FLD-A's maximum water injection rate cannot exceed 25,000 stb/d In terms of oil rates, GCONPROD defines a maximum oil rate of 20,000 stb/d of which 5,000 stb/d is from FLD-B. If any of the constraints on the GCONINJE and GCONPROD keywords are violated, then the appropriate action will occur only for FLD-A, in order to ensure the group constraints are honored.

12.3.120 GSATPROD – DEFINE GROUP SATELLITE PRODUCTION RATES

Description

The GSATPROD keyword defines a satellite group's oil, gas and water production rates in the model. Satellite groups are not connected to the reservoir model and therefore have no wells or subordinate groups associated with them, they are nevertheless connected to other higher level groups and higher level groups within a network model (if activated). They thus provide a means to "add-in" outside injection and production to the model without modeling the "add-in" reservoir model.

The keyword is used to import additional fluids into the model from other sources (fields, reservoirs etc.) that are not defined in the current run. For example, if a several fields are supplying gas to a power plant (field A, B and C), but only one is being modeled in the current input deck (A), then production from the other two fields (B and C) can be incorporated into model in order to meet the plant demand. Note in this case the import gas rates from fields B and C are fixed, and therefore field A acts like a "swing" producer to match the gas demand target.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	GRPNAME	A character string of up to eight characters in length that defines the satellite group name for which the group production import rates are being defined. The group named FIELD is the top most group and should not be used with this keyword. Note that the group hierarchy should be defined by the GRUPTREE keyword in the SCHEDULE section when there is more than one level of groups, otherwise all the groups will sit directly under the FIELD group in the group tree hierarchy. Note that a satellite group cannot have subordinate groups or wells.			None
2	ORAT	A real positive value, greater than or equal to zero, that defines the satellite's surface oil production rate to be imported into the model.			
		stb/d	sm^3/day	scc/hour	0.0
3	WRAT	A real positive value, greater than or equal to zero, that defines the satellite's surface water production rate to be imported into the model.			
		stb/d	sm^3/day	scc/hour	0.0
4	GRAT	A real positive value, greater than or equal to zero, that defines the satellite's surface gas production rate to be imported into the model.			
		Mscf/d	sm^3/day	scc/hour	0.0
5	RESV	A real positive value, greater than or equal to zero, that defines the satellite's reservoir volume production rate to be imported into the model.			
		rtb/d	rm^3/day	rcc/hour	0.0

No.	Name	Description			Default
		Field	Metric	Laboratory	
6	GASLIFT	A real value, greater than or equal to zero, that defines the satellite's gas lift gas to be imported for this group. A value for GASLIFT is only required if externally supplied gas lift is being imported into the model, and: 1) The Network Model is active in the input deck via the NETWORK keyword in the RUNSPEC section. Here, GASLIFT is applied to the production network in order to calculate the pressure drop through the network, for branches that have: (1) OPTION2 set to FLO on the GRUPNET keyword in the SCHEDULE section, or (2) the GASLIFT parameter on the NODEPROP keyword in the SCHEDULE section set to YES, for when the Extended Network Model is being used. as per the BRANPROP and NODEPROP keywords, in the SCHEDULE section. 2) The Gas Lift Optimization Model is active in the deck via the LIFTOPT keyword in the SCHEDULE section. In this case GASLIFT is applied to GRUPNAME's superordinate group's gas supply constraints based on the GLIFTOPT keyword in the SCHEDULE section. For clarity, GLIFTOPT constraint violations result in the groups being modeled having their rates adjusted, satellite group rates are not effected.			
		Mscf/d	sm^3/day	scc/hour	0.0
7	CALRATE	Calorific production rate used in the commercial compositional simulator. Not used and should be defaulted with 1* or zero.			0.0

Notes:

1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".

Table 12.48: GSATPROD Keyword Description

See also the GSATINJE and GRUPTREE keywords to define satellite injection rates and the group hierarchy, respectively. For non-satellite groups see the GCONINJE and GCONPROD keywords. All the aforementioned keywords are in the SCHEDULE section.

Note

Once a group has been defined to be a satellite group, via the GSATINJE and GSATPROD keywords, then the equivalent modeled group keywords, GCONINJE and GCONPROD in the SCHEDULE section, cannot be used to set the operating conditions for satellite groups, only the GSATINJE and GSATPROD keywords may be used.

Example

The example below is based on a dry gas field, with the main field FLD-A being modeled in the input deck with two groups (FLD-A1 and FLD-A2), combined with two satellite gas fields, FLD-B and FLD-C.

```
-- ==================================================================================
--
-- SCHEDULE SECTION
--
-- ==================================================================================
SCHEDULE
--
--       DEFINE GROUP TREE HIERARCHY
--       LOWER     HIGHER
--       GROUP     GROUP
GRUPTREE
         FLD-A     FIELD                                                          /
         FLD-A1    FLD-A                                                          /
         FLD-A2    FLD-A                                                          /
         FLD-B     FIELD                                                          /
         FLD-C     FIELD                                                          /
/
--
--       GROUP PRODUCTION CONTROLS
--
-- GRUP  CNTL  OIL    WAT    GAS    LIQ    CNTL  GRUP  GUIDE  GUIDE  CNTL
-- NAME  MODE  RATE   RATE   RATE   RATE   OPT   CNTL  RATE   DEF    WAT
GCONPROD
FIELD    GRAT  1*     1*     450E3  1*     1*    1*                           /
FLD-A    FLD   1*     1*     450E3  1*     1*    1*                           /
/
--
--       LOAD INCLUDE FILE - LOAD ALL WELLS AND VFP TABLES
--
INCLUDE
         'FLD-A-P50-WELLS.INC'  /
--
--       SATELLITE GROUP PRODUCTION CONTROLS
--
-- GRUP  OIL    WAT    GAS    RESV   GAS   CALORIFIC
-- NAME  RATE   RATE   RATE   RATE   LIFT  RATE
GCONPROD
FLD-B    10.0   0.00   30E3   1*     1*                  /
FLD-C    20.0   5.50   20E3   0.0    1*                  /
/
--
--       ADVANCE SIMULATION BY REPORTING DATE
--
DATES
         1  FEB   2021  /
/
--
--       SATELLITE GROUP PRODUCTION CONTROLS
--
-- GRUP  OIL    WAT    GAS    RESV   GAS   CALORIFIC
-- NAME  RATE   RATE   RATE   RATE   LIFT  RATE
GCONPROD
FLD-B    10.0   0.00   30E3   0.0    1*                  /
FLD-C    20.0   9.00   20E3   0.0    1*                  /
/
```

Since the field gas rate is set to 450 MMscf/d and the satellite production is 50 MMscf/d, then FLD-A will produce only 400 MMscf/d and not the stipulated 450 MMscf/d on the GCONPROD keyword.

12.3.121 GSEPCOND – ASSIGN GROUP SEPARATORS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The GSEPCOND keyword assigns previously defined separators to a group. Group separators are specified by the SEPVALS keyword in the SCHEDULE section. The facility is used in black-oil modeling to re-scale the PVT data entered via the PROPS section, based on the saturation point oil formation volume factor (Bob) and the initial saturated gas-oil ratio (Rsi) entered on the SEVPALS keyword.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.122 GSSCPTST – PERFORM SUSTAINABLE CAPACITY TEST

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The GSSCPTST keyword instructs the simulator to perform a sustainable capacity test. This causes the model to be saved in it's current state via the RESTART file, and the test performed by running the simulation under the current conditions combine with the parameters on this keyword. After the test is perform, the simulator will restart from the point prior to the test by loading in the RESTART file. This type of testing is normally applied to gas fields for which the gas sales contracts stipulate that the gas sales rates are based on a sustainable capacity rate over a fixed period of time.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.123 GSWINGF – DEFINE GROUP MULTIPLE GAS CONTRACT PARAMETERS

Description

This keyword, GSWINGF, defines the gas contract parameters, swing factor and the monthly seasonal profile factor, for when there are multiple gas contracts being used in the model. The keyword is used with the Gas Field Operations option which is activated by the GASFIELD keyword in the RUNSPEC section. Gas contracts are commonly based on a Daily Contract Quantity ("DCQ") that determines the gas rate that the field should be produced at, which is normally expressed as a multiple of the DCQ, for example 1.33, and is often referred to as the "swing factor". Some gas contracts also define a maximum DCQ ("Max DCQ") and/or a minimum take or pay DCQ ("Min DCQ"), as well as seasonal demand characteristics. For example, gas rates may be set higher in the winter months in order to meet heating demand compared with summer months in colder climates, and the opposite in warmer climates where air conditioning demand is high.

Thus, the DCQ must be calculated first over a contract period, where the contract period is commonly contract years, or in some instances contract quarters. This is performed by the simulator using the current DCQ and checking to see if the (DCQ x Swing Factor) can be satisfied throughout the current contract period, if not the DCQ is re-calculated so that the (DCQ x Swing Factor) condition is satisfied. Once this condition is met, the second and final pass uses the calculated DCQ in conjunction with the monthly scaling profile data to set the monthly gas rate for the field:

$$Q_{month} = DCQ \times SWINGFAC_{month} \tag{12.27}$$

Where:

Qmonth = the monthly gas production target
DCQ = Daily Contract Quantity
$SWINGFAC_{month}$ = monthly rate scaling factor that takes into account seasonal demand, etc.

Here the GSWINGF keyword allows for different gas contract parameters to be assign to different groups and is mutually exclusive to the SWINGFAC keyword in the SCHEDULE section, that sets the gas contract parameters for a single contract at the FIELD group level.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.124 GTADD – ADD A CONSTANT TO A GROUP TARGET OR CONSTRAINT

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, GTADD, adds a numerical constant to a group's target or constraint value. The group must have been initially fully defined using the GCONPROD or GCONPRI keywords for producers or GCONINJE for injectors. Variables not changed by the GTADD keyword remain the same as those previously entered via the group control keywords or previously entered GTADD keywords. See also the GRUPTARG keyword that sets the values for a group's target and constraints and the GTMULT keyword that multiplies a group target or constraint by a constant. All the aforementioned keywords are described in the SCHEDULE section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.125 GTMULT – MULTIPLY GROUP TARGET OR CONSTRAINT BY A CONSTANT

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, GTMULT, multiplies a group's target or constraint value by a numerical constant. The group must have been initially fully defined using the GCONPROD or GCONPRI keywords for producers or GCONINJE for injectors. Variables not changed by the GTMULT keyword remain the same as those previously entered via the group control keywords or previously entered GTMULT keywords. See also the GRUPTARG keyword that sets the values for a group's target and constraints, and the GTADD keyword that adds a constant to a group's target or constraint. All the aforementioned keywords are described in the SCHEDULE section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.126 GUIDECAL – SCALE GUIDE RATES BASED ON GAS CALORIFIC VALUE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The GUIDECAL keyword defines a well or group's guide rate as a function of their calorific values, for when the individual wells and groups are under guide rate control. Group and well guide rates that have not been directly defined are set equal to their production potentials at the start of each time step. In this case the GUIDECAL keyword can be used to specify the coefficients of a function that takes into account the calorific value of the produced gas, effectively scaling the guide rates based on the calorific value of the gas being produced.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.127 GUIDERAT – DEFINE GROUP GUIDE RATE FORMULA

Description

This keyword defines a general formulae used to define a group's and well's guide rate as a function of the their potential. The default behavior, that is when this keyword is not invoked, is to set the target control mode and rate via the GCONCON keyword in the SCHEDULE section. In this case the target rate is distributed between the group's wells that are under group control using a well's guide rate. If a well's guide rate has not been defined, for example by this keyword, then the well potential of the group controlling phase at the beginning of the time step is used. For example, if the group target rate and phase is oil, then the well's under group control will have their oil rates determined by their oil rate potential[313]. The GUIDERAT keyword substitutes the potential calculation with a more general formulae in the aforementioned distribution and allocation of the rates:

$$Phase\,Guide\,Rate = \frac{\left(Potental_{Phase}\right)^{A}}{B + C\left(Potential\,Ratio_{1}\right)^{D} + E\left(Potential\,Ratio_{2}\right)^{F}} \qquad (12.28)$$

Where:

Potential$_{Phase}$ = the potential of the phase,
A to F = constants define on this keyword,
Potential Ratio$_1$ = the potential phase ratio as defined by this keyword,
Potential Ratio$_2$ = the potential phase ratio as defined by this keyword.

The formulae can be used to control high water cut or high GOR wells in an oil field, such as the offending wells are given progressively smaller guide rates as they water out or gas out.

Note that groups can only have potential guide rates if they are subordinate in another group and required to produce a proportion of the superior group's target rate. In this case the GUIDERAT keyword can optionally be applied by setting GUIPHASE variable on the GCONPROD keyword in the SCHEDULE section. Group potentials are the sum of the potentials of their subordinate open wells.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	TSTEP	A real positive value that defines the minimum time interval to re-calculate the guide rates. The guide rates are calculated at the start of a time step and the default value of zero means that the guide rates are calculated for each time step. A non-zero value for TSTEP resets the minimum interval, for example setting TSTEP equal to 30 would mean the guide rates are calculate every 30 days, or to the nearest associated time step. Calculating guide rates every time step may cause issues due to the rate dependent behavior, for example gas cusping or water coning causing the well rates to oscillate. In this case using a non-zero value of TSTEP may eliminate this oscillating behavior.			
		days	days	hours	0.0

[313] *Production and injection potentials are based on rates that are unrestricted. For wells this implies that well potential is calculated based on either the BHP or THP limit, which ever is the more constraining.*

No.	Name	Description			Default
		Field	Metric	Laboratory	
2	PHASE	A defined character string that sets the potential phase guide rate for the group and well, the resulting Phase Guide Rate in equation (12.28). PHASE should be set to one of the following character strings: 1) OIL: is set as the potential phase guide rate and in this case the Potential Ratio1 variable is the water-oil ratio ("WOR") and Potential Ratio2 refers to the gas-oil ratio ("GOR") in equation (12.28). 2) LIQ: is set as the potential phase guide rate and the Potential Ratio1 variable is the water cut ("WCT") and Potential Ratio2 refers to the gas-liquid ratio ("GLR") in equation (12.28). 3) GAS: is set as the potential phase guide rate with the Potential Ratio1 variable being the water cut ("WCT") and Potential Ratio2 referring to the oil-gas ratio ("OGR") in equation (12.28). 4) RES: here the potential phase guide rate is defined as the reservoir fluid volume rate and the Potential Ratio1 variable is the water-oil ratio ("WOR") and Potential Ratio2 refers to the gas-oil ratio ("GOR") in equation (12.28). 5) COMB: this option is uses the linearly combined phase guide rate based on the values entered on the LINCOM keyword in the SCHEDULE section. Here the *Potential Ratio*$_1$ variable is the water divided by linearly combined phase and *Potential Ratio*$_2$ refers to the gas divided by linearly combined phase in equation (12.28). This option is not available in OPM Flow. 6) NONE: the Phase Guide Rate calculation is switch off and the well guides rates revert to the well potentials of their group target phase. For reference, the units for the various options is given below .			None
		WOR: dimensionless WCT: dimensionless WGR: stb/Mscf	dimensionless dimensionless dimensionless	dimensionless dimensionless dimensionless	
		GOR: Mscf/stb GLR: Mscf/stb OGR: stb/Mscf	dimensionless dimensionless dimensionless	dimensionless dimensionless dimensionless	
3	A	A real value greater than or equal to -3 and less than or equal to 3, that defines coefficient A in equation (12.28).			0.0
4	B	B is a real positive value that defines coefficient B in equation (12.28).			0.0
5	C	C is a real value that defines coefficient C in equation (12.28).			0.0
6	D	D is a real value greater than or equal to -3 and less than or equal to 3, that defines coefficient A in equation (12.28).			0.0
7	E	E is a real value that defines coefficient E in equation (12.28).			0.0
8	F	D is a real value greater than or equal to -3 and less than or equal to 3, that defines coefficient A in equation (12.28).			0.0

No.	Name	Description			Default
		Field	Metric	Laboratory	
9	GROPT01	A defined character string that determines if calculated phase guide rates should be allowed to increase (YES) or not (NO), and should be set to one of the following: 1) NO: The phase guide rates calculated from equation (12.28) are not allowed to increase above the current value, and will be reset to the current value if the calculated value does exceed the current value. 2) YES: The phase guide rates calculated from equation (12.28) are allowed to increase at each calculation. This may increase the propensity for oscillations in the rates if if the water cut and GOR are rate dependent. Note only the default value is currently supported by OPM Flow.			YES
10	GROPT02	A real positive value greater than or equal to zero and less than or equal to one that "dampens" the calculated phase guide rate based on the following formula: $(Phase\ Guide\ Rate)_t^{new} = GROPT02 \times (Phase\ Guide\ Rate)_t + (1 - GROPT02) \times (Phase\ Guide\ Rate)_{(t-1)}$ The option is intended to have a similar effect as the GROPT01 NO option to reduce oscillations as a result of either the water cut or GOR being rate dependent. Values approaching one allows the calculated phase guide rates to change instantaneously with the phase potentials, whereas values approaching zero dampen the potential guide rates towards the previously calculated values, thereby reducing the potential for oscillating behavior.			1.0
11	GROPT03	A defined character string that determines if "free" gas potential rates for the Potential Ratio2 variable in equation (12.28) should be used (YES), or if "free and associated" gas should be used (NO), and should be set to one of the following: 1) NO: Use "free and associated" gas, that is total gas in the Potential Ratio2 variable. 2) YES: Only utilize "free" gas in the Potential Ratio2 variable.			NO
12	GROPT04	A real positive value that sets the minimum potential guide rate. If the calculated potential guide is below this value it will be reset to GROPT04. The option is meant to avoid groups and wells being ignored due to the calculated potential guide rates being minuscule.			1.0 x10-6
Notes: 1) The keyword is terminated by a "/".					

Table 12.49: GUIDERAT Keyword Description

Note that the GUIDERAT keyword only applies to production groups and wells, injection groups and wells are still control by their potential guide rates.

Finally, as mentioned previously, if the GUIDERAT or WGRUPCON keywords are not present in the input deck then the group and well potential guide rates will be calculated using the well's potential rates. The WGRUPCON keyword in the SCHEDULE section can be used to set a constant potential guide rate for a well.

Note

GUIDERAT can be used to penalize wells producing excessive water by utilizing the C and D coefficients in equation (12.28), and to discriminate against wells that are gassing out by setting the E and F coefficients.

Note that the value range through which Potential Ratio1 and Potential Ratio2 vary is variable. For example, if Potential Ratio1 is water cut, then the value should be between zero and one, whereas for the water-oil ratio the value can vary between zero and infinity. The same applies to the units of Potential Ratio2 which are on dependent on if the GOR, GLR or OGR ratio is used in the calculation.

One can use the C and E coefficients to scale these terms to the required relative magnitudes in the denominator and the D and F powers to influence how quickly the penalty increases with increasing water and gas fractions. High positive value for D and F coefficients will make production fall off rapidly as the water or gas fraction increases, while a negative values will favor producing these type of wells.

Note that the B coefficient should always be positive to prevent the denominator's going to zero.

Finally, if one wishes each well to produce in proportion to its potential when the water fraction and gas fraction are equal (the usual case), then the A coefficient should be set to one.

Examples

The first example sets the guide phase to oil and the resulting Phase Guide Rate based on oil potential based on setting the A and B coefficients to to one, that is:

$$Phase\,Guide\,Rate = \frac{(Potental_{Phase})^{A}}{B + C(Potential\,Ratio_{1})^{D} + E(Potential\,Ratio_{2})^{F}} = \frac{Oil\,Potential^{1.0}}{1.0} \tag{12.29}$$

with all the other parameters defaulted, except for the minimum time interval to re-calculate the guide rates which is set to 30 days..

```
--
--        SETS GUIDE RATES FOR GROUPS AND WELLS UNDER GUIDE RATE CONTROL
--
--        TIME  GUIDE  A      B     C     D     E     F     INCR   DAMP   FREE
--        STEP  PHASE  POW    CON   CON   POW   CON   POW   OPTN   OPTN   GAS
GUIDERAT
          30    'OIL'  1.0    1.0    1*   1*    1*    1*    1*     1*     1*      /
```

The next example sets the Phase Guide Rate to the reservoir fluid volume rate, with preference given to low GOR wells and with high GOR wells penalized, based on setting A and B to one, C and D to zero, E equal to 10 and F equal to two, that is:

$$Phase\,Guide\,Rate = \frac{\left(Potental_{Phase}\right)^{A}}{B + C\left(Potential\,Ratio_{1}\right)^{D} + E\left(Potential\,Ratio_{2}\right)^{F}} \tag{12.30}$$

$$= \frac{Reservoir\ \ Fluid\ \ Volume\ \ Potential^{1.0}}{1.0 + 10\times\left(GOR\right)^{2}}$$

with all the other parameters defaulted.

```
--
--       SETS GUIDE RATES FOR GROUPS AND WELLS UNDER GUIDE RATE CONTROL
--
--       TIME  GUIDE  A      B     C     D     E    F     INCR   DAMP   FREE
--       STEP  PHASE  POW    CON   CON   POW   CON  POW   OPTN   OPTN   GAS
GUIDERAT
         1*    'OIL'  1.0    1.0    1*   1*    10   2     1*     1*     1*      /
```

The GUIDERAT keyword is very flexible but can also lead to unexpected results, thus it is probably useful to perform some manual calculations outside of the simulator before implementing the selected scheme in the input deck.

12.3.128 GUPFREQ – INSTANTANEOUS GRADIENT OPTION UPDATE FREQUENCY

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The GUPFREQ keyword sets the update frequency of the Instantaneous Gradient option for when this option has been activated by the GDIMS keyword in the RUNSPEC section. The Instantaneous Gradient option calculates derivatives of solution quantities at the current time step with respect to variations in the variables at the current time step. This is different to Gradient option that calculates the derivatives of solution quantities at the current time step with respect to variations in the variables at the initial time step, that is a time equal to zero. Consequently, the Instantaneous Gradient option can be switched on and off by the GUPFREQ keyword in the SCHEDULE section, whereas the Gradient option cannot.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.129 GWRTWCV – INSTANTANEOUS GRADIENT OPTION WELL VARIABLES

Description

The GWRTWCV keyword defines the wells and instantaneous gradient parameters to be calculated and exported as SUMMARY vectors to the summary file, for when the Instantaneous Gradient option has been activated by the GDIMS keyword in the RUNSPEC section. The Instantaneous Gradient option calculates derivatives of solution quantities at the current time step with respect to variations in the variables at the current time step. This is different to Gradient option that calculates the derivatives of solution quantities at the current time step with respect to variations in the variables at the initial time step, that is a time equal to zero. Consequently, the Instantaneous Gradient option can be switched on and off by the GUPFREQ keyword in the SCHEDULE section, whereas the Gradient option cannot.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.130 HMWPIMLT – HISTORY MATCH WELL PRODUCTIVITY INDEX PARAMETERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, HMWPIMLT, defines the history match gradient parameters for well productiviity indices, for when the History Match Gradient option has been activated by the HMDIMS keyword in the RUNSPEC section. Wells must be specified using the WELPSECS keyword in the SCHEDULE section and their connections defined by the COMPDAT and/or COMPDATL keywords, also in the SCHEDULE section

See also the HMDIMS keyword in the RUNSPEC section that specifies the dimensions for the gradient option, including the maximum number of gradient wellss that can be used with the History Match Gradient option.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.131 INCLUDE – LOAD ANOTHER DATA FILE AT THE CURRENT POSITION

The INCLUDE keyword informs OPM Flow to continue reading input data from the specified INCLUDE file. When the end of the INCLUDE file is reached, or the ENDINC keyword is encountered in the included file, input data is read from the next keyword in the current file.

See *INCLUDE – Load Another Data File at the Current Position* in the GLOBAL section for a full description.

12.3.132 LGRFREE – ACTIVATE LOCAL GRID REFINEMENT INDEPENDENT TIME STEPS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The LGRFREE keyword activates the Local Grid Refinement ("LGR") Independent Time Step option that allows the LGR to have solution time steps independent of the host grid for the stated LGR, and for when LGRs have been declared by the LGR keyword in the RUNSPEC section, and defined by the CARFIN (Cartesian LGR grid) or RADIN/RADIN4 (radial LGR grid) keywords in the GRID section. LGR independent solution time stepping can be deactivated by the LGRLOCK keyword in the SCHEDULE section.

Currently, OPM Flow does not support the local grid refinement feature and therefore this keyword is ignored by the simulator.

No.	Name	Description	Default
1	LGRNAME	A character string of up to eight characters in length that defines the LGR name for which independent solution time stepping is to be activated. The LGR must have been previously defined by the CARFIN (Cartesian LGR grid) or RADIN/RADIN4 (radial LGR grid) keywords in the GRID section.	None

Notes:

1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".

Table 12.50: LGRFREE Keyword Description

Example

The example below defines three oil LGRs(LGR-OP01,-OP02, and -OP03) and all the gas well LGRs (LGR-GP*) that should use independent solution time steps.

```
--
--        ACTIVATE LOCAL GRID REFINEMENT INDEPENDENT TIME STEPS
--
--        LGRNAME
LGRFREE
          LGR-OP01                                                    /
          LGR-OP02                                                    /
          LGR-OP03                                                    /
          LGR-GP*                                                     /
/
```

12.3.133 LGRLOCK – DEACTIVATE LOCAL GRID REFINEMENT INDEPENDENT TIME STEPS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The LGRLOCK keyword deactivates the Local Grid Refinement ("LGR") Independent Time Step option that allows the LGR to have solution time steps independent of the host grid for the stated LGR, that is the LGR will now follow the global grid solution time steps. LGRs must have declared by the LGR keyword in the RUNSPEC section, and defined by the CARFIN (Cartesian LGR grid) or RADIN/RADIN4 (radial LGR grid) keywords in the GRID section. LGR independent solution time stepping can be activated by the LGRFREE keyword in the SCHEDULE section.

Currently, OPM Flow does not support the local grid refinement feature and therefore this keyword is ignored by the simulator.

No.	Name	Description	Default
1	LGRNAME	A character string of up to eight characters in length that defines the LGR name for which independent solution time stepping is to be deactivated. The LGR must have been previously defined by the CARFIN (Cartesian LGR grid) or RADIN/RADIN4 (radial LGR grid) keywords in the GRID section.	None

Notes:

1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".

Table 12.51: LGRLOCK Keyword Description

Example

The example below defines three oil LGRs(LGR-OP01,-OP02, and -OP03) and all the gas well LGRs (LGR-GP*) that should have their independent solution time steps deactivated.

```
--
--        DEACTIVATE LOCAL GRID REFINEMENT INDEPENDENT TIME STEPS
--
--        LGRNAME
LGRLOCK
          LGR-OP01                                                    /
          LGR-OP02                                                    /
          LGR-OP03                                                    /
          LGR-GP*                                                     /
/
```

12.3.134 LGROFF – DEACTIVATE A LOCAL GRID REFINEMENT

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The LGROFF keyword deactivates a stated Local Grid Refinement ("LGR") and optionally sets the minimum number of wells below which the LGR will be automatically deactivated. LGRs must have declared by the LGR keyword in the RUNSPEC section, and defined by the CARFIN (Cartesian LGR grid) or RADIN/RADIN4 (radial LGR grid) keywords in the GRID section. LGRs can subsequently be activated by the LGRON keyword in the SCHEDULE section.

Currently, OPM Flow does not support the local grid refinement feature and therefore this keyword is ignored by the simulator.

No.	Name	Description	Default
1	LGRNAME	A character string of up to eight characters in length that defines the LGR name for which the LGR is being deactivated. The LGR must have been previously defined by the CARFIN (Cartesian LGR grid) or RADIN/RADIN4 (radial LGR grid) keywords in the GRID section.	None
2	MNWELLS	A positive integer greater than or equal to zero that defines the minimum number of active wells, below which the LGR will be automatically deactivated. The default value of zero implies that there is no limit to the number of wells and results in the LGR being unconditionally being deactivated.	0

Notes:

1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".

Table 12.52: LGROFF Keyword Description

Example

The example below unconditionally deactivates LGR-OP01, and sets the minimum number of active wells for deactivating LGR-OP02 and LGR-OP03 to one. For all the gas well LGRs (LGR-GP*) the minimum number of wells for deactivation is set to two.

```
--
--        DEACTIVATE LOCAL GRID REFINEMENTS
--
--        LGRNAME    MNWELLS
LGROFF
          LGR-OP01                                                    /
          LGR-OP02   1                                                /
          LGR-OP03   1                                                /
          LGR-GP*    2                                                /
/
```

12.3.135 LGRON – ACTIVATE A LOCAL GRID REFINEMENT

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The LGRON keyword activates a stated Local Grid Refinement ("LGR") and optionally sets the minimum number of wells above which the LGR will remain active. LGRs must have declared by the LGR keyword in the RUNSPEC section, and defined by the CARFIN (Cartesian LGR grid) or RADIN/RADIN4 (radial LGR grid) keywords in the GRID section. LGRs can subsequently be deactivated by the LGROFF keyword in the SCHEDULE section.

Currently, OPM Flow does not support the local grid refinement feature and therefore this keyword is ignored by the simulator.

No.	Name	Description	Default
1	LGRNAME	A character string of up to eight characters in length that defines the LGR name for which the LGR is being activated. The LGR must have been previously defined by the CARFIN (Cartesian LGR grid) or RADIN/RADIN4 (radial LGR grid) keywords in the GRID section.	None
2	MNWELLS	A positive integer greater than or equal to zero that defines the minimum number of active wells, below which the LGR will be automatically deactivated. The default value of zero implies that there is no limit to the number of wells and results in the LGR being unconditionally being activated.	0
Notes: 1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".			

Table 12.53: LGRON Keyword Description

Example

The example below unconditionally activates LGR-OP01, and sets the minimum number of active wells for activating LGR-OP02 and LGR-OP03 to one. For all the gas well LGRs (LGR-GP*) the minimum number of wells for activating these LGRs is set to two.

```
--
--       ACTIVATE LOCAL GRID REFINEMENTS
--
--       LGRNAME   MNWELLS
LGRON
         LGR-OP01                                                    /
         LGR-OP02  1                                                 /
         LGR-OP03  1                                                 /
         LGR-GP*   2                                                 /
/
```

12.3.136 LIFTOPT – ACTIVATE GAS LIFT OPTIMIZATION

Description

The LIFTOPT keyword actives the gas lift optimization option and defines the gas lift gas increment size, the minimum incremental oil improvement, as well as the timing of the calculations. Note that the LIFTOPT keyword should precede any GLIFTOPT and WLIFTOPT keywords in the SCHEDULE section in order to activate the gas lift optimization facility.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	GASLIFT	A real positive number that defines the gas lift gas size increment that is used to increase the gas lift size quantity in steps. For example, if GASLIFT is set to 0.5 MMscf/d then gas lift gas will be allocated in step of 0.5 MMscf/d to each well during the optimization process. A zero or negative value switches off gas lift optimization.			
		Mscf/d	sm^3/day	scc/hour	None
2	MINOIL	MINOIL is a real positive value that defines the minimum increase in oil rate for a given quantity of gas lift gas, for when gas lift gas should be applied to a well. Additional GASLIFT will only be assigned to a well if: $\frac{\Delta Q_{Oil} \times OPTWGT}{GASLIFT} > MINOIL$ Where ΔQ_{Oil} is the incremental oil and OPTWGT is the well's weighting factor defined by the OPTWGT variable on the WLIFTOPT keyword in the SCHEDULE section.			
		stb/Mscf	sm^3/sm^3	scc/scc	None
3	TSTEP	TSTEP is a real positive value that defines the frequency of the gas lift optimization calculations, for example setting TSTEP equal to 30 days would result in the gas lift optimization calculation being performed approximately every 30 days. The default value of zero will result in the calculations being performed every time step. Note if the group or well is part of a production network then gas lift optimization is performed at the same time as the network is being balance, that is this parameter is ignored in this scenario. See the NETBALAN keyword in the SCHEDULE section to set the network balancing frequency in this case.			
		days	days	hours	0.0

No.	Name	Description			Default
		Field	Metric	Laboratory	
4	OPTLIFT	A defined character string that determines if the gas lift optimization iterations should be performed for the same number of Newton iterations within a time step as used to update well targets, or to just use the first Newton iteration only. The NUPCOL keyword in the RUNSPEC section determines the number of Newton iterations used to update well targets during a time step. OPTLIFT should be set to one of the following: 1) NO: In this case the gas lift optimization is only performed for the first Newton iteration for a time step and the distributed gas lift gas is then held constant for the groups and wells through resultant Newton iterations during the time step. This leads to better numerical performance, but may lead to the targets and constraints not being exactly satisfied if the reservoir conditions change during the time step calculations. 2) YES: Sets the number gas lift optimization iterations to the same number of Newton iterations within a time step as used to update well targets, as per the NUPCOL keyword in the RUNSPEC section. This results in greater target and constraint accuracy even if the reservoir conditions change during the time step calculations, but at the expense of numerical performance. Similar to the NO option, after the NUPCOL Newton iteration during the time step, the distributed gas lift gas is then held constant for the groups and wells in subsequent Newton iterations.			
Notes: 1) The keyword is terminated by a "/".					

Table 12.54: LIFTOPT Keyword Description

See also the GLIFTOPT keyword to define the group gas lift optimization controls and the WLIFTOPT keyword to define the wells under gas lift optimization control, both keywords are described in the SCHEDULE section. The NUPCOL keyword in the RUNSPEC section that determines the number of Newton iterations used to update well targets and gas lift optimization calculations during a time step may also be of interest.

Example

The following example activates gas lift optimization for the field and defines the optimization parameters.

```
--
--       ACTIVATE GAS LIFT OPTIMIZATION AND PARAMETERS
--
--       INCR    INCR    TSTEP     NEWTON
--       GAS     OIL     INTVAL    OPTN
LIFTOPT
        12.5E3   5E-3    0.0       YES     /
```

Here the maximum incremental gas lift gas quantity is set to 12.5 x 103 m3, the minimum incremental oil gain per m3 of gas lift gas is set to 5.0 x 10-3 m3, the time step interval is set to zero to perform the gas optimization every time step, and finally the gas lift optimization will be performed NUPCOL Newton iterations for the time step.

12.3.137 LINCOM – DEFINE LINEAR COMBINATION COEFFICIENTS

Description

The LINCOM keyword defines the oil, gas and water coefficients for the Linear Combination facility which allows for a linear combination of the aforementioned phase rates and volumes to be used as targets and constraints in controlling group and well production and injection data. See also the LCUNIT in the PROPS section that defines the units for linear combination equation.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.138 MATCORR – ACTIVATE THE MATERIAL BALANCE CORRECTION OPTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The MATCORR keyword activates the Material Balance Correction option used to adjust the accumulated material balance error in the simulation.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.139 MESSAGE – OUTPUT USER MESSAGE

The MESSAGE keyword outputs a user message to the terminal, as well as to the print (*.PRT) and debug (*.DBG) files. Note this is different to the MESSAGES keyword, that defines OPM Flows message print limits and stop limits generated by the simulator.

See *MESSAGE – Output User Message* in the GLOBAL section for a full description.

12.3.140 MESSAGES – DEFINE MESSAGE PRINT LIMITS AND STOP LIMITS

The MESSAGES keyword defines the print and stops levels for various messages. The "print limits" set the maximum number of messages that will be printed, after which no more messages will be printed and the "stop limits" terminate the run when these limits are exceeded. There are six levels of message that increase in severity from informative all the way to programming errors, as outlined in Error: Reference source not found.

See *MESSAGES – Define Message Print Limits and Stop Limits* in the GLOBAL section for a full description.

12.3.141 MESSOPTS – RESET SEVERITY LEVEL FOR FORCED TIME STEPS

Description

This keyword, MESSOPTS, resets the severity level for time steps that are forced to be accepted by the simulator. The normal severity level for this type of simulator generated message is PROBLEM and this can result in the run stopping depending on the parameters entered on the MESSAGES keyword. MESSOPTS can be used to reset the severity level to MESSAGE, COMMENT, WARNING, or PROBLEM; for example, to avoid the run terminating due to too many PROBLEM messages.

Note that the MESSAGES keyword is a global keyword can therefore be used in any section; however, only the last instance of the keywords is active. The MESSOPTS keyword can only be used in the SCHEDULE section but can be used multiple times to change the severity level for forced time steps. Again, only the last occurrence of the keyword is active.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

12.3.142 MULSGGD – MULTIPLY MATRIX-FRACTURE COUPLING FOR OIL-GAS GRAVITY DRAINAGE FOR ALL CELLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, MULSGGD, defines a constant multiplier to modify the matrix-fracture coupling transmissibility for dual porosity and dual permeability models, for when the alternative matrix-fracture coupling transmissibilities for oil-gas gravity drainage has been selected. The alternative matrix-fracture coupling transmissibilities for oil-gas gravity drainage option is activated via the SIGMAGD or SIGMAGDV keywords in the GRID section, and the dual porosity or dual permeability models are activated by the DUALPORO or DUALPERM keywords in the RUNSPEC section, respectively.

This keyword applies the multiplier for all cells in model; whereas, the MULSGGDV keyword in the SCHEDULE section applies the multiplier to individual grid blocks.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.143 MULSGGDV – MULTIPLY MATRIX-FRACTURE COUPLING FOR OIL-GAS GRAVITY DRAINAGE FOR INDIVIDUAL CELLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, MULSGGDV, defines a constant multiplier to modify the matrix-fracture coupling transmissibility for dual porosity and dual permeability models, for when the alternative matrix-fracture coupling transmissibilities for oil-gas gravity drainage has been selected. The alternative matrix-fracture coupling transmissibilities for oil-gas gravity drainage option is activated via the SIGMAGD or SIGMAGDV keywords in the GRID section, and the dual porosity or dual permeability models are activated by the DUALPORO or DUALPERM keywords in the RUNSPEC section, respectively.

This keyword applies the multiplier for individual cells in model; whereas, the MULSGGD keyword in the SCHEDULE section applies the multiplier to all grid blocks.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.144 MULTFLT – MULTIPLY THE TRANSMISSIBILITY OF A DEFINED FAULT BY A CONSTANT

MULTFLT enables the transmissibilities across defined faults, as declared by the FAULTS keyword, to be modified. They keyword allows for the re-scaling of the existing fault transmissibilities calculated by OPM Flow, for example setting a fault to be completely sealing by setting the multiplier to zero..

See *MULTFLT – Multiply the Transmissibility of a Defined Fault by a Constant* in the GRID section for a full description.

12.3.145 MULTPV – MULTIPLY CELL PORE VOLUMES BY A CONSTANT

MULTPV multiples the pore volumes of a cell by a real positive constant for all the cells in the model via an array. An alternative to defining the complete array is to use the BOX keyword to define an area of the grid and then use the MULTPV keyword to set the multipliers just for the area defined by the BOX keyword (see the example).

See *MULTPV – Multiply Cell Pore Volumes by a Constant*in the GRID section for a full description.

12.3.146 MULTR - MULTIPLY CELL TRANSMISSIBILITY IN THE +R DIRECTION

MULTR multiples the transmissibility between two cell faces in the +R direction for all the cells in the model via an array, that is the keyword sets the transmissibility multiplier of block (I, J, K) between the cells (I, J, K) and (I+I, J, K). The keyword should only be used with radial grids, as declared by the RADIAL keyword in the RUNSPEC section.

See *MULTR - Multiply Cell Transmissibility in the +R Direction* in the GRID section for a full description.

12.3.147 MULTR- - MULTIPLY CELL TRANSMISSIBILITY IN THE -R DIRECTION

MULTX- multiples the transmissibility between two cell faces in the -X direction for all the cells in the model via an array, that is the keyword sets the transmissibility multiplier of block (I, J, K) between the cells (I-1, J, K) and (I, J, K). The keyword should only be used with radial grids, as declared by the RADIAL keyword in the RUNSPEC section.

See *MULTR- - Multiply Cell Transmissibility in the -R Direction* in the GRID section for a full description.

12.3.148 MULTREGT – MULTIPLY TRANSMISSIBILITIES BETWEEN REGIONS

The MULTREGT keyword multiplies the transmissibility between two regions by a constant. The region number array can be FLUXNUM, MULTNUM or OPERNUM and these arrays must be defined and be available before the MULTREGT keyword is read by the simulator. The constant should be a real number.

See *MULTREGT – Multiply Transmissibilities Between Regions* in the GRID section for a full description.

12.3.149 MULTSIG – MULTIPLY MATRIX-FRACTURE COUPLING FOR ALL CELLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, MULTSIG, defines a constant multiplier to modify the matrix-fracture coupling transmissibility for dual porosity and dual permeability models, for when the matrix-fracture coupling transmissibilities have been specified via the SIGMAGD or SIGMAGDV keywords in the GRID section, and the dual porosity or dual permeability models have been activated by the DUALPORO or DUALPERM keywords in the RUNSPEC section, respectively.

This keyword applies the multiplier for all cells in model; whereas, the MULTSIGV keyword in the SCHEDULE section applies the multiplier to individual grid blocks.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.150 MULTSIGV – MULTIPLY MATRIX-FRACTURE COUPLING FOR INDIVIDUAL CELLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, MULTSIGV, defines a constant multiplier to modify the matrix-fracture coupling transmissibility for dual porosity and dual permeability models, for when the matrix-fracture coupling transmissibilities have been specified via the SIGMAGD or SIGMAGDV keywords in the GRID section, and the dual porosity or dual permeability models have activated by the DUALPORO or DUALPERM keywords in the RUNSPEC section, respectively.

This keyword applies the multiplier for individual cells in model; whereas, the MULTSIG keyword in the SCHEDULE section applies the multiplier to all grid blocks.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.151 MULTTHT - MULTIPLY CELL TRANSMISSIBILITY IN THE +THETA DIRECTION

MULTTHT multiples the transmissibility between two cell faces in the +Theta direction for all the cells in the model via an array, that is the keyword sets the transmissibility multiplier of block (I, J, K) between the cells (I, J, K) and (I, J+1, K). The keyword should only be used with radial grids, as declared by the RADIAL keyword in the RUNSPEC section.

See *MULTTHT - Multiply Cell Transmissibility in the +Theta Direction* in the GRID section for a full description.

12.3.152 MULTTHT- - MULTIPLY CELL TRANSMISSIBILITY IN THE -THETA DIRECTION

MULTTHT- multiples the transmissibility between two cell faces in the -Theta direction for all the cells in the model via an array, that is the keyword sets the transmissibility multiplier of block (I, J, K) between the cells (I, J-1, K) and (I, J, K). The keyword should only be used with radial grids, as declared by the RADIAL keyword in the RUNSPEC section.

See *MULTTHT- - Multiply Cell Transmissibility in the -Theta Direction* in the GRID section for a full description.

12.3.153 MULTX - MULTIPLY CELL TRANSMISSIBILITY IN THE +X DIRECTION

MULTX multiples the transmissibility between two cell faces in the +X direction for all the cells in the model via an array, that is the keyword sets the transmissibility multiplier of block (I, J, K) between the cells (I, J, K) and (I+1, J, K).

See *MULTX - Multiply Cell Transmissibility in the +X Direction* in the GRID section for a full description.

12.3.154 MULTX- - MULTIPLY CELL TRANSMISSIBILITY IN THE -X DIRECTION

MULTX- multiples the transmissibility between two cell faces in the -X direction for all the cells in the model via an array, that is the keyword sets the transmissibility multiplier of block (I, J, K) between the cells (I-1, J, K) and (I, J, K).

See *MULTX- - Multiply Cell Transmissibility in the -X Direction* in the GRID section for a full description.

12.3.155 MULTY - MULTIPLY CELL TRANSMISSIBILITY IN THE +Y DIRECTION

MULTY multiples the transmissibility between two cell faces in the +Y direction for all the cells in the model via an array, that is the keyword sets the transmissibility multiplier of block (I, J, K) between the cells (I, J, K) and (I, J+1, K).

See *MULTY - Multiply Cell Transmissibility in the +Y Direction* in the GRID section for a full description.

12.3.156 MULTY- - MULTIPLY CELL TRANSMISSIBILITY IN THE -Y DIRECTION

MULTY- multiples the transmissibility between two cell faces in the -Y direction for all the cells in the model via an array, that is the keyword sets the transmissibility multiplier of block (I, J, K) between the cells (I, J-1, K) and (I, J, K).

See *MULTY- - Multiply Cell Transmissibility in the -Y Direction* in the GRID section for a full description.

12.3.157 MULTZ - MULTIPLY CELL TRANSMISSIBILITY IN THE +Z DIRECTION

MULTZ multiples the transmissibility between two cell faces in the +Z direction for all the cells in the model via an array, that is the keyword sets the transmissibility multiplier of block (I, J, K) between the cells (I, J, K) and (I, J, K+1).

See *MULTZ - Multiply Cell Transmissibility in the +Z Direction* in the GRID section for a full description.

12.3.158 MULTZ- - MULTIPLY CELL TRANSMISSIBILITY IN THE -Z DIRECTION

MULTZ- multiples the transmissibility between two cell faces in the -Z direction for all the cells in the model via an array, that is the keyword sets the transmissibility multiplier of block (I, J, K) between the cells (I, J, K-1) and (I, J, K).

See *MULTZ- - Multiply Cell Transmissibility in the -Z Direction* in the GRID section for a full description.

12.3.159 NCONSUMP - NODE GAS CONSUMPTION (EXTENDED NETWORK)

Description

The NCONSUMP keyword defines an extended network node's gas consumption rate, for when the Extended Network option has been activated by the NETWORK keyword in the RUNSPEC section. The keyword can also be used to attribute the gas consumption to a previously defined group. See also the GCONSUMP keyword in the SCHEDULE section that overs more flexibility and can also be used with the Extended Network option.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.160 NEFAC - NODE EFFICIENCY FACTORS (EXTENDED NETWORK)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The NEFAC keyword defines an extended network node's efficiency factor, for when the Extended Network option has been activated by the NETWORK keyword in the RUNSPEC section. See also the GEFAC keyword in the SCHEDULE section that can also be used with the Extended Network option.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.161 NETBALAN – NETWORK BALANCING PARAMETERS

Description

NETBALAN keyword causes the simulator to perform a network balancing operation at the next time step. In addition, the keyword defines the network balancing parameters used to control how network balancing is perform on a network, as well as the frequency of the network balancing calculations. If any of the parameters on the keyword are defaulted, then either the previously entered values are used, or if there are no previously entered values, then the keyword default values are employed.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	NTSTEP	NSTEP is a real value that defines the criteria for the network balancing interval, used to define the frequency of the network balancing algorithm. NSTEP may be a negative number, a value of zero, or a positive number, as described below: 1) Negative Value: If NSTEP is any negative value then the time stepping interval for the network balancing is based on the number of Newton iterations used in updating the well targets, as set by the NUPCOL keyword in the RUNSPEC section, or the NUPCOL parameter on the GCONTOL keyword in the SCHEDULE section. If NUPCOL is defined on both keywords, then the GCONTOL value takes precedence. 2) Thus, any negative value of NSTEP means the network will be balance for the first NUPCOL Newton iterations for each time step. 3) Consequently, if this option is used it may result in some oscillations as the THP of the wells are updated at each Newton network re-balance computation. 4) Zero Value: Here the network is balanced at the beginning of every time step, and should result in small network balancing errors for the time step, provided of course the reservoir conditions are fairly constant over the time step. This is the default value. 5) Positive Value: In this case NSTEP is a time interval between the previous network balance and the next schedule network balancing. For example, if NSTEP is set equal to 91.25 days (365/4), then network balancing will occur every quarter. Again, the balancing occurs at the beginning of the time step. Note only negative or zero values are currently supported.			
		days or dimensionless	days or dimensionless	hours or dimensionless	0.0
2	NCNV	NCNV is a real positive value that defines the nodal pressure convergence variance, used to define if convergence has been satisfied.			
		psia 1.45	barsa 0.10	atma 0.09869	Defined

No.	Name	Description			Default
		Field	Metric	Laboratory	
3	NMXITER	A positive integer value that defines the maximum number of network balancing operations that may be performed during a network balancing computation.			
		dimensionless	dimensionless	dimensionless	10
4	GRPCNV	GRPCNV is a real positive value less than one that sets the group tolerance criteria used to define if convergence has been satisfied for: 1) Wells in subsea completion manifold groups under rate control. In this case GRPCNV is applied to the THP values of the wells within the group. In the field, these type of wells must flow against a common manifold pressure, that is all the wells must flow at approximately the same THP, for a given group's flow rate. 2) Automatic chokes for group's under rate control. Here, GRPCNV is applied to the pressure drop across the automatic chokes. In both cases GRPCNV is an acceptable fraction of the group's rate target. Thus, the default value 0.01 means that network balance calculated rate must be within 0.01 (or 1.0%) of the groups' production target. Note that this criteria may not be satisfied if the number of Newton iterations used in updating the well targets, as set by the NUPCOL keyword in the RUNSPEC section, or the NUPCOL parameter on the GCONTOL keyword in the SCHEDULE section, is exceeded. Group convergence criteria can also be set by the GRPCNV parameter on the GCONTOL keyword in the SCHEDULE section. If both values of GRPCNV have been entered, then the minimum of the two is used.			
		dimensionless	dimensionless	dimensionless	0.01
5	THPMXITE	A positive integer value that defines the maximum number of well THP iterations for wells in subsea completion.anifold groups under rate control. There is no equivalent iteration maximum for automatic chokes in groups under rate control, as the pressure drop across the chokes is automatically calculated in the network balancing calculation. Thus, NMXITER is used for these calculations.			
		dimensionless	dimensionless	dimensionless	10
6	NTRGERR	A real positive value that stipulates the maximum target branch error in a network balance calculation, at the end of the time step. NTRGERR is compared with the branch error (Error), where Error is the difference between the pressure drop along the branch from the previous network balance and the current calculation, again, at the end of the time step. Subject to existing targets and constraints, the simulator will attempt to adjust the time step size in order to roughly match NTRGERR. Note also that NTRGERR should be a value that is sufficient to allow convergence as defined by NCNV and GRPNCN parameters. The default value of 1.0 x 1020 means that this parameter has no effect in the selection of the time step size.			
		psia	barsa	atma	1.0 x 1020

No.	Name	Description			Default
		Field	Metric	Laboratory	
7	NMAXERR	A real positive value that stipulates the maximum branch error in a network balance calculation, at the end of the time step. Again, NMAXERR is compared against the branch error (Error), where Error is the difference between the pressure drop along the branch from the previous network balance and the current calculation, at the end of the time step. If Error is greater than NMAXERR then the simulator will enforce a time step chop. Time step chops are computationally expensive and should therefore be minimized. Hence, care should be used in setting this value, which should be significantly greater than NTRGERR. The default value of 1.0 x 1020 means that this parameter has no effect in the selection of the time step size.			
		psia	barsa	atma	1.0 x 1020
8	NTSMIN	NTSMIN is a real positive value that sets the minimum time step size for when NTRGERR and NMAXERR have values. Large network balancing errors (Error) can take place for various reasons, a common occurrence is for wells producing near their operating limit which may shut in during a network balance operation. If NTRGERR and NMAXERR have values, then this will result in the simulator enforcing time step chops to perhaps very small time steps. NTSMIN can therefore be used to set the minimum time step size under, and only, these circumstances. The default value, TSMINZ, is taken from TSMINZ parameter on the TUNING keyword in the SCHEDULE section.			
		days	days	hour	TSMINZ
Notes: 1) The keyword is terminated by a "/".					

Table 12.55: NETBALAN Keyword Description

Examples

The first example sets network balancing to occur at the beginning of each time step using a 1.0 psia convergence criteria, and a maximum of 10 iterations. All the other parameters are defaulted.

```
--
--        NETWORK BALANCE CONTROL OPTIONS
--
--        BAL    CONV   MAX    WTHP   WTHP   ERR    ERR    TSTEP
--        INTV   TOL    ITRS   TOL    ITS    TARG   MAX    MIN
NETBALAN
          0.0    1.0    10     1*     1*     1*     1*     1*       /
```

The next example sets network balancing to occur for the first NUPCOL Newton iterations for each time step. All the other parameters are defaulted.

```
--
--       NETWORK BALANCE CONTROL OPTIONS
--
--       BAL    CONV   MAX    WTHP   WTHP   ERR    ERR   TSTEP
--       INTV   TOL    ITRS   TOL    ITS    TARG   MAX   MIN
NETBALAN
         -1     1*     1*     1*     1*     1*     1*    1*       /
```

12.3.162 NETCOMPA – DEFINE AUTOMATIC COMPRESSORS (EXTENDED NETWORK)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The NETCOMPA keyword defines automatic compressors in an extended network, for when the Extended Network option has been activated by the NETWORK keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.163 NEXT – MAXIMUM NEXT TIME STEP SIZE (ALIAS FOR NEXTSTEP)

Description

This keyword defines the maximum time step size the simulator should take for the next time step. This keyword can be used to reset the time step for when known large changes to the model are taking place that may result in time step chops. For example, if the reporting time size is using monthly reporting steps via the DATES keyword in the SCHEDULE section, then if for example, a group of wells start production at a given date, then the NEXT keyword can be used to shorten the next step in order to avoid a time step chop.

Time steps chops are computationally expensive as the simulator cannot solve the current time step at the given tolerance, and therefore has to reduce the time step size. For example, if the previous completed time step was at day 365 and the current time step ending at 396 days cannot be solved, then the simulator will reduce the current time step to perhaps end at day 370, if this still cannot solved then the time step will be chopped back again to perhaps to less then one day. Using the NEXT or NEXTSTEP keyword, the simulator is instructed to take a small time step in the anticipation that this will avoid time step chops and thus improve computational performance.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	NSTEP1	NSTEP1 is a real positive value that defines the maximum length of the next time step.			
		days	days	hours	None
2	NSTEP2	NSTEP2 is a character string that should be set to either NO or YES to state if the NSTEP1 should be applied to future reporting time steps. 1) NO: Means that NSTEP1 should not be applied to subsequent reporting time steps. 2) YES: Means that NSTEP1 should be applied to subsequent reporting time steps. The default value of NO means that NSTEP1 will only be applied once.			NO
Notes: 1) The keyword is terminated by a "/".					

Table 12.56: NEXT Keyword Description

See also the DATES and TSTEP keywords in the RUNSPEC section that are used to advance the simulation through time.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored. See section *2.2 Running OPM Flow 2023-10 From The Command Line* on how to control time stepping for OPM Flow.

Example

```
--       NEXT   ALL
--       STEP   TIME
--       ----   ----
NEXT
         1     'NO'                                        /
```

Here the next step size is set to one day and should only be used once.

12.3.164 NEXTSTEP – MAXIMUM NEXT TIME STEP SIZE

Description

This keyword defines the maximum time step size the simulator should take for the next time step. This keyword can be used to reset the time step for when known large changes to the model are taking place that may result in time step chops. For example, if the reporting time size is using monthly reporting steps via the DATES keyword in the SCHEDULE section, then if for example, a group of wells start production at a given date, then the NEXTSTEP keyword can be used to shorten the next step in order to avoid a time step chop.

Time steps chops are computationally expensive as the simulator cannot solve the current time step at the given tolerance, and therefore has to reduce the time step size. For example, if the previous completed time step was at day 365 and the current time step ending at 396 days cannot be solved, then the simulator will reduce the current time step to perhaps end at day 370, if this still cannot solved then the time step will be chopped back again to perhaps to less then one day. Using the NEXT or NEXTSTEP keyword, the simulator is instructed to take a small time step in the anticipation that this will avoid time step chops and thus improve computational performance.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	NSTEP1	NSTEP1 is a real positive value that defines the maximum length of the next time step.			
		days	days	hours	None
2	NSTEP2	NSTEP2 is a character string that should be set to either NO or YES to state if the NSTEP1 should be applied to future reporting time steps. 1) NO: Means that NSTEP1 should not be applied to subsequent reporting time steps. 2) YES: Means that NSTEP1 should be applied to subsequent reporting time steps. The default value of NO means that NSTEP1 will only be applied once.			NO

Notes:

1) The keyword is terminated by a "/".

Table 12.57: NEXTSTEP Keyword Description

See also the DATES and TSTEP keywords in the RUNSPEC section that are used to advance the simulation through time.

See section *2.2 Running OPM Flow 2023-10 From The Command Line* on how to control time stepping for OPM Flow that allows for greater flexibility.

Examples

The first example shows the direct use of the NEXTSTEP keyword:, Here the next step size is set to one day and should only be used once.

```
--       NEXT   ALL
--       STEP   TIME
--       ----   ----
NEXTSTEP
         1     'NO'                                    /
```

The next example shows a more complete use of the keyword for when the field oil production has increased dramatically from 10,000 stb/d to 50,000 stb/d as indicated by the two GCONPROD keywords.

```
-- ------------------------------------------------------------------------
-- SCHEDULE SECTION - 2021-01-01
-- ------------------------------------------------------------------------
--
-- GROUP PRODUCTION CONTROLS
--
-- GRUP    CNTL  OIL    WAT    GAS    LIQ    CNTL  GRUP  GUIDE  GUIDE  CNTL
-- NAME    MODE  RATE   RATE   RATE   RATE   OPT   CNTL  RATE   DEF    WAT
GCONPROD
'FIELD'   'ORAT' 10E3   60E3   300E3  60E3   1*     1*    1*     1*     1*     /
/
RPTSCHED
'WELLS=2'    'WELSPECS'    'CPU=2'     'FIP=2'                              /

DATES
 2  JAN   2021  /
/
RPTSCHED
'NOTHING'                                                                 /

DATES
 1  FEB   2021  /
 1  MAR   2021  /
/
-- GROUP PRODUCTION CONTROLS
--
-- GRUP    CNTL  OIL    WAT    GAS    LIQ    CNTL  GRUP  GUIDE  GUIDE  CNTL
-- NAME    MODE  RATE   RATE   RATE   RATE   OPT   CNTL  RATE   DEF    WAT
GCONPROD
'FIELD'   'ORAT' 50E3   90E3   300E3  90E3   1*     1*    1*     1*     1*     /
/
--
--       NEXT   ALL
--       STEP   TIME
--       ----   ----
NEXTSTEP
         1     'NO'   /
DATES
 1  APR   2021  /
 1  MAY   2021  /
 1  JUN   2021  /
 1  JLY   2021  /
 1  AUG   2021  /
 1  SEP   2021  /
 1  OCT   2021  /
 1  NOV   2021  /
 1  DEC   2021  /
/
```

Given a start date of January 1, 2020 set via the START keyword in the RUNSPEC section, the above example shows the initial oil production of 10,000 stb/d starting in January 1, 2020, and continuing up to March 1, 2021. At the March 1, 2021 time step the field oil production rate is increased to 50,000 stb/d and the maximum next time step is set to one day. After the one day time step is completed (March 2, 2012), the simulator will progressively in increase the time step size until a maximum of 31 days is reached. The 31 day maximum is a result of requesting monthly time steps via the DATES keyword. The intent of using the NEXTSTEP keyword in this case is to prevent time step chops occurring due to the "shock" to the system caused by the large increase in oil production.

12.3.165 NEXTSTPL – MAXIMUM NEXT TIME STEP SIZE (LGR)

Description

This keyword defines the maximum time step size the simulator should take for the next time step for all Local Grid Refinements (LGR"). This keyword can be used to reset the time step for when known large changes to the model are taking place that may result in time step chops. For example, if the reporting time size is using monthly reporting steps via the DATES keyword in the SCHEDULE section, then if for example, a group of wells start production at a given date, then the NEXTSTPL keyword can be used to shorten the next step in all the LGRs in order to avoid a time step chop.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	NSTEP1	NSTEP1 is a real positive value that defines the maximum length of the next time step.			
		days	days	hours	None
2	NSTEP2	NSTEP2 is a character string that should be set to either NO or YES to state if the NSTEP1 should be applied to future reporting time steps. 1) NO: Means that NSTEP1 should not be applied to subsequent reporting time steps. 2) YES: means that STEP1 should be applied to subsequent reporting time steps. The default value of NO means that NSTEP1 will only be applied once.			NO
Notes: 1) The keyword is terminated by a "/".					

Table 12.58: NEXTSTPL Keyword Description

See also the NEXT and NEXTSTEP keywords in the SCHEDULE section that are used to control the global grid's next time step.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped. See section *2.2 Running OPM Flow 2023-10 From The Command Line* on how to control time stepping for OPM Flow.

Example

```
--
--        NEXT   ALL
--        STEP   TIME
--        ----   ----
NEXTSTEP
          1     'NO'                                  /
```

Here the next step size for all LGRs is set to one day and should only be used once.

12.3.166 NODEPROP – DEFINE NETWORK NODE PROPERTIES FOR EXTENDED NETWORK

Description

This keyword defines the network node properties for the extended network option for when the Extended Network Model has been invoked by the NETWORK keyword in the RUNSPEC section. There are two types of network facilities in the simulator, the Standard Network model, which is defined with the GRUPNET keyword in the SCHEDULE section and the Extended Network Model defined by the BRANPROP and NODEPROP keywords, again in the SCHEDULE section.

For the Extended Network Model the group hierarchy can be different to that defined by the GRUPTREE keyword; however, the bottom most nodes in the network tree associated with wells, must be the same as that defined by the GRUPTREE keyword.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	NODE	A character string of up to eight characters in length that defines the node name for the data on this keyword record.			None
2	PRESS	A real value that sets the terminal fixed pressure for the node, this should be set to: 1) A real positive value to define the fixed pressure for the node if the node is a terminal node, otherwise: 2) PRESS should be set to the default value of 1* or a real negative value if the node is not a terminal node.			
		psia	bars	atm	1*
3	CHOKE	CHOKE is a defined character string that sets if the downstream branch (or uptree branch, being the one closer towards the terminal node) from this node should have the capability to choke back the flow rate in order impose a flow constraint. Here the downstream branch is the node furthermost from the wells. Thus for a production network, this will be an outlet branch as the wells are exporting fluid from the branch node. Whereas for an injection node, this is an inlet branch as the wells are importing the injection fluid.			NO
4	GASLIFT	A defined character string that sets if the associated subordinate well's produced gas lift gas should be included in the node's flow stream (YES), or not (NO). GASLIFT should be set to either: 1) NO: Do not include gas lift gas in the node's production. This means that only the produced gas from the node's subordinate wells will be included in the node's production stream. 2) YES: Include both gas lift gas and produced gas in the node's production. This means that all gas from the subordinate wells will be included in the node's production stream. If NODE does not have any subordinate wells or satellite groups (see the GSATPROD keyword in the SCHEDULE section) directly attached to the node, then GASLIFT should be defaulted (1*) or set to NO. The option is only valid for producing networks			NO

No.	Name	Description			Default
		Field	Metric	Laboratory	
5	GROUP	A character string of up to eight characters in length that defines the group for which the automatic choke will be applied in order to match the group's target rate (the TARGET variable on the GCONPROD keyword in the SCHEDULE section). The target rate is matched by adjusting the pressure drops across the automatic choke. The default value of 1* uses NODE (item one) as the group if it exists in the run. In addition, if NODE is just connected to subordinate wells then GROUP should also be defaulted.			1*
6	GRPNAME	GRPNAME defines the name of the source or sink group in the commercial compositional simulator; however, the variable is not used as sources and sinks nodes must have the same name as their comparable groups in the Extended Network Model in both OPM Flow and the commercial black-oil simulator.			1*
7	NETYPE	NETYPE defines the network type in the commercial compositional simulator: PROD, WINJ or GINJ. However, the variable is not used as only production networks are supported in the Extended Network Model in both OPM Flow and the commercial black-oil simulator.			1*

Notes:

1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".

Table 12.59: NODEPROP Keyword Description

See also the NETWORK keyword in the RUNSPEC section and the BRANPROP in the SCHEDULE section.

Example

Given the following Extended Network model in Figure 12.5.

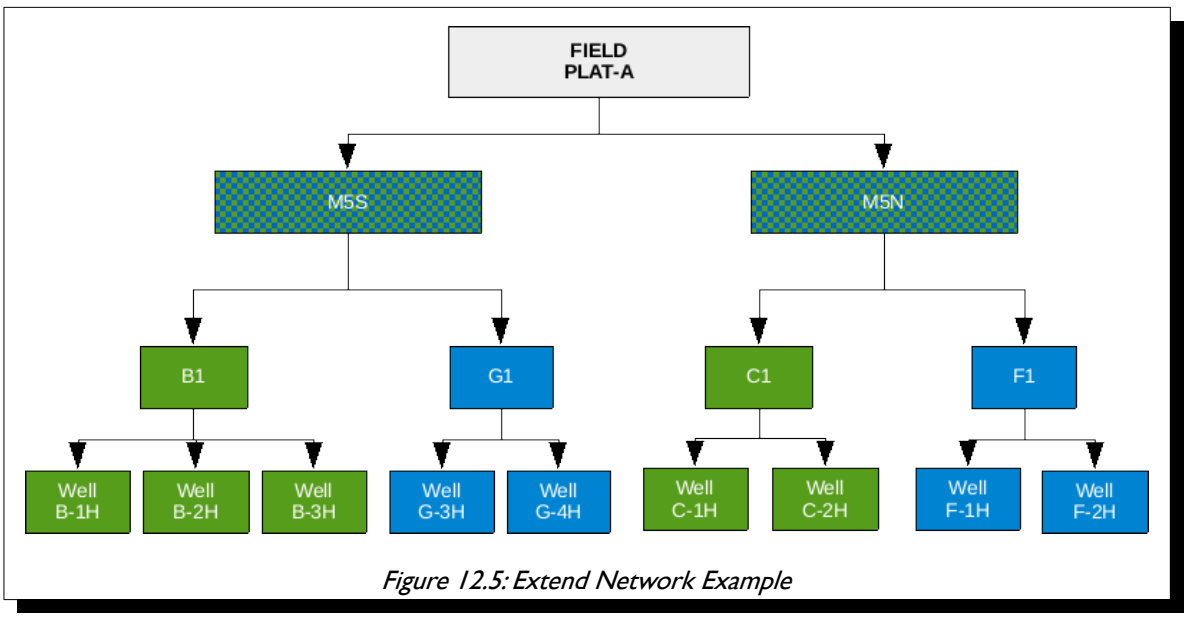


Figure 12.5: Extend Network Example

First the Extended Network model should be used invoked in the RUNSPEC section, and then the BRANPROP keyword should be used to define the branch network, and finally the NODEPROP keyword is used to describe the node properties with the network.

```
-- ================================================================================
--
-- RUNSPEC SECTION
--
-- ================================================================================
RUNSPEC
--
--       ACTIVATE THE EXTENDED NETWORK OPTION AND DEFINE PARAMETERS
--
--       MAX.     MAX      NOT
--       NODE     LINK     USED
NETWORK
         3         2       1*                                                    /
….‥……‥‥
‥……‥‥…‥
-- ================================================================================
--
-- SCHEDULE SECTION
--
-- ================================================================================
SCHEDULE
--
--       EXTENDED NETWORK BRANCH PROPERTIES
--
-- DOWN  UP       VFP     VFP
-- NODE  NODE     TABLE   ALFQ
BRANPROP
B1       PLAT-A   5       1*                                                   /
C1       PLAT-A   4       1*                                                   /
/
--
--       EXTENDED NETWORK NODE PROPERTIES
--
-- NODE  NODE   CHOKE  GAS   CHOKE  SOURCE  NETWORK
-- NAME  PRESS  OPTN   LIFT  GROUP  SINK    TYPE
NODEPROP
PLAT-A   21.0   NO     NO                                                     /
B1       1*     NO     NO                                                     /
C1       1*     NO     NO                                                     /
/
```

Here the main platform for the field, PLAT-A, has a fixed 21 barsa pressure applied as an operating constraint.

12.3.167 NOECHO – DEACTIVATE ECHOING OF USER INPUT FILES TO THE PRINT FILE

Turns off echoing of all the input files to the print file. Note by default echoing of the inputs files is active. but can subsequently be switched off by the NOECHO activation keyword.

See *NOECHO – Deactivate Echoing of User Input Files to the Print File* in the GLOBAL section for a full description.

12.3.168 NOHMD – DEACTIVATE HISTORY MATCH GRADIENT DERIVATIVE CALCULATIONS

The NOHMD deactivates various history match gradient derivative calculations for when the History Match Gradient option has been activated by the HMDIMS keyword in the RUNSPEC section. The keyword consists of a series of character strings that define which derivative should be switch off based on the keyword that requested the derivatives to be calculated, for example HMFAULTS keyword in the GRID section. If an empty list is entered then all the gradient derivative calculations previously requested are switch off. The keyword is useful for changing from history matching runs to predication cases, as the prediction cases will be more computationally efficient without the burden of the gradient derivative calculations.

See *NOHMD – Deactivate History Match Gradient Derivative Calculations* in the SOLUTION section for a full description.

12.3.169 NOHMO – DEACTIVATE HISTORY MATCH GRADIENT DERIVATIVE CALCULATIONS (ALIAS)

The NOHMO deactivates various history match gradient derivative calculations for when the History Match Gradient option has been activated by the HMDIMS keyword in the RUNSPEC section. The keyword consists of a series of character strings that define which derivative should be switch off based on the keyword that requested the derivatives to be calculated, for example HMFAULTS keyword in the GRID section. If an empty list is entered then all the gradient derivative calculations previously requested are switch off. The keyword is useful for changing from history matching runs to predication cases, as the prediction cases will be more computationally efficient without the burden of the gradient derivative calculations.

See *NOHMO – Deactivate History Match Gradient Derivative Calculations (Alias)* in the SOLUTION section for a full description.

12.3.170 NOSIM – ACTIVATE THE NO SIMULATION MODE FOR DATA FILE CHECKING

NOSIM switches the mode of OFM Flow to data input checking mode. In this mode the input file is read and all messages and print instructions are sent to the respective output files. The SCHEDULE section is read but the simulation is not performed.

See *NOSIM – Activate the No Simulation Mode for Data File Checking* in the RUNSPEC section for a full description

12.3.171 NOWARN – DEACTIVATE WARNING MESSAGES

Turns off warning messages to be printed to the print file; note that this keyword is deactivated by default and can subsequently be switched off by the WARN activation keyword. The warning messages may be turned on and off using keywords WARN and NOWARN.

See *NOWARN – Deactivate Warning Messages* in the GLOBAL section for a full description.

12.3.172 NUPCOL – DEFINE THE NUMBER OF NEWTON ITERATIONS USED TO UPDATE WELL TARGETS

The NUPOL keyword defines the maximum number of Newton iterations within a time step that may be used to update the well production and injection targets, after which the well targets will be frozen until the time step calculations have converged and the time step is complete.

See *NUPCOL – Define the Maximum Number of Newton Iterations Used to Update Well Targets* in the RUNSPEC section for a full description and also section *2.2 Running OPM Flow 2023-10 From The Command Line* on how to set various other numerical control parameters for OPM Flow.

12.3.173 NWATREM – NODE WATER REMOVAL (EXTENDED NETWORK)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The NWATREM keyword defines an extended network node as a point where water is removed from the network, for when the Extended Network option has been activated by the NETWORK keyword in the RUNSPEC section. The water to be removed can be specified as a rate or as a fraction of the total volume passing through the node.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.174 OUTSOL – DEFINE DATA TO BE WRITTEN TO THE RESTART FILE (RETIRED)

This keyword defines the data and frequency of the data to be written to the RESTART file at each requested restart point. The keyword has been replaced by the RPTRST keyword in the SOLUTION and SCHEDULE sections and is therefore considered retired.

See *OUTSOL – Define Data to be Written to the RESTART File (Retired)* in the SOLUTION section for a full description.

12.3.175 PICOND – DEFINE THE GENERALIZED PSEUDO PRESSURE PARAMETERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The PICOND keyword defines the Generalized Pseudo Pressure ("GPP")[314] and [315] parameters used in a gas condensate well connection inflow equations. GPP accounts for both the impact of condensate drop out and compressibility in the mobility inflow term . If the keyword is absent from the input deck then the default values are applied.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

[314] *Whitson, C. H. and Fevang, Ø. "Generalised Pseudopressure Well Treatment in Reservoir Simulation," Presented at the IBC Technical Services Conference on Optimisation of Gas Condensate Fields, Aberdeen, UK (June 26-27, 1997)*

[315] *Whitson, C. H. and Fevang, Ø. "Modeling Gas Condensate Well Deliverability," paper SPE 30714, SPE Reservoir Engineering (1996) 11, No. 4, 221-230; also presented at the SPE Annual Technical Conference and Exhibition, Dallas, Texas, USA (October 22-25, 1995).*

12.3.176 PIMULTAB – DEFINE WELL PRODUCTIVITY INDEX VERSUS WATER CUT TABLES

Description

PIMULTAB defines productivity index multiplier versus water cut tables that are used to scaled a well's connection factors based on connection's current producing water cut. The tables are used for modeling the productivity decline due to increasing water cut. Allocation of the tables to a well is via the WPITAB keyword in the SCHEDULE section.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WCUT	A real monotonically increasing positive columnar vector that defines the maximum surface water cut for the corresponding PIMULT vector. Water cut is defined as $f_w = \frac{q_w}{q_w + q_o}$.			
		dimensionless	dimensionless	dimensionless	None
2	PIMULT	A real positive decreasing columnar vector that defines the productivity index multiplier used to scale a well's connection factors, for the corresponding WCUT vector.			
		dimensionless	dimensionless	dimensionless	None

Notes:

1) The keyword is followed by NTPIMT tables as stated on the PIMTDIMS keyword in the RUNSPEC section.
2) Each table must contain a minimum of two rows and a maximum of NRPIMT rows as declared on the PIMTDIMS keyword in the RUNSPEC section.
3) Each table is terminated by a "/" and there is no "/" terminator for the keyword.

Table 12.60: PIMULTAB Keyword Description

See also the WPITAB keyword that allocates the tables to the wells, and also the WPIMULT keyword that scales a well's productivity index by a constant value, both of which are in the SCHEDULE section.
This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

Examples

Given NTPIMT equals two and NRPIMT equals four on PIMTDIMS keyword in the RUNSPEC section, then:

```
--        DEFINE WELL PRODUCTIVITY INDEX VERSUS WATER CUT TABLES
--
--        MAX       PI
--        WCUT      MULT
--        ------    ------
PIMULTAB
          0.0000    1.0000
          0.2500    0.9500
          0.5000    0.8500
          0.7500    0.7500                                  /
--
```

```
--
         0.0000   1.0000
         0.2500   0.9500
         0.5000   0.8500
         0.7500   0.7500                                    /
```

The next example is summarized from the Norne model with NTPIMT equals one and NRPIMT equals to 51 on the PIMTDIMS keyword in the RUNSPEC section.

```
--
--       DEFINE WELL PRODUCTIVITY INDEX VERSUS WATER CUT TABLES
--       The following is the reviewed model in Aug-2006, low-high case
--       a=0.25, b=0.1; PIMULT=(1-a)/exp(fw/b)+a
--
--       MAX      PI
--       WCUT     MULT
--       ------   ------
PIMULTAB
         0.000    1.0000
         0.025    0.8341
         0.050    0.7049
         0.075    0.6043
         0.100    0.5259
         0.125    0.4649
         0.150    0.4173
         0.175    0.3803
         0.200    0.3515
         0.225    0.3290
         0.250    0.3116
         0.275    0.2979
         0.300    0.2873
         0.325    0.2791
         0.350    0.2726
         0.375    0.2676
         0.400    0.2637
         0.425    0.2607
         0.450    0.2583
         0.475    0.2565
         0.500    0.2551
         0.525    0.2539
         0.550    0.2531
         0.575    0.2524
         0.600    0.2519
         0.625    0.2514
         0.650    0.2511
         0.675    0.2509
         0.700    0.2507
         0.725    0.2505
         0.750    0.2504
         0.775    0.2503
         0.800    0.2503
         0.825    0.2502
         0.850    0.2502
         0.875    0.2501
         0.900    0.2501
         0.925    0.2501
         0.950    0.2501
         0.975    0.2500
         1.000    0.2500 /
```

12.3.177 PLYADS - DEFINE POLYMER ROCK ADSORPTION TABLES

The PLYADS keyword defines the rock polymer adsorption tables for when the polymer option has been activated by the POLYMER keyword in the RUNSPEC section. Alternatively, the functions can be entered via the PLYADSS keyword in the PROPS section for when salt sensitivity is to be considered.

See *PLYADS - Define Polymer Rock Adsorption Tables* in the PROPS section for a full description.

12.3.178 PLYDHFLF - DEFINE POLYMER THERMAL DEGRADATION HALF-LIFE TABLES

The PLYDHFLF keyword defines the polymer thermal degradation half-life with respect to temperature functions for when the polymer option has been activated by the POLYMER keyword in the RUNSPEC section.

See *PLYDHFLF - Define Polymer Thermal Degradation Half-Life Tables* in the PROPS section for a full description.

12.3.179 PLYMAX - DEFINE POLYMER-SALT VISCOSITY MIXING CONCENTRATIONS

The PLYMAX keyword defines maximum polymer and salt concentrations that are to be used in the mixing parameter calculation of the fluid component viscosities, for when the polymer option has been activated by the POLYMER keyword in the RUNSPEC section.

See *PLYMAX - Define Polymer-Salt Viscosity Mixing Concentrations* in the PROPS section for a full description.

12.3.180 PLYROCKM - MODIFY POLYMER-ROCK PROPERTIES

Description

The PLYROCKM keyword modifies rock properties entered via the PLYCAMAX, PLYKRRF, PLYRMDEN, and PLYROCK keywords in the PROPS section, for when the Polymer option has been activated by the POLYMER keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.181 PLYSHEAR – ACTIVATE AND DEFINE POLYMER SHEARING PARAMETERS

The PLYSHEAR keyword activates and the defines the polymer shear thinning-thickening option for when the polymer option has been activated by the POLYMER keyword in the RUNSPEC section.

See *PLYSHEAR – Activate and Define Polymer Shearing Parameters* in the PROPS section for a full description.

12.3.182 PLYSHLOG - ACTIVATE AND DEFINE THE POLYMER SHEARING LOGARITHMIC PARAMETERS

This keyword activates and defines the parameters for the logarithm-based polymer shear thinning/thickening option.

See *PLYSHLOG - Activate and Define the Polymer Shearing Logarithmic Parameters* in the PROPS section for a full description.

12.3.183 PLYVISC – DEFINE POLYMER VISCOSITY SCALING FACT

PLYSVISC defines the polymer viscosity scaling factors used to determine the relationship of pure water viscosity with respect to increasing polymer saturation within a grid block. The polymer option must be activated by the POLYMER keyword in the RUNSPEC section in order to use this keyword.

See *PLYVISC – Define Polymer Viscosity Scaling Factors* in the PROPS section for a full description.

12.3.184 PLYVISCS – DEFINE POLYMER-SALT VISCOSITY SCALING FACTORS

PLYSVISCS defines the polymer-salt viscosity scaling factor tables applied to pure water that are used to determine the viscosity of a polymer-salt mixture with respect to increasing polymer saturation within a grid block. The polymer option must be activated by the POLYMER keyword, as well as the brine phase declared by the BRINE keyword in the RUNSPEC section in order to use this keyword. However the ECLM keyword in the RUNSPEC must not be used with this keyword.

See *PLYVISCS – Define Polymer-Salt Viscosity Scaling Factors* in the PROPS section for a full description.

12.3.185 PLYVISCT – DEFINE POLYMER-TEMPERATURE VISCOSITY SCALING FACTORS

PLYSVISCT defines the polymer-temperature viscosity scaling factor tables applied to pure water that are used to determine the viscosity of the polymer at a given temperature with respect to increasing polymer saturation within a grid block. Both the polymer option must be activated by the POLYMER keyword and the temperature option invoked by the TEMP keyword in the RUNSPEC section in order to use this keyword. However the BRINE keyword in the RUNSPEC must not be used with this keyword, that is the salt sensitivity options should be deactivated.

See *PLYVISCT – Define Polymer-Temperature Viscosity Scaling Factors* in the PROPS section for a full description.

12.3.186 PLYVSCST – DEFINE POLYMER-SALT-TEMPERATURE VISCOSITY SCALING FACTORS

PLYVSCST defines the polymer-salt-temperature viscosity scaling factor tables applied to pure water that are used to determine the viscosity of the polymer at a given salt concentration and for a given temperature, with respect to increasing polymer saturation within a grid block. Both the polymer option must be activated by the POLYMER keyword and the temperature option invoked by the TEMP keyword in the RUNSPEC section in order to use this keyword. In addition, the BRINE keyword in the RUNSPEC must also be invoked. The keyword is used in conjunction with the SALTNODE keyword to define the various salt concentrations and the TEMPNODE keyword to define the various reservoir temperatures. Both keywords are in the PROPS section.

See *PLYVSCST – Define Polymer-Salt-Temperature Viscosity Scaling Factors* in the PROPS section for a full description.

12.3.187 PRIORITY – ACTIVATE AND DEFINE WELL PRIORITIZATION COEFFICIENTS

Description

The PRIORITY keyword activates the Well Priority option and defines the coefficients in the well priority equation. Wells under group control are ranked based on their well potential in order to satisfy group controls. For example if a group's oil target is exceeded, then the group may shut-in the lease productive oil wells based on their well potential. The Priority option is an alternative form of ranking the wells based on the following equation:

$$\text{Priority} = \frac{a_1 + a_2 Q_{oil} + a_3 Q_{water} + a_4 Q_{gas}}{b_1 + b_2 Q_{oil} + b_3 Q_{water} + b_4 Q_{gas}} \quad (12.31)$$

Where:

Q_{oil} = well oil potential
Q_{water} = well water potential
Q_{gas} = well gas potential
$a_{1\text{-}4}$ = priority coefficients supplied by this keyword
$b_{1\text{-}4}$ = priority coefficients supplied by this keyword

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	TIME	A real positive integer that defines the minimum time interval between executing the well priority calculation. The calculation is performed at the beginning of the time step that exceeds the previous calculation (t0) by a minimum of TIME, that is for when tn ≥ (t0 + TIME). Note that the default value of zero means that the calculation is performed at each time step. As a consequence, this may result in some oscillation as well wells are switched on/off at subsequent time steps.			0
		days	days	hours	
2	A1	A real positive integer greater than or equal to zero that defines a_1 priority coefficient in equation (12.31).			0
3	A2	A real positive integer greater than or equal to zero that defines a_2 priority coefficient in equation (12.31).			0
4	A3	A real positive integer greater than or equal to zero that defines a_3 priority coefficient in equation (12.31).			0
5	A4	A real positive integer greater than or equal to zero that defines a_4 priority coefficient in equation (12.31).			0
6	B1	A real positive integer greater than or equal to zero that defines b_1 priority coefficient in equation (12.31).			0

No.	Name	Description			Default
		Field	Metric	Laboratory	
7	B2	A real positive integer greater than or equal to zero that defines b_2 priority coefficient in equation (12.31).			0
8	B3	A real positive integer greater than or equal to zero that defines b_3 priority coefficient in equation (12.31).			0
9	B4	A real positive integer greater than or equal to zero that defines b_4 priority coefficient in equation (12.31).			0
Notes: 1) The keyword is terminated by a "/".					

Table 12.61: PRIORITY Keyword Description

Example

```
--
--       SETS COEFFICIENTS FOR WELL PRIORITIZATION OPTION
--
--       TIME    A      B      C      D      E      F      G      H
--       STEP           Qo     Qw     Qg            Qo     Qw     Qg
PRIORITY
         0.0     0.0    1.0    0.0    0.0    1.0    0.0    0.0    0.0 / High Oil Pot
--       0.0     0.0    1.0    1.0    0.0    0.0    0.0    1.0    0.0 / Low Water Cut Pot
--       0.0     0.0    1.0    0.0    0.0    0.0    0.0    0.0    1.0 / Low GOR Pot
```

The above example defines the well priority calculation to be based on a well's oil potential, with calculation to be performed at each time step. Note that the low water cut and low GOR options are given for reference but are commented out and therefore ignored by the simulator.

12.3.188 PRORDER – DEFINE A GROUP PRODUCTION RULES SEQUENCE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

PRORDER defines the order of group production rules to be implemented fore when a group's target is not satisfied.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.189 PYACTION – DEFINE PYTHON BASED ACTION CONDITIONS AND COMMAND PROCESSING

Description

The PYACTION keyword is part of OPM Flow's Python scripting facility that loads a standard Python script file that can be used to define a series of conditions and actions as the simulation proceeds through time. The "included" Python script file is executed by the standard Python interpreter. Thus, OPM Flow's Python scripting facility offers greater flexibility compared to the commercial simulator's ACTION series of keywords (ACTION, ACTIONG, ACTIONR, ACTIONS, ACTIONW and ACTIONX) that can apply Boolean conditional tests to variables at the field, group, region, well segment and well levels. Note that OPM Flow has also implemented the commercial simulator's ACTIONX keyword, but not the ACTION, ACTIONG, ACTIONR, ACTIONS and ACTIONW keywords, as the ACTIONX keyword implements their functionality with greater flexibility.

This keyword starts the definition of a PYACTION section that stipulates the name of the action and a string indicating the number of times the action should be run; this is then followed file containing the Python script. Noted that unlike the commercial simulator's ACTIONX keyword there is no terminating keyword like the commercial simulator's ENDACTIO keyword.

Although this keyword is read by OPM Flow and the script processing has been implemented, one should use caution when using this facility as it may result in OPM Flow aborting. This is because the PYACTION facility allows the user to implement complex functionality and the implementation is new for the 2020-04 release. Users should therefore use caution when using this facility.

Note

This is an OPM Flow specific keyword for the simulator's scripting facility using the standard Python interpreter, as such it gives more flexibility than the commercial simulator's ACTIONX keyword, although OPM Flow also supports this as well.

The keyword should be considered experimental as details of the OPM Flow - Python interface might change for future releases. In particular, the current implementation is quite minimal; however, future releases are expected to add more entry points in the Schedule class which can be used to manipulate the reservoir model as the simulation progresses. As a user you are encouraged to come with wishes in this regard.

The PYACTION keyword is a very powerful keyword and allows for any piece of Python code to be included and run, including potentially malicious code. The important point is to scrutinize any PYACTION keyword in a deck you receive from other parties.

No.	Name	Description	Default
PYACTION		PYACTION declares the start of a PYACTION Definition Section. This is then followed by one record that defines the name of the action and a string indicating the number of times the action should be run; this is then followed by a second record indicating the file containing the Python script.	Not Applicable
1-1	ACTNAME	ACTNAME is a character sting of any length enclose in quotes that defines the name of this action definition.	Not Applicable

No.	Name	Description	Default
1-2	ACTNSTEP	ACTNSTEP is a defined character string enclosed in quotes that indicates the number of times the action should be performed, and should be set to one of the following: 1) FIRST_TRUE: This option forces the action to be run at the end of every time step until the script returns a value of True. 2) SINGLE: Here the action script is run only once. 3) UNLIMITED: If this option is selected then the action is run at the end of every time step.	Not Applicable
1-4	/	Record terminated by a "/"	Not Applicable
2-1	FILENAME	A character string enclosed in quotes that defines the Python module/script file to read in and to be processed by OPM Flow.	None
2-2	/	Record terminated by a "/"	Not Applicable
Notes: 1) There is no terminating "/" for this keyword.			

Table 12.62: PYACTION Keyword Description

The PYACTION keyword is a result of combining two programming languages, the interactive Python interpreter and OPM Flow's source code language C++. When combing two languages one extends and embeds one into the other. When extending Python with C++ the functionality implemented in C++ is made available to Python applications, when embedding Python in C++ one can call Python functions from within C++. The PYACTION keyword is based on embedding a Python interpreter in the C++ OPM Flow simulator, but the Python code actually runs as part of the PYACTION keyword is based on wrapping C++ objects in Python, that is extending Python.

In order to fully benefit from the power of the PYACTION keyword one should familiarize oneself with the Python wrapper classes for the various OPM Flow C++ classes. These classes are essential to share the simulator state with the PYACTION run() function. That is, the function provides the interface between the two programming languages.

To learn more about these classes you can use the pydoc utility, for example, to learn more about the SummaryState class, in a Linux terminal one would issue the following bash command:

```
bash% pydoc opm.io.sim.SummaryState
```

The Python script file (FILENAME in the PYACTION keyword) should be a standard Python module[316] that defines the run() function with a pre-defined argument definition and should consist of 100% pure Python. The PYACTION Python module (FILENAME) is imported during processing of the input deck and as such this implies:

1) Basic Python syntax checking is performed during reading the FILENAME script file.
2) It is verified that the module has the required run() function with the correct format.
3) Code which is at module level, that is outside of any function call is run immediately at the time import time – this is probably not one wants and should therefore be avoided.

[316] *A Python module is a file containing Python definitions and statements. The file name is the module name with the suffix .py appended. Within a module, the module's name (as a string) is available as the value of the global variable __name__.*

The syntax of the run() command is given in Table 12.63 together with a description of the function's call variables.

RUN() Function Definition		

	`def run(ecl_state, schedule, report_step, summary_state):`
	`#`
	`# OPM Flow PYACTION Module Script`
	`#`
	`.....`
	`.....`
	`return`	

No.	**Name**	**Description**
1	ecl_state	The ecl_state argument is a opm.io.ecl_state.EclipseState instance which is initialized with all the static information in the simulation, for example the 3D grid properties and the PVT and saturation tables This object is read by the run() function and the available data should not be modified as any updates will not be passed back to the simulator.
2	schedule	The schedule argument is a opm.io.sched.Schedule instance which has all the SCHEDULE information internalized (Schedule object). Currently, there are only a few call points in opm.io.sched.Schedule which can be used to update the state of the simulation, most notably: `schedule.shut_well(well_name, report_step)` Which can be used to shut a well at a particular report step. Having additional call points to the opm.io.ecl_state.EclipseState instance in order to update the Schedule object in more ways is an obvious candidate for improving the PYACTION functionality.
3	report_step	The current report step.

4	summary_state	The summary_state argument is an instance of the opm.io.sim.SummaryState class, the purpose of this class is to serve as a container for SUMMARY variables that contain the simulation results; for example, the Field Oil Rate (FOPR) or a well's Water CuT (WWCT). See the *SUMMARY SECTION* for a detailed description of the variables available.
		The summary_state variable will typically be the variable one will use to access the state of the simulation. For example, to check if the water cut in well OP01exceeds 0.50 one would use the following statement:
		`if summary_state["WWCT:OP01"] > 0.50:`
		`...`
		The summary_state variable can also be use to update variables including UDQ variables, i.e:
		`summary_state.update_well_var("OP01", "WUXX", 0.25)`
		The above assigns a value of 0.25 to the well UDQ variable WUXX for well OP01.

Table 12.63: PYACTION Module Script Definition

See also the PYINPUT and PYEND keywords in the GRID[317] section which are also part of OPM Flow's Python scripting facility, that process standard Python commands that can be used to manipulate and define

[317] *Note PYINPUT and PYEND can be used in the GRID, EDIT, PROPS, SOLUTION, SUMMARY and SCHEDULE sections, but is described in the GRID section.*

the simulators input parameters during processing of the input deck. The main purpose of the facility is to script the construction of the various keywords.

Examples

The first example checks if well OP01 has a water cut greater than 0.80 and if so then the well is just in. In the input deck we would have:

```
--
--       START OF PYACTION SECTION
--
--       ACTNAME       ACTNSTEP
PYACTION
         'MAXWCUT'     'FIRST_TRUE'                        /
         'pthon/script/MAXWCUT.py'                         /
--
--       END OF PYACTION SECTION
--
```

And then in the Python module file 'pthon/script/MAXWCUT.py' one would have:

```
#
# OPM Flow PYACTION Module Script
#
def run(ecl_state, schedule, report_step, summary_state):
if summary_state.get_well_var("OP1", "WWCT") > 0.80:
    schedule.shut_well("OP1", report_step)
    print("Well OP01 has been shut-in due to WWCT > 0.80")
    return
```

The next example is based on the first example from the ACTIONX keyword (*ACTIONX – Define Action Conditions and Command Processing*). The Python script first checks if the field's water production is greater than 30,000 stb/d, and if not returns control back to the simulator. If the field water production is greater than 30, 000 stb/d then the script uses a Python variable count to keep track of the number of times the script has been executed, and then sorts the wells from high water cut to low, via the wct_list variable, and then shuts in the worst offending well. If a well is shut-in the count variable is increase by one and control is passed back to the simulator.

```
--
--       START OF PYACTION SECTION
--
--       ACTNAME       ACTNSTEP
PYACTION
         'WSHUTIN'     'FIRST_TRUE'                        /
         'pthon/script/WSHUTIN.py'                         /
--
--       END OF PYACTION SECTION
--
```

And then in the Python module file 'pthon/script/WSHUTIN.py' one would have:

```
#
# OPM Flow PYACTION Module Script
#
def run(ecl_state, schedule, report_step, summary_state):
if summary_state["FWPR"] < 3000:
    return False
#
# Define Counter and Check Value
#
count = storage.get("COUNT", 0)
if count == 10:
    return True
#
```

```
        # Get Sorted Well List
        #
        wct_list = sorted( [ (well, summary_state.well_var(well, "WWCT")) for
                           well in summary_state.wells] )
        well, wwct = wct_list[0]
        #
        # Shut-in Wells
        #
        if wwct > 0:
            schedule.shut_well( well, report_step )
            storage["COUNT"] = count +1
        else:
            return False
```

Note that by using PYACTION it makes sense to combine the UDQ variable into the PYACTION statement, like shown in the example, although this is not necessary, as one could in principle use a normal UDQ statement and then access the variables in the PYACTION script using the summary_state variable.

The final example checks to see if the field's gas rates is below 600 MMscf/d and if the simulation time is greater that January 1, 2030. If it is, then compression is installed by re-setting all the gas producing well's THP and BHP pressures to 450 psia and 300 psia respectively. In addition all gas wells currently shut-in are tested to see if they can be opened up under the new THP and BHP constraints.

```
--
--       START OF PYACTION SECTION
--
--       ACTNAME       ACTNSTEP
PYACTION
         'WSHUTIN'     'FIRST_TRUE'                          /
         'pthon/script/PHASE3.py'                            /
--
--       END OF PYACTION SECTION
--
```

And then in the Python module file 'pthon/script/PHASE3.py' one would have:

```
    #
    # OPM Flow PYACTION Module Script
    #
    # Import Python Modules
    #
    import datetime
    #
    # Start of Routine
    #
    def run(ecl_state, schedule, report_step, summary_state):
        #
        # Check Field Rate
        #
        if summary_state["FGPR"] >= 600000:
            return False
        #
        # Check Date
        #
        sim_time = schedule.start +
                   datetime.timedelta( seconds = summary_state.elapsed() )
        if sim_time > datetime.datetime(2030, 1, 1):
            #
            # Do WELTARG and WTEST action – Currently Not Supported
            #
            …….…..…..…...…..…..…...................
            #
            # Return
            #
            return True
```

12.3.190 PYEND – END THE DEFINITION OF A PYINPUT SECTION

The PYINPUT and PYEND keywords are a part of OPM Flow's Python scripting facility that processes standard Python commands that can be used to manipulate and define the simulators input parameters during processing of the input deck. The main purpose of the facility is to script the construction of the various keywords. A PYINPUT Definition Section is terminated by a PYEND keyword on a separate single line.

See *PYEND – End the Definition of a PYINPUT Section* in the GRID section for a full description.

12.3.191 PYINPUT – DEFINE THE START OF A PYINPUT SECTION

The PYINPUT and PYEND keywords are a part of OPM Flow's Python scripting facility that processes standard Python commands that can be used to manipulate and define the simulators input parameters during processing of the input deck. The main purpose of the facility is to script the construction of the various keywords used by the simulator. PYINPUT declares the start of a PYINPUT Definition Section on a single separate line, which is then followed by various standard Python commands, with one command per line.

See *PYINPUT – Define the Start of a PYINPUT Section* in the GRID section for a full description.

12.3.192 QDRILL – DEFINE SEQUENTIAL DRILLING QUEUE WELLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The QDRILL keyword places previously defined wells in the Sequential Drilling Queue. Wells in this type of queue will be automatically drilled and completed in the sequence entered in order to satisfy group targets, as defined by the GCONPROD, GCONINJE and GCONSALE keywords. or a group's production potential as per the GDRILLPOT keyword. All the previously mentioned keywords are in the SCHEDULE section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.193 RAINFALL – CONSTANT FLUX AQUIFER RAINFALL FLUX BY MONTH

This keyword, RAINFALL, defines the month by month rainfall flux for constant flux aquifers.

See *RAINFALL – Constant Flux Aquifer Rainfall Flux by Month* in the GRID section for a full description.

12.3.194 RCMASTS – RESERVOIR COUPLING GROUP MINIMUM TIME STEP FOR FLOW RESTRICTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

RCMASTS is used when reservoir coupling is invoked by the GRUPMAST and SLAVES keywords in the SCHEDULE section. The keyword should be placed within the master file and it sets the minimum time step size for groups for when a group is being restricted by a group's limiting flow rate fractional change (see the GRUPMAST keyword in the SCHEDULE section).

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.195 REACHES – DEFINE RIVER REACHES STRUCTURE

Description

The REACHES keyword defines the reach structure of a previously characterized river system using the RIVERSYS keyword in the SCHEDULE section. The River option must be activated via the RIVRDIMS keyword in the RUNSPEC section in order to use these keywords.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.196 READDATA – READ SCHEDULE DATA BASED ON CURRENT TIME STEP

Description

The READDATA keyword enables the simulator to read SCHEDULE data files generated by external programs on the fly, that is as the run is progressing. The external program can "hold" the simulation by using a file lock, in order for the external program to evaluate the current simulation results, then write out a SCHEDULE data file for the next time step, and finally releasing the "hold" by deleting the file lock and continuing with newly written SCHEDULE data file. The mechanism can be repeated so that the external program is dictating how the simulation progresses through time,

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.197 REFINE – START THE DEFINITION OF A LOCAL GRID REFINEMENT

The REFINE keyword defines the start of a Cartesian or radial local grid refinement ("LGR") definition that sets the properties of the selected LGR. The keyword is then followed by the property keywords associated with the section where the keyword is being invoked. For example, if the REFINE keyword is used in the GRID section then most of the keywords in that section can be used to set the grid properties for the LGR.

See *REFINE – Start the Definition of a Local Grid Refinement* in the GRID section for a full description.

12.3.198 RIVDEBUG – DEFINE THE DEBUG DATA TO BE PRINTED TO FILE (RIVERS)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword defines the debug data associated with rivers to be written to the debug file (*.DBG), for when the River option has been activated via the RIVRDIMS keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.199 RIVERSYS - DEFINE RIVER SYSTEM (BRANCH STRUCTURE AND BOUNDARY CONDITIONS)

RIVERSYS defines a river system by specifying the branch structure of the river together with the branch's associated boundary conditions, for when the River option has been activated via the RIVRDIMS keyword in the RUNSPEC section.

See *RIVERSYS - Define River System (Branch Structure and Boundary Conditions)* in the SOLUTION section for a full description.

12.3.200 RIVRPROP – MODIFY RIVER REACHES PROPERTIES

Description

The RIVRPROP keyword modifies the individual reaches in a river structure of a previously characterized river system using the RIVERSYS and the REACHES keywords in the SCHEDULE section. The River option must be activated via the RIVRDIMS keyword in the RUNSPEC section in order to use this keyword. RIVRPROP is an alternative and a more concise way to changing the individual reaches in a river structure than the REACHES keyword.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.201 RIVSALT – DEFINE RIVER UPSTREAM FLOW SALT CONCENTRATIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The RIVSALT keyword defines the injected salt concentration in individual river branches in a previously characterized river system using the RIVERSYS and the REACHES keywords in the SCHEDULE section. The River option must be activated via the RIVRDIMS keyword in the RUNSPEC section in order to use this keyword. In addition, the Brine option must also be enabled via the BRINE keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.202 RIVTRACE – DEFINE RIVER UPSTREAM FLOW TRACER CONCENTRATIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The RIVTRACE keyword defines the injected tracer concentration in individual river branches in a previously characterized river system using the RIVERSYS and the REACHES keywords in the SCHEDULE section. The River option must be activated via the RIVRDIMS keyword in the RUNSPEC section in order to use this keyword. In addition, the Tracer option must also be enabled by the TRACER keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.203 RPTHMG - DEFINE WELL HISTORY MATCH GRADIENT REPORTING (GROUPS)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, RPTHMG, either enables or disables history match output reporting to the history match file (*.HMD) for the named group, for when the History Match Gradient option has been activated by the HMDIMS keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

12.3.204 RPTHMW - DEFINE WELL HISTORY MATCH GRADIENT REPORTING (WELLS)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, RPTHMG, either enables or disables history match output reporting to the history match file (*.HMD) for the named well, for when the History Match Gradient option has been activated by the HMDIMS keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

12.3.205 RPTONLY - ACTIVATE THE REPORT TIME STEPS ONLY OPTION FOR THE SUMMARY FILE

This keyword activates the writing out of the SUMMARY file and RSM file data, if the RSM file option has been requested by the RUNSUM keyword in the SUMMARY section, at report time steps only. The default is for all time steps to be written out to the files. This keyword reduces the file size at the expense of lower resolution in the time domain. There is no data required for this keyword.

See *RPTONLY - Activate the Report Time Steps Only Option for the SUMMARY File* in the SUMMARY section for a full description.

12.3.206 RPTONLYO - DEACTIVATE THE REPORT TIME STEPS ONLY OPTION FOR THE SUMMARY FILE

This keyword deactivates the writing out of the SUMMARY file and RSM file data, if the RSM file option has been requested by the RUNSUM keyword in the SUMMARY section, at report time steps only, and switches on writing out all the time steps to the files. This option is the default behavior for when RPTONLY has not been activated. There is no data required for this keyword.

See *RPTONLYO - Deactivate the Report Time Steps Only Option for the SUMMARY File* in the SUMMARY section for a full description.

12.3.207 RPTRST – DEFINE DATA TO BE WRITTEN TO THE RESTART FILE

This keyword defines the data to be written to the RESTART file at each requested restart point.

See *RPTRST – Define Data to be Written to the RESTART File* in the SOLUTION section for a full description.

12.3.208 RPTSCHED – DEFINE SCHEDULE SECTION REPORTING

Description

This keyword defines the data in the SCHEDULE section that is to be printed to the output print file in human readable format. The keyword has two distinct forms, the first of which consists of the keyword followed by a series of integers on the next line indicating the data to be printed (see the first example). This is the original format in the commercial simulator and was subsequently superseded by the second format. The second format consists of the keyword followed by a series of character strings that indicate the data to be printed. In most cases the character string is the keyword used to define the data in the OPM Flow input deck, for example WELSPECS to defined the basic well definitions. OPM Flow only supports the functionality of the second format, the first format although recognized will be completely ignored.

No.	Name	Description	Default
1	FIP	Print the fluid in-place report. The parameter is assigned a value, OPTION, using the form FIP = OPTION, where OPTION is an integer variable set to: 1) OPTION = 1 then the report is for the field only. 2) OPTION = 2 then in addition to the field report, a report is produced for each FIPNUM region, as defined by the FIPNUM keyword in the REGIONS section. Note the commercial simulator also prints the flows to other regions as well as the flows from the wells. This additional reporting option has not been implemented in OPM Flow. 3) OPTION = 3 then in addition to the above, a balance report is also produced for fluid in-place regions defined by the FIP keyword in the REGIONS section.	FIP=2
2	FIPRESV	Print the reservoir volumes in-place report.	None
3	NOTHING	Switches off all printed reports in the SCHEDULE section.	N/A
4	SALT	Print grid block salt concentration values. Note this is an OPM Flow specific keyword.	N/A
5	RESTART	RESTART defines the frequency at which the restart data for restarting a run is written to the RESTART file. The parameter is assigned a value, OPTION, using the form RESTART = OPTION, where OPTION is an integer variable set to: 1) OPTION = 1 then the restart files are written at every report time, but only the last one in the run is kept. This minimizes the restart file size but only the final results are stored, limiting the visualization in OPM ResInsight. 2) OPTION = 2 then the phase inter-blocks are written to the restart files, in addition to the standard data. 3) OPTION = 3 then the fluid in-place and phase potentials are also written to the restart file. 4) OPTION = 6 then the restart files are written at every time step. See the RPTRST keyword in the SOLUTION section for a more flexible way to write out restart files.	

No.	Name	Description	Default
** Expression is faulty **	WELLS	The WELLS option turns on production and injection rate and cumulative volume reporting for produced and injected fluids. The parameter has several levels of reporting details set by the assigned OPTION value, using the form WELLS = OPTION, where OPTION is an integer variable set to: 1) OPTION = 1 report volumes at the well level. 2) OPTION = 2 report volumes at the well and the well connection levels. 3) OPTION = 3 report volumes for layer totals. 4) OPTION = 4 report volumes for layer totals and for wells. 5) OPTION = 5 report volumes for layer totals and for wells and well connections. Only OPTION equal to one is supported by OPM Flow.	WELLS=1
** Expression is faulty **	WELSPECS	WELSPECS switches on reporting of the well connections, wells and groups at each report time step. There are numerous reports associated with this option. Unlike the other reporting parameters that produce a report for each reporting time step, the WELSPECS report option only produces a report if an associated keyword has been activated at the current reporting time step. For example, if the reporting time steps are January, February, and March 2020, and the RPTSCHED WELSPECS option is activated in January, with wells OP01 and OP02 being declared via the WELSPECS and COMPDAT keywords, then a report will be printed for January for these two wells. If there are no further well activations until March, with well OP03 being declared, then there will be no report for February, and only well OP03 will reported at the March reporting time step.	
Notes: 1) The keyword is terminated by a "/".			

Table 12.64: RPTSCHED Keyword Description

Development is current progressing on developing reports in a similar format to the commercial simulator and this section will be updated as additional reports are added to OPM Flow's functionality.

Note

Unlike the other reporting keywords in the RUNSPEC, GRID, EDIT, PROPS and SOLUTION keywords, the requested reports on the this keyword remain in effect until they are switched off by this keyword, that is, the reports are written out every report time step until requested to stop. Use the 'NOTHING' parameter to switch off all reporting.

An example FIP report is shown in Figure 12.6 from the Norne field, note only the field and the first two region reports are shown.

```
                                                  ================================================
                                                  :                Field Totals                  :
                                                  :      PAV  =      268.423 BARSA               :
                                                  :      PORV =    673813759   RM3               :
                                                  : Pressure is weighted by hydrocarbon pore volume :
                                                  : Porv volumes are taken at reference conditions  :
                        :--------------- Oil    SM3 ---------------:-- Wat    SM3 --:--------------- Gas    SM3 ---------------:
                        :      Liquid        Vapour        Total   :      Total     :      Free        Dissolved       Total   :
:-----------------------:----------------------------------------:----------------:----------------------------------------:
:Currently   in place   :      92852273        506017      93358290:    480610201   :   11202115432    9841722029   21043837461:
:-----------------------:----------------------------------------:----------------:----------------------------------------:
:Originally  in place   :     160196505        559893     160756399:    402039147   :    9744843573   17311090896   27055934469:
:=======================:========================================:================:========================================:

                                                  ================================================
                                                  :        FIPNUM report region   1              :
                                                  :      PAV  =      269.913 BARSA               :
                                                  :      PORV =    44729956   RM3               :
                        :--------------- Oil    SM3 ---------------:-- Wat    SM3 --:--------------- Gas    SM3 ---------------:
                        :      Liquid        Vapour        Total   :      Total     :      Free        Dissolved       Total   :
:-----------------------:----------------------------------------:----------------:----------------------------------------:
:Currently   in place   :       5308282        259439       5567722:     12179015   :    5263878362     594212703    5858091065:
:-----------------------:----------------------------------------:----------------:----------------------------------------:
:Originally  in place   :       5358346        311293       5669639:     11497281   :    5426018016     591714035    6017732051:
:=======================:========================================:================:========================================:

                                                  ================================================
                                                  :        FIPNUM report region   2              :
                                                  :      PAV  =      266.851 BARSA               :
                                                  :      PORV =    14745585   RM3               :
                        :--------------- Oil    SM3 ---------------:-- Wat    SM3 --:--------------- Gas    SM3 ---------------:
                        :      Liquid        Vapour        Total   :      Total     :      Free        Dissolved       Total   :
:-----------------------:----------------------------------------:----------------:----------------------------------------:
:Currently   in place   :       2744221         88023       2832244:      3078782   :    1655632008     318616757    1974248764:
:-----------------------:----------------------------------------:----------------:----------------------------------------:
:Originally  in place   :       3229038         88236       3317274:      3087870   :    1535013525     362544609    1897558134:
:=======================:========================================:================:========================================:
```

Figure 12.6: RPTSCHED: FIP (Balance) Report

Figure 12.7 illustrates the reservoir volumes in-place report for the first two regions from the Norne field.

```
                                                                  ===================================
                                                                  :  RESERVOIR VOLUMES      M3      :
:---------:---------------:---------------:---------------:---------------:---------------:
: REGION  :  TOTAL PORE   :  PORE VOLUME  :  PORE VOLUME  : PORE VOLUME   :  PORE VOLUME  :
:         :   VOLUME      :  CONTAINING   :  CONTAINING   : CONTAINING    :  CONTAINING   :
:         :               :     OIL       :    WATER      :    GAS         :  HYDRO-CARBON :
:---------:---------------:---------------:---------------:---------------:---------------

:        1:       44719142:        7048934:       11937133:       25733075:       32782009:
:---------:---------------:---------------:---------------:---------------:---------------:

:        2:       14747585:        4266286:        3206400:        7274899:       11541185:
:---------:---------------:---------------:---------------:---------------:---------------:
```

Figure 12.7: RPTSCHED: FIPRESV (Reservoir Volumes) Report

Note

Note that the "PORV" quantity in the FIP (Balance) Report, as shown in Figure 12.6, is reported at reference conditions, meaning there is no pressure dependence involved. However, the "TOTAL PORE VOLUME" values in the Reservoir Volumes Report (Figure 12.7) are pressure dependent pore volumes. Thus, for region one the "PORV" value is 44,729,956 rm3 (Figure 12.6) and the "TOTAL PORE VOLUME" (Figure 12.7) is 44,719,142 rm3.

This is the same as the commercial simulator.

The WELLS report consists of several sub-reports depending on the selected option for this report type. Figure 12.8 and Figure 12.9 show example Injection and Production sub-reports

INJECTION REPORT

WELL NAME	LOCATION (I,J,K)	CTRL MODE OIL	CTRL MODE WAT	CTRL MODE GAS	OIL RATE SCM/DAY	WATER RATE SCM/DAY	GAS RATE SCM/DAY	FLUID RES.VOL. RCM/DAY	BHP OR CON.PR. BARSA	THP OR BLK.PR. BARSA
FIELD:					0.0:	14237.0:	208381.1:	15765.5:		
INJE:					0.0:	14237.0:	208381.1:	15765.5:		
PROD:					0.0:	0.0:	0.0:	0.0:		
C-4H:	11, 35:			WRAT:	0.0:	0.0:	0.0:	0.0:	0.0:	0.0:
C-1H:	26, 44:			GRAT:	0.0:	0.0:	56404.2:	266.1:	277.9:	0.0:
C-2H:	23, 15:			WRAT:	0.0:	0.0:	0.0:	0.0:	264.1:	0.0:
C-3H:	9, 13:			GRAT:	0.0:	0.0:	143344.6:	676.4:	265.0:	0.0:
F-1H:	12, 85:			WRAT:	0.0:	9662.4:	0.0:	10032.4:	493.3:	0.0:
F-2H:	18, 83:			WRAT:	0.0:	4574.6:	0.0:	4749.8:	475.1:	0.0:
F-3H:	6, 57:			WRAT:	0.0:	0.0:	0.0:	0.0:	306.7:	0.0:
F-4H:	36, 68:			WRAT:	0.0:	0.0:	0.0:	0.0:	510.4:	0.0:
C-4AH:	29, 51:			GRAT:	0.0:	0.0:	8632.4:	40.7:	282.5:	0.0:

Figure 12.8: RPTSCHED: WELLS - Injection Sub-Report

PRODUCTION REPORT

WELL NAME	LOCATION (I,J,K)	CTRL MODE	OIL RATE SCM/DAY	WATER RATE SCM/DAY	GAS RATE SCM/DAY	FLUID RES.VOL. RCM/DAY	WATER CUT SCM/SCM	GAS/OIL RATIO SCM/SCM	WAT/GAS RATIO SCM/SCM	BHP OR CON.PR. BARSA	THP OR BLK.PR. BARSA
FIELD:			7281.1:	16714.6:	2287465.9:	33859.3:	0.697:	314.16:	0.0073:		
INJE:			-0.0:	-0.0:	-0.0:	-0.0:	0.000:	0.00:	0.0000:		
PROD:			7281.1:	16714.6:	2287465.9:	33859.3:	0.697:	314.16:	0.0073:		
B-2H:	15, 31:	RESV:	767.0:	1987.2:	106848.9:	3176.7:	0.722:	139.30:	0.0186:	169.7:	0.0:
D-1H:	22, 22:	ORAT:	-0.0:	-0.0:	-0.0:	-0.0:	0.000:	0.00:	0.0000:	0.0:	0.0:
D-2H:	14, 28:	RESV:	1017.2:	2138.4:	343553.3:	4637.7:	0.678:	337.75:	0.0062:	164.9:	0.0:
B-4H:	10, 32:	ORAT:	-0.0:	-0.0:	-0.0:	-0.0:	0.000:	0.00:	0.0000:	0.0:	0.0:
D-4H:	19, 38:	ORAT:	-0.0:	-0.0:	-0.0:	-0.0:	0.000:	0.00:	0.0000:	0.0:	0.0:
E-3H:	12, 72:	ORAT:	-0.0:	-0.0:	-0.0:	-0.0:	0.000:	0.00:	0.0000:	0.0:	0.0:
B-1H:	14, 34:	ORAT:	0.0:	0.0:	0.0:	0.0:	0.252:	128.50:	0.0026:	243.0:	0.0:
B-3H:	9, 37:	RESV:	8.7:	77.0:	931.9:	91.3:	0.899:	107.22:	0.0827:	242.9:	0.0:
E-1H:	18, 68:	RESV:	610.7:	6251.1:	66970.5:	7292.8:	0.911:	109.67:	0.0933:	185.6:	0.0:

Figure 12.9: RPTSCHED: WELLS - Production Sub-Report

The third and final report is the Cumulative Production and Injection sub-report, shown in Figure 12.10.

CUMULATIVE PRODUCTION/INJECTION REPORT

WELL NAME	LOCATION (I,J,K)	WELL TYPE	CTRL: MODE:	OIL PROD MSCM	WATER PROD MSCM	GAS PROD MMSCM	Prod RES.VOL. MRCM	OIL INJ MSCM	WATER INJ MSCM	GAS INJ MMSCM	INJ RES.VOL. MRCM
FIELD:				67201.8:	23494.0:	14651446.1:	149177.9:	0.0:	102036.7:	8684539.9:	149925.7:
INJE:				0.0:	178.7:	0.0:	185.7:	0.0:	102036.7:	8684539.9:	149925.7:
PROD:				67201.8:	23315.3:	14651446.1:	148992.2:	0.0:	0.0:	0.0:	0.0:
C-4H:	11, 35:	INJ:WRAT:		0.0:	0.0:	0.0:	0.0:	0.0:	5468.3:	2927677.5:	20664.3:
B-2H:	15, 31:	PROD:RESV:		10803.4:	1766.2:	1840432.3:	19370.5:	0.0:	0.0:	0.0:	0.0:
D-1H:	22, 22:	PROD:ORAT:		5497.5:	507.5:	908176.2:	9406.7:	0.0:	0.0:	0.0:	0.0:
D-2H:	14, 28:	PROD:RESV:		7533.6:	1245.2:	2698602.4:	20280.4:	0.0:	0.0:	0.0:	0.0:
B-4H:	10, 32:	PROD:ORAT:		1307.0:	0.6:	269357.3:	2383.2:	0.0:	0.0:	0.0:	0.0:
D-4H:	19, 38:	PROD:ORAT:		3179.7:	84.5:	680216.1:	6036.1:	0.0:	0.0:	0.0:	0.0:
C-1H:	26, 44:	INJ:GRAT:		0.0:	0.0:	0.0:	0.0:	0.0:	12795.6:	2186307.8:	24278.4:
E-3H:	12, 72:	PROD:ORAT:		1687.3:	198.9:	179311.1:	2441.4:	0.0:	0.0:	0.0:	0.0:
C-2H:	23, 15:	INJ:WRAT:		0.0:	0.0:	0.0:	0.0:	0.0:	21661.2:	0.0:	22504.3:
B-1H:	14, 34:	PROD:ORAT:		3581.4:	476.1:	1413055.5:	10326.8:	0.0:	0.0:	0.0:	0.0:

Figure 12.10: RPTSCHED: WELLS - Cumulative Production and Injection Sub-Report

Similarly as for the WELLS report, the WELSPECS report consists of several sub-reports, including the Well Production Control report shown in Figure 12.11 for the Volve field.

WELSPECS AT 0.00 DAYS * * FLOW
REPORT 0 31 DEC 2007 * * RUN AT 12:41 ON 12 SEP 2016

WELL SPECIFICATION DATA

WELL NAME	GROUP NAME	WELLHEAD LOCATION (I, J)	B.H.REF DEPTH METRES	PREF- ERRED PHASE	DRAINAGE RADIUS METRES	GAS INFL EQUN	SHUT-IN INSTRCT	CROSS FLOW ABLTY	PVT TAB	WELL DENS CALC	FIP REG	WELL D-FACTOR 1 DAY/SM3
I-F-5	AAR	84, 41	2997.3	WAT	P.EQUIV.R:	STD	OPEN	YES	4	SEG	0	0
P-F-14	AAR	65, 48	2476.1	OIL	P.EQUIV.R:	STD	OPEN	YES	2	SEG	0	0
I-F-4	SRAR	73, 28	2931.0	WAT	P.EQUIV.R:	STD	OPEN	YES	3	SEG	0	0
I-F4G	SRAR	73, 28	2931.0	GAS	P.EQUIV.R:	STD	OPEN	YES	3	SEG	0	0
P-F-12	SRAR	58, 37	2705.0	OIL	P.EQUIV.R:	STD	OPEN	YES	1	SEG	0	0
P-F-15C	SRAR	55, 28	2551.0	OIL	P.EQUIV.R:	STD	OPEN	YES	11	SEG	0	0
P-F-11B	SRAR	66, 42	2451.0	OIL	P.EQUIV.R:	STD	SHUT	YES	2	SEG	0	0
I-F-1B	P_NW	66, 58	2988.5	WAT	P.EQUIV.R:	STD	SHUT	YES	6	SEG	0	0
P-F-15D	AAR	63, 47	2844.9	OIL	P.EQUIV.R:	STD	SHUT	YES	2	SEG	0	0
P-F-1C	P_NW	57, 51	2889.1	OIL	P.EQUIV.R:	STD	SHUT	YES	10	SEG	0	0
PIL-N	P_NORTH	80, 54	3153.9	OIL	P.EQUIV.R:	STD	OPEN	YES	5	SEG	0	0
PIL-NW	P_NW	64, 59	2959.5	OIL	P.EQUIV.R:	STD	SHUT	YES	6	SEG	0	0

1: The WELL D-FACTOR is not implemented - and the report will always show the default value 0.

Figure 12.11: RPTSCHED: WELSPECS - Production Well Control Sub-Report

The COMPDAT keyword data is listed on the Well Connection Data sub-report as depicted in Figure 12.12 for the I-F-5 well. Note that the data is repeated for all connections and for all wells declared at the reporting time step.

WELL CONNECTION DATA

WELL NAME	GRID BLOCK	CMPL: NO#	CENTRE DEPTH METRES	OPEN: SHUT:	SAT TAB	CONNECTION: FACTOR* CPM3/D/B	INT DIAM METRES	K H VALUE MD.METRE	SKIN FACTOR	CONNECTION D-FACTOR 1 DAY/SM3	SATURATION SCALING DATA SWMIN SWMAX SGMIN SGMAX 2:
I-F-5	84, 41, 1	1	2997.3	SHUT:	1	0.000000	0.216000	0.000000:0.000000:		0	
I-F-5	84, 41, 2	2	2998.3	SHUT:	1	0.000000	0.216000	0.000000:0.000000:		0	
I-F-5	84, 41, 3	3	2999.4	SHUT:	1	0.000000	0.216000	0.000000:0.000000:		0	
I-F-5	84, 41, 4	4	3000.4	SHUT:	1	0.000000	0.216000	0.000000:0.000000:		0	
I-F-5	84, 41, 5	5	3001.5	SHUT:	1	0.000000	0.216000	0.000000:0.000000:		0	
I-F-5	84, 41, 6	6	3002.5	SHUT:	1	0.000000	0.216000	0.000000:0.000000:		0	
I-F-5	84, 41, 7	7	3003.6	SHUT:	1	0.000000	0.216000	0.000000:0.000000:		0	
I-F-5	84, 41, 8	8	3004.6	SHUT:	1	0.000000	0.216000	0.000000:0.000000:		0	
I-F-5	84, 41, 9	9	3005.6	SHUT:	1	0.000000	0.216000	0.000000:0.000000:		0	
I-F-5	84, 41, 10	10	3006.6	SHUT:	1	0.000000	0.216000	0.000000:0.000000:		0	
I-F-5	84, 41, 11	11	3007.7	SHUT:	1	0.000000	0.216000	0.000000:0.000000:		0	
I-F-5	84, 41, 12	12	3008.8	SHUT:	1	0.000000	0.216000	0.000000:0.000000:		0	
I-F-5	84, 41, 13	13	3009.8	SHUT:	1	0.000000	0.216000	0.000000:0.000000:		0	
I-F-5	84, 41, 14	14	3010.9	SHUT:	1	0.000000	0.216000	0.000000:0.000000:		0	
I-F-5	84, 41, 15	15	3012.0	SHUT:	1	0.000000	0.216000	0.000000:0.000000:		0	
I-F-5	85, 41, 17	18	3035.9	SHUT:	1	0.000000	0.216000	0.000000:0.000000:		0	
I-F-5	85, 41, 19	19	3038.6	SHUT:	1	0.000000	0.216000	0.000000:0.000000:		0	
I-F-5	85, 41, 20	20	3040.0	OPEN:	1	0.000000	0.216000	0.000000:0.000000:		0	
I-F-5	85, 42, 20	21	3050.7	OPEN:	1	0.000000	0.216000	0.000000:0.000000:		0	
I-F-5	85, 42, 21	22	3051.9	OPEN:	1	0.000000	0.216000	0.000000:0.000000:		0	
I-F-5	85, 42, 22	23	3053.0	OPEN:	1	0.000000	0.216000	0.000000:0.000000:		0	
I-F-5	85, 42, 23	24	3054.2	OPEN:	1	0.000000	0.216000	0.000000:0.000000:		0	
I-F-5	85, 42, 24	25	3055.4	OPEN:	1	0.000000	0.216000	0.000000:0.000000:		0	
I-F-5	85, 42, 25	26	3056.6	OPEN:	1	0.000000	0.216000	0.000000:0.000000:		0	
I-F-5	85, 42, 26	27	3058.1	OPEN:	1	0.000000	0.216000	0.000000:0.000000:		0	
I-F-5	85, 42, 27	28	3059.9	OPEN:	1	0.000000	0.216000	0.000000:0.000000:		0	
I-F-5	85, 42, 28	29	3061.7	OPEN:	1	0.000000	0.216000	0.000000:0.000000:		0	
I-F-5	85, 42, 29	30	3063.6	OPEN:	1	0.000000	0.216000	0.000000:0.000000:		0	
I-F-5	85, 42, 30	31	3065.4	OPEN:	1	0.000000	0.216000	0.000000:0.000000:		0	

1: The well connection D-FACTOR is not implemented in opm and the report will always show 0.
2: The saturation scaling data has not been implemented in the report and will show blank

Figure 12.12: RPTSCHED: WELSPECS - Well Connection Data Sub-Report

For multi-segment wells both the Production Well Control and Well Connection sub-reports are printed as per Figure 12.11 and Figure 12.12, and in addition the equivalent multi-segment well data is printed as well, as shown in Figure 12.13 and Figure 12.14, as shown on the following page.

Figure 12.13 shows the Multi-Segment Well Segment Structure sub-report for a single multi-segment well, OP01. See the WELSEGS keyword *Example* in the SCHEDULE section to see how the OP01 well is defined.

MULTI-SEGMENT WELL: SEGMENT STRUCTURE

WELLNAME AND SEG TYPE	SEG NO	BRN NO	MAIN INLET SEGMENT	OUTLET SEGMENT	SEGMENT LENGTH METRES	TOT LENGTH: TO END METRES	DEPTH CHANGE METRES	T.V. DEPTH: AT END METRES	DIA OR F: SCALING: METRES	VFP TAB OR: ABS ROUGHN: METRES	AREA X-SECTN M**2	VOLUME M3	P DROP MULT FACTOR
OP01	1	1	2	0		2512.5		2512.5	0	0	0	0.000	
HFA:	2		3	1		2537.5		2534.5	0.3000	0.000100	0.07068	1.767	
	3		4	2		2562.5		2560.5	0.3000	0.000100	0.07068	1.767	
	4		5	3		2587.5		2593.5	0.3000	0.000100	0.07068	1.767	
	5		6	4		2612.5		2614.5	0.3000	0.000100	0.07068	1.767	
	6		22	5		2637.5		2635.5	0.3000	0.000100	0.07068	1.767	
	7	2	8	2		2737.5		2538.5	0.2000	0.000100	0.03141	6.283	
	8		9	7		2937.5		2537.5	0.2000	0.000100	0.03141	6.283	
	9		10	8		3137.5		2539.5	0.2000	0.000100	0.03141	6.283	
	10		11	9		3337.5		2535.5	0.2000	0.000100	0.03141	6.283	
	11		0	10		3537.5		2536.5	0.2000	0.000100	0.03141	6.283	
	12	3	13	3		2762.5		2563.5	0.2000	0.000100	0.03141	6.283	
	13		14	12		2962.5		2562.5	0.1000	0.000100	0.00785	1.570	
	14		15	13		3162.5		2562.5	0.1000	0.000100	0.00785	1.570	
	15		16	14		3362.5		2564.5	0.1000	0.000100	0.00785	1.570	
	16		0	15		3562.5		2562.5	0.1000	0.000100	0.00785	1.570	
	17	4	18	5		2812.5		2613.5	0.2000	0.000100	0.03141	6.283	
	18		19	17		3012.5		2612.5	0.1000	0.000100	0.00785	1.570	
	19		20	18		3212.5		2612.5	0.1000	0.000100	0.00785	1.570	
	20		21	19		3412.5		2612.5	0.1000	0.000100	0.00785	1.570	
	21		0	20		3612.5		2613.5	0.1000	0.000100	0.00785	1.570	
	22	5	23	6		2837.5		2634.5	0.2000	0.000100	0.03141	6.283	
	23		24	22		3037.5		2637.5	0.2000	0.000100	0.03141	6.283	
	24		25	23		3237.5		2638.5	0.2000	0.000100	0.03141	6.283	
	25		26	24		3437.5		2639.5	0.1000	0.000100	0.00785	1.570	
	26		0	25		3637.5		2639.5	0.1000	0.000100	0.00785	1.570	

Figure 12.13: RPTSCHED: WELSPECS - Multi Segment Well Structure Sub-Report

And Figure 12.14 depicts Multi-Segment Well Connection Data sub-report for the same well.

MULTI-SEGMENT WELL: CONNECTION DATA

WELL NAME	CONNECTION:	SEGMENT NUMBER	BRANCH ID	TUB LENGTH: START PERFS: METRES	TUB LENGTH: END PERFS METRES	TUB LENGTH: CENTR PERFS: METRES	TUB LENGTH: END SEGMT METRES	CONNECTION: DEPTH METRES	SEGMENT DEPTH METRES	GRID BLOCK: DEPTH METRES
OP01	10, 10, 1:	1	1				2512.5	2512.5	2512.5	2512.5
	10, 10, 2:	2	1				2537.5	2534.5	2534.5	2537.5
	10, 10, 3:	3	1				2562.5	2560.5	2560.5	2562.5
	10, 10, 4:	4	1				2587.5	2593.5	2593.5	2587.5
	10, 10, 5:	5	1				2612.5	2614.5	2614.5	2612.5
	10, 10, 6:	6	1				2637.5	2635.5	2635.5	2637.5
	9, 10, 2:	7	2				2737.5	2538.5	2538.5	2537.5
	8, 10, 2:	8	2				2937.5	2537.5	2537.5	2537.5
	7, 10, 2:	9	2				3137.5	2539.5	2539.5	2537.5
	6, 10, 2:	10	2				3337.5	2535.5	2535.5	2537.5
	5, 10, 2:	11	2				3537.5	2536.5	2536.5	2537.5
	10, 9, 3:	12	3				2762.5	2563.5	2563.5	2562.5
	10, 8, 3:	13	3				2962.5	2562.5	2562.5	2562.5
	10, 7, 3:	14	3				3162.5	2562.5	2562.5	2562.5
	10, 6, 3:	15	3				3362.5	2564.5	2564.5	2562.5
	10, 5, 3:	16	3				3562.5	2562.5	2562.5	2562.5
	9, 10, 5:	17	4				2812.5	2613.5	2613.5	2612.5
	8, 10, 5:	18	4				3012.5	2612.5	2612.5	2612.5
	7, 10, 5:	19	4				3212.5	2612.5	2612.5	2612.5
	6, 10, 5:	20	4				3412.5	2612.5	2612.5	2612.5
	5, 10, 5:	21	4				3612.5	2613.5	2613.5	2612.5
	10, 9, 6:	22	5				2837.5	2634.5	2634.5	2637.5
	10, 8, 6:	23	5				3037.5	2637.5	2637.5	2637.5
	10, 7, 6:	24	5				3237.5	2638.5	2638.5	2637.5
	10, 6, 6:	25	5				3437.5	2639.5	2639.5	2637.5
	10, 5, 6:	26	5				3637.5	2639.5	2639.5	2637.5

Figure 12.14: RPTSCHED: WELSPECS - Multi Segment Well Connection Sub-Report

Example

The first example shows the original format of this keyword; although the keyword and format are recognized by OPM Flow, the format is ignored and is unlikely to be implemented in in the simulator.

```
--
--       DEFINE SCHEDULE SECTION REPORT OPTION (ORIGINAL FORMAT)
--
RPTSCHED
         1        2*0      1        3*1                                    /
```

The next example shows the second format of the keyword which is supported by OPM Flow.

```
-- ================================================================================
--
-- SCHEDULE SECTION
--
-- ================================================================================
SCHEDULE

-- --------------------------------------------------------------------------------
-- SCHEDULE SECTION - 2000-01-01
-- --------------------------------------------------------------------------------
RPTSCHED
         'WELLS=2'    'WELSPECS'    'CPU=2'     'FIP=2'                        /

DATES
         1  JAN   2000  /
/

RPTSCHED
         'NOTHING'                                                            /

DATES
         1  FEB   2000  /
         1  MAR   2000  /
         1  APR   2000  /
         1  MAY   2000  /
         1  JUN   2000  /
         1  JLY   2000  /
         1  AUG   2000  /
         1  SEP   2000  /
         1  OCT   2000  /
         1  NOV   2000  /
         1  DEC   2000  /
/
```

In the above example monthly reporting time steps have been used with a SCHEDULE section report on the January 1, 2000; after which all reports are switched off for the subsequent reporting time steps.

12.3.209 SAVE – ACTIVATE OUTPUT OF A SAVE FILE FOR FAST RESTARTS

This keyword activates output of a SAVE file for fast restarts. There is no data required for this keyword.

See *SAVE – Activate Output of a SAVE File for Fast Restarts* in the RUNSPEC section for a full description.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

12.3.210 SCDATAB – WELL CONNECTION PI MULTIPLIERS VERSUS SCALE DEPOSIT

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

SCDATAB defines well connection Productivity Index ("PI") reduction multipliers versus scale deposited per unit length of the perforated interval tables, for when the Scale Deposition option has been activated by declaring the dimensions of the scaling deposition tables using the SCDPDIMS keyword in the RUNSPEC section. The SCDATAB tables are allocated to individual wells using the WSCTAB keyword and the rate of scale accumulation around the well connections is given by the SCDPTAB keyword; both keywords are in the SCHEDULE section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.211 SCDETAB – WELL CONNECTION KARST AQUIFER PROPERTIES FOR SCALE DEPOSIT

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

SCDETAB defines well connection karst[318] aquifer properties for modeling scale deposited by dissolution of calcite from the aquifer water, for when the Scale Deposition option has been activated by declaring the dimensions of the scaling deposition tables using the SCDPDIMS keyword in the RUNSPEC section. The SCDETAB tables are allocated to individual wells using the WSCTAB keyword in the SCHEDULE section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

[318] *Karst is a topography formed from the dissolution of soluble rocks such as limestone, dolomite, and gypsum. Karst aquifers are characterized by a network of conduits and caves, with the conduits and caves draining the pore space between the limestone grains (intergranular or primary porosity) and the fractures (secondary porosity) formed by joints, bedding planes, and faults.*

12.3.212 SCDPTAB – WELL CONNECTION SCALE DEPOSITION TABLES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

SCDATAB defines the well connection scale deposition rate as a function of sea water flow rate, for when the Scale Deposition option has been activated by declaring the dimensions of the scaling deposition tables using the SCDPDIMS keyword in the RUNSPEC section. The SCDATAB tables are allocated to individual wells using the WSCTAB keyword and the sea water fraction is based on a water tracer entered via the SCDPTRAC keyword; both keywords are in the SCHEDULE section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.213 SCDPTRAC – ALLOCATE SEA WATER TRACER FOR SCALE DEPOSITION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The SCDPTRAC keyword is used to allocate an existing passive water tracer defined by the TRACER keyword in the PROPS section, to represent the sea water flowing into a well connection as a fraction of the total water influx. The keyword is used together with the SCDPTAB keyword in the SCHEDULE section to calculated the volume of scale deposited around the well connections.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	NAME	A three letter character string defining the tracer's name that has previously been defined by the TRACER keyword in the PROPS section			None
Notes: 1) The keyword is terminated by a "/".					

Table 12.65: SCDPTRAC Keyword Description

Example

In the PROPS section define a tracer in the water phase, for example:

```
--
--        DEFINE TRACER NAMES
--
--        TRACER    TRACER
--        NAME      PHASE
--        ------    ------
TRACER
         'SEA'      'WAT'                                  / SEA WATER TRACER

/
```

Then in the SCHEDULE section allocate the previously defined water tracer as a sea water tracer to be used with the scale deposition facility, that is:

```
--
--        ALLOCATE SEA WATER TRACER FOR SCALE DEPOSITION
--
--        TRACER
--        NAME
--        ------
SCDPTRAC
         'SEA'                                            / SEA WATER TRACER

/
```

12.3.214 SCHEDULE - DEFINE THE START OF THE SCHEDULE SECTION OF KEYWORDS

Description

The SCHEDULE activation keyword marks the end of the SUMMARY section and the start of the SCHEDULE section that defines the group and well definitions, operating and economic constraints, as well as how OPM Flow should advance through time. Numerical controls are also defined in this section and all parameters can be varied through time.

There is no data required for this keyword.

Example

```
-- ======================================================================
--
-- SCHEDULE SECTION
--
-- ======================================================================
SCHEDULE
```

The above example marks the end of the SUMMARY section and the start of the SCHEDULE section in the OPM Flow data input file.

12.3.215 SEPVALS – DEFINE SEPARATOR OIL FORMATION VOLUME FACTOR AND GOR

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The SEPVALS keyword defines the initial and subsequent separator oil formation volume factor (Bo) and Gas Oil Ratio ("GOR" or Rs). The facility is used in black-oil modeling to re-scale the PVT data entered via the PROPS section, based on the saturation point oil formation volume factor (Bob) and the initial saturated gas-oil ratio (Rsi) entered on the SEVPALS keyword. The first occurrence of this keyword sets the initial conditions and must be followed by the GSEPCOND keyword that assigns previously defined separators to a group.

Note that the keyword can only be used in runs with oil and dissolve gas only, with no vaporized oil (condensate) in the gas phase.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.216 SHRATE - ACTIVATE AND DEFINE THE POLYMER SHEARING LOGARITHMIC PARAMETERS

This keyword activates the logarithm-based polymer shear thinning/thickening option and defines the shear rate constant. This keyword can only be used in conjunction with the PLYSHLOG in the PROPS section

See *SHRATE - Activate and Define the Polymer Shearing Logarithmic Parameters* in the PROPS section for a full description.

12.3.217 SIMULATE - ACTIVATE THE SIMULATION MODE

Description

SIMULATE switches the mode of the simulation to run simulation mode from the data input checking mode activated by the NOSIM keyword in the SCHEDULE section. Note that if NOSIM has been used in the RUNSPEC section then SIMULATE will have no effect.

There is no data required for this keyword and there is no terminating "/" for this keyword.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

Example

The example below switches OPM Flow to no simulation mode for data checking of the input deck.

```
--
--       ACTIVATE SIMULATION MODE TO RUN THE MODEL
--
SIMULATE
```

12.3.218 SKIP – ACTIVATE SKIPPING OF ALL KEYWORDS AND INPUT DATA

The SKIP keyword turns on skipping of keywords until the ENDSKIP activation keyword is encountered. All keywords and between the SKIP and ENDSKIP keywords are ignored by OPM Flow.

See *SKIP – Activate Skipping of All Keywords and Input Data* in the GLOBAL section for a full description.

12.3.219 SKIP100 – ACTIVATE SKIPPING OF BLACK-OIL KEYWORDS AND INPUT DATA

This keyword turns on skipping of black-oil keywords until the ENDSKIP activation keyword is encountered. All black-oil keywords between the SKIP100 and ENDSKIP keywords are ignored by OPM Flow.

See *SKIP100 – Activate Skipping of Black-Oil Keywords and Input Data* in the GLOBAL section for a full description.

12.3.220 SKIP300 – ACTIVATE SKIPPING OF "COMPOSITIONAL" KEYWORDS AND INPUT DATA

Turns on skipping of "compositional" keywords until the ENDSKIP activation keyword is encountered. All "compositional" keywords between the SKIP300 and ENDSKIP keywords are ignored by OPM Flow.

See *Error: Reference source not found* in the GLOBAL section for a full description.

12.3.221 SKIPREST – ACTIVATE SKIPPING OF RESTART SCHEDULE DATA

Description

This keyword turns on skipping of keywords up to the start of the restart point, as defined on the RESTART keyword in the RUNSPEC section. The RESTART keyword defines the parameters to restart the simulation from a previous run that has written a RESTART file out to disk. Activating the SKIPREST keyword causes the simulator to only read in data it requires for restarting the run up to the RESTART point (RSNUM on the RESTART keyword in the SOLUTION section). Note that certain keywords always need to be present in a restart run in the SCHEDULE section as the data is not stored on the RESTART file, for example the VFP tables (VFPPROD and VFPINJ keywords). The SKIPREST keyword automatically processes the input deck and reads the required data.

There is no data required for this keyword.

Example

The example below defines a restart from the previously run NOR-OPM-A01 case at time step number 40.

```
-- ==================================================================================
--
-- SOLUTION SECTION
--
-- ==================================================================================
SOLUTION
--
--       FLEXIBLE RESTART FROM PREVIOUS SIMULATION RUN
--
--       FILE                   RESTART   RESTART   FILE
--       NAME                   NUMBER    TYPE      FORMAT
RESTART
         'NOR-OPM-A01'          40        1*        1*     /
```

Then in the SCHEDULE section the SKIPREST keyword is used to correctly read in the schedule data up to the RESTART point.

```
-- ==================================================================================
--
-- SCHEDULE SECTION
--
-- ==================================================================================
SCHEDULE
--
--       ACTIVATE SKIPREST OPTION TO AVOID MODIFYING SCHEDULE SECTION
--
SKIPREST
```

12.3.222 SLAVES – DEFINE SLAVE RESERVOIR SIMULATION PARAMETERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, SLAVES, defines the name of the slave reservoirs and their associated simulation input files, for when the Reservoir Coupling option has been declared active by the GRUPMAST and SLAVES keywords in the SCHEDULE section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.223 SUMTHIN – DEFINE SUMMARY DATA REPORTING TIME STEPS

This keyword defines a time interval for writing out the SUMMARY data to the SUMMARY file and the RSM file, if the RUNSUM keyword has been has also been activated in the SUMMARY section. Only the data for the first time step in the time interval is written out and the other time steps are skipped until the next time interval. This enable the size of the SUMMARY files to be reduced depending on the size of the time interval. However, the keyword will produce irregular time steps reports of the SUMMARY data.

See *SUMTHIN – Define SUMMARY Data Reporting Time Steps* in the SUMMARY section for a full description.

12.3.224 SURFVISC – SURFACTANT SOLUTION VISCOSITY VERSUS CONCENTRATION

SURFSVISC defines the surfactant viscosity relationship of solution water viscosity with respect to increasing surfactant concentration within a grid block. The surfactant option must be activated by the SURFACT keyword in the RUNSPEC section in order to use this keyword.

See *SURFVISC – Surfactant Solution Viscosity versus Concentration* in the PROPS section for a full description.

12.3.225 SWINGFAC – DEFINE FIELD GAS CONTRACT PARAMETERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, SWINGFAC, defines the gas contract parameters, swing factor and the monthly seasonal profile factor, for when there is a single gas contract being used in the model. The keyword is used with the Gas Field Operations option which is activated by the GASFIELD keyword in the RUNSPEC section. Gas contracts are commonly based on a Daily Contract Quantity ("DCQ") that determines the gas rate that the field should be produced at, which is normally expressed as a multiple of the DCQ, for example 1.33, and is often referred to as the "swing factor". Some gas contracts also define a maximum DCQ ("Max DCQ") and/or a minimum take or pay DCQ ("Min DCQ"), as well as seasonal demand characteristics. For example, gas rates may be set higher in the winter months in order to meet heating demand compared with summer months in colder climates, and the opposite in warmer climates where air conditioning demand is high.

Thus, the DCQ must be calculated first over a contract period, where the contract period is commonly contract years, or in some instances contract quarters. This is performed by the simulator using the current DCQ and checking to see if the (DCQ x Swing Factor) can be satisfied throughout the current contract period, if not the DCQ is re-calculated so that the (DCQ x Swing Factor) condition is satisfied. Once this condition is met, the second and final pass uses the calculated DCQ in conjunction with the monthly scaling profile data to set the monthly gas rate for the field:

$$Q_{month} = DCQ \times SWINGFAC_{month} \tag{12.32}$$

Where:

Qmonth = the monthly gas production target
DCQ = Daily Contract Quantity
$SWINGFAC_{month}$ = monthly rate scaling factor that takes into account seasonal demand, etc.

Here the SWINGFAC keyword sets the gas contract parameters for a single contract at the FIELD group level and is mutually exclusive to GSWINGF keyword in the SCHEDULE section that allows for different gas contract parameters to be assign to different groups.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.226 TIGHTEN – TIGHTEN AND RELAX NUMERICAL CONTROLS

Description

The TIGHTEN keyword tightens up or slackens the numerical controls for the linear, non-linear and material balance convergence targets and also tightens or relaxes the maximum values for the aforementioned parameters. The keyword should be used with caution as it may result in significantly increasing the run times.

Note that any subsequent use of the TUNING keyword in the SCHEDULE section will result in resetting the numerical controls. See also the TIGHTENP in the SCHEDULE section that allows for greater flexibility in modifying the numerical controls.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.227 TIGHTENP – TIGHTEN AND RELAX NUMERICAL CONTROLS INDIVIDUALLY

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The TIGHTENP keyword is similar to the TIGHTEN keyword in the SCHEDULE section, in that it tightens up or slackens the numerical controls for the linear, non-linear and material balance convergence targets and also tightens or relaxes the maximum values for the aforementioned parameters. However, TIGHTENP allows for greater flexibility as there are four parameters on this keyword, as opposed to just one on the TIGHTEN keyword, that can be used to modify the numerical controls. The keyword should be used with caution as it may result in significantly increasing the run times.

Note that any subsequent use of the TUNING keyword in the SCHEDULE section will result in resetting the numerical controls. See also the TIGHTEN keyword in the SCHEDULE section that has more limited flexibility in modifying the numerical controls.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.228 TIME – ADVANCE SIMULATION BY CUMULATIVE REPORTING TIME

Description

This keyword advances the simulation to a given cumulative report time after which additional keywords may be entered to instruct OPM Flow to perform additional functions via the SCHEDULE section keywords, or further TIME keywords may be entered to advance the simulator to the next report time.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	TIME	A vector of real positive numbers that define the cumulative length of the of report times.			
		days	days	hours	None

Notes:

1) The keyword is followed by a vector of numbers separated by a space and the keyword is terminated by a "/".

Table 12.66: TIME Keyword Description

See also the DATES and TSTEP keyword in the SCHEDULE section. Note that since OPM Flow uses the standard Gregorian calendar and therefore leap years are accounted for in the DATES keyword. Thus, it is more accurate to use the DATES keyword to progress the simulator through time if one is matching actual production data.

Whenever possible it is a good idea to always set the start date to be at the beginning of the year, as like most simulators, OPM Flow reports are always stated at the number of days from the start date (and sometimes at a given date). If the start date is at the beginning of the year, then calculating the actual date is relatively straight forward and simple.

Examples

The fist example shows how to advance the simulation three years using the TIME keyword, from the given start date of January 1, 2022 set via the START keyword in the RUNSPEC section.

```
-- ================================================================================
--
-- SCHEDULE SECTION
--
-- ================================================================================
SCHEDULE

-- --------------------------------------------------------------------------------
-- SCHEDULE SECTION - 2022-01-01
-- --------------------------------------------------------------------------------
--
--       ADVANCE SIMULATION BY REPORTING TIME
--
TIME
        365.25  730.50  1095.75
/
```

The second example shows the same advance but using the TSTEP keyword instead.

```
-- ======================================================================
--
-- SCHEDULE SECTION
--
-- ======================================================================
SCHEDULE

-- ----------------------------------------------------------------------
-- SCHEDULE SECTION - 2022-01-01
-- ----------------------------------------------------------------------
RPTSCHED
          'WELLS=2'    'WELSPECS'    'CPU=2'     FIP=2'                        /
--
--       ADVANCE SIMULATION BY REPORTING TIME
--
TSTEP
         3*365.25
/
```

Again, if the simulated production targets are actual production data or the results are going to be used in economic evaluations then the DATES keyword may be more useful in advancing the simulation via the reporting time steps, as the exact dates will be honored.

12.3.229 TSTEP – ADVANCE SIMULATION BY REPORTING TIME

Description

This keyword advances the simulation to a given report time after which additional keywords may be entered to instruct OPM Flow to perform additional functions via the SCHEDULE section keywords, or further TSTEP keywords may be entered to advance the simulator to the next report time.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	TSTEP	A vector of real positive numbers that define the length of the time intervals to subsequent report steps. Repeat counts may be used, for example 10*365.25.			
		days	days	hours	None
Notes: 1) The keyword is followed by a vector of numbers separated by a space and the keyword is terminated by a "/".					

Table 12.67: TSTEP Keyword Description

See also the DATES and TIME keywords in the SCHEDULE section. Note that since OPM Flow uses the standard Gregorian calendar and therefore leap years are accounted for in the DATES keyword. Thus, it is more accurate to use the DATES keyword to progress the simulator through time if one is matching actual production data.

Whenever possible it is a good idea to always set the start date to be at the beginning of the year, as like most simulators, OPM Flow reports are always stated at the number of days from the start date (and sometimes at a given date). If the start date is at the beginning of the year, then calculating the actual date is relatively straight forward and simple.

Examples

The first example shows how to advance the simulation via the reporting time steps from the given start date of January 1, 2022 set via the START keyword in the RUNSPEC section, to the next year, without any actions or reporting taking place.

```
-- ================================================================================
--
-- SCHEDULE SECTION
--
-- ================================================================================
SCHEDULE

-- --------------------------------------------------------------------------------
-- SCHEDULE SECTION - 2022-01-01
-- --------------------------------------------------------------------------------
--
--       ADVANCE SIMULATION BY REPORTING TIME
--
--       JAN   FEB   MAR   APR   MAY   JUN   JLY   AUG   SEP   OCT   NOV   DEC
TSTEP
         31    28    31    30    31    30    31    31    30    31    30    31
/
```

The second example is similar to the previous example but with quarterly reporting time steps used instead based on $\frac{365.25}{4} = 91.3125$ days per quarter

```
-- ================================================================================
--
-- SCHEDULE SECTION
--
-- ================================================================================
SCHEDULE

-- --------------------------------------------------------------------------------
-- SCHEDULE SECTION - 2022-01-01
-- --------------------------------------------------------------------------------
RPTSCHED
          'WELLS=2'    'WELSPECS'    'CPU=2'     FIP=2'                          /

--
--       ADVANCE SIMULATION BY REPORTING TIME
--
--       QUARTERLY
TSTEP
         4*91.3125
/
```

Again, if the simulated production targets are actual production data or the results are going to be used in economic evaluations then the DATES keyword may be more useful in advancing the simulation via the reporting time steps, as the exact dates will be honored.

12.3.230 TUNING - NUMERICAL TUNING CONTROL

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

TUNING defines the parameters used for controlling the commercial simulator's numerical convergence parameters for the global grid. The keyword is similar to the TUNINGDP keyword in the SCHEDULE section that is optimized for high throughput runs. The keyword is mostly ignored by OPM Flow; however, the simulator can be instructed to read the first record of the TUNING keyword if the appropriate command line parameter has been activated (see section *2.2 Running OPM Flow 2023-10 From The Command Line*).

No.	Name	Description			Default
		Field	Metric	Laboratory	
1-1	TSINIT	TSINT is a real positive value that defines the maximum length of the next time step. Note that whenever the keyword is used TSINIT is always set back to the default value of one, unless explicitly over written.			
		days	days	hours	1.0
1-2	TSMAXZ	TSMAXZ is a real positive value that defines the maximum length of the next time step following TSINIT.			
		days	days	hours	365.0
1-3	TSMINZ	TSMINZ is a real positive value that defines the minimum length of all time steps.			
		days	days	hours	0.1
1-4	TSMCHP	TSMCHP is a real positive values that sets the minimum length of all chopped time steps.			
		days	days	hours	0.15
1-5	TSFMAX	TSFMAX is a real positive value that specifies the maximum growth rate a time step can be increased by, subject to the maximum allowable time step size set by TSMAXZ. For example, if the current time step has converged at 10 days and TSFMAX is set to the default value, then the next time step will be 3.0 x 10 days, that is 30 days provided it is less than TSMAXZ.			
		dimensionless	dimensionless	dimensionless	3.0
1-6	TSFMIN	TSFMIN is a real positive value that specifies the minimum decay rate a time step can be decreased by, subject to the minimum allowable time step size set by TSMINZ. For example, if the current time step has not converged at 10 days and TSFMAX is set to the default value, then the next time step will be 0.3 x 10 days, that is the maximum of 0.3 days and TSMINZ.			
		dimensionless	dimensionless	dimensionless	0.3

No.	Name	Description			Default
		Field	Metric	Laboratory	
1-7	TSFCNV	TSFCNV real positive value that specifies the decay rate a time step can be decreased by after the number of target iterations has been exceeded.			
		dimensionless	dimensionless	dimensionless	0.1
1-8	TFDIFF	TFDIFFA is a real positive value that sets the time step growth factor of the time step after a convergence failure. For example, if the chopped current convergent time step is 10 days and TFDIFF is set to the default value, then the time step will be increased to 1.25 x 10 days, that is the minimum of 11.25 days and TSMAXZ.			
		dimensionless	dimensionless	dimensionless	1.25
1-9	THRURPT	THRURPT is a real positive value that specifies the maximum throughput ratio over a time step.			
		dimensionless	dimensionless	dimensionless	1.0×10^{20}
1-10	TMAXWC	TMAXWC is a real double precision value that defines maximum allowed time step after a well event; for example, when a well is opened or closed, etc.			
		days	days	hours	None
1-11	/	Record terminated by a "/"			Not Applicable
2-1	TRGTTE	TRGTTE is a real positive value that sets the time truncation error target.			
		dimensionless	dimensionless	dimensionless	0.1
2-2	TRGCNV	TRGCNV a real positive value that defines the non-linear convergence error.			
		dimensionless	dimensionless	dimensionless	0.001
2-3	TRGMBE	TRGMBE is a real positive value that specifies the target material balance error.			
		dimensionless	dimensionless	dimensionless	1.0×10^{-7}
2-4	TRGLCV	TRGLCV is a real positive value that specifies the linear convergence error target.			
		dimensionless	dimensionless	dimensionless	0.00001
2-5	XXXTTE	XXXTTE is a real positive value that sets the maximum time truncation error.			
		dimensionless	dimensionless	dimensionless	10.0
2-6	XXXCNV	XXXCNV is a real positive value that defines the maximum non-linear convergence error.			
		dimensionless	dimensionless	dimensionless	0.01

No.	Name	Description			Default
		Field	Metric	Laboratory	
2-7	XXXMBE	XXXMBE is a real positive value that specifies the maximum mass balance error, that is the tolerated mass balance error relative to total mass present.			
		dimensionless	dimensionless	dimensionless	1.0×10^{-6}
2-8	XXXLCV	XXXLCV is a real positive values that sets the maximum linear convergence error.			
		dimensionless	dimensionless	dimensionless	0.0001
2-9	XXXWFL	XXXWFL is a real positive values that fixes the maximum well flow convergence error.			
		dimensionless	dimensionless	dimensionless	0.001
2-10	TRGFIP	TRGFIP is a real positive value that stipulates the target fluid in-place error in Local Grid Refinements.			
		dimensionless	dimensionless	dimensionless	0.025
2-11	TRGSFT	TRGSFT is a real positive values that defines the target surfactant change when the Surfactant Model is active in the run.			
		dimensionless	dimensionless	dimensionless	None
2-12	THIONX	THIONX is a positive real value used to set the threshold for damping in the ion echange calculation for when the Brine Model is active in the run.			
		dimensionless	dimensionless	dimensionless	0.01
2-13	TRWGHT	TRWGHT is a positive integer that stipulates the implicitness for active tracer updates within the Newton iterations, and should be set to: 1 : The calculation is explicit, that is fully decoupled. 2 : The calculation is implicit, that is fully coupled.			
		dimensionless	dimensionless	dimensionless	1
2-14	/	Record terminated by a "/"			Not Applicable
3-1	NEWTMX	NEWTMX is a positive integer greater or equal to NEWTMN that stipulates the maximum number of Newton iterations for a time step.			
		dimensionless	dimensionless	dimensionless	12
3-2	NEWTMN	NEWTMN is a positive integer that is less or equal to NEWTMX that defines the minimum number of Newton iterations for a time step.			
		dimensionless	dimensionless	dimensionless	1
3-3	LITMAX	LITMAX is a positive integer greater or equal to LITMIN that sets the maximum number of linear iterations within a Newton iteration.			
		dimensionless	dimensionless	dimensionless	25

No.	Name	Description			Default
		Field	Metric	Laboratory	
3-4	LITMIN	LITMIN is a positive integer less or equal to LITMAX that sets the minimum number of linear iterations within a Newton iteration.			
		dimensionless	dimensionless	dimensionless	1
3-5	MXWSIT	MXWSIT is a positive integer that defines the maximum number of iterations within a well flow calculation.			
		dimensionless	dimensionless	dimensionless	8
3-6	MXWPIT	MXWPIT is a positive integer that stipulates the maximum number of iterations for solving the bottom-hole pressure for wells under tubing head pressure control within a well flow calculation.			
		dimensionless	dimensionless	dimensionless	8
3-7	DDPLIM	DDPLIM a real positive value that stipulates the maximum pressure change at the last Newton iteration.			
		psia	barsa	atma	1.0×10^{-6}
3-8	DDSLIM	DDSLIM a real positive value that sets the maximum saturation change at the last Newton iteration.			
		dimensionless	dimensionless	dimensionless	1.0×10^{-6}
3-9	TRGDPR	TRGDP is a real positive value that defines the target pressure change within a time step.			
		psia	barsa	atma	1.0×10^{-6}
3-10	XXXDPR	XXXDPR is a real positive value that stipulates the maximum tolerable pressure change within a time step.			
		psia	barsa	atma	1.0×10^{-6}
3-11	MNWRFP	MNWRFP is a positive integer greater than one and less than NEWTMX that defines the minimum number of Newton iterations before invoking the bisection algorithm for when the polymer phase is active in the model via the POLYMER keyword in the RUNSPEC section.			
		dimensionless	dimensionless	dimensionless	4
3-12	/	Record terminated by a "/"			Not Applicable

Notes:

1) The keyword consists of three records, with items 1-1 to 1-11 representing record one items, 2-1 to 2-14 record two and 3-1 to 3-12 representing record number three. Each of the records are terminated by a "/" and is explicitly shown in the above rows.
2) There is no keyword terminating "/".

Table 12.68: TUNING Keyword Description

Note that for record number two (items 2-1 to 2-13) the maximum values should always be greater than the associated target value; for example, XXXCNV should be greater than TRGCNV. Also note that the TUNING keyword is stored on the restart files (see *RPTRST – Define Data to be Written to the RESTART File*) enabling the parameters to be utilized in a restart run without re-specifying the keyword.

Example

```
--
--       DEFAULT TUNING PARAMETERS
--
TUNING
         1.0    365.0     0.1     0.15     3     0.3     0.1   1.25  1E20  1*   /
                                                                                 /

    /
```

The above example explicitly sets the default parameters for OPM Flow for when the appropriate command line parameter has been activated (see section *2.2 Running OPM Flow 2023-10 From The Command Line*) to instruct the simulator to read the first record of the TUNING keyword. Alternatively one could just use:

```
TUNING
/
/
/
```

To accomplish the same thing.

12.3.231 TUNINGDP – NUMERICAL TUNING CONTROL FOR HIGH THROUGHPUT CASES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

Defines the parameters used for controlling the commercial simulator's numerical convergence parameters. The keyword is similar to the TUNING keyword in the SCHEDULE section, but the defaults on this keyword are optimized for high throughput runs. See section *2.2 Running OPM Flow 2023-10 From The Command Line* on how to invoke various numerical schemes via the OPM Flow command line interface.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	TRGLCV	TRGLCV is a real positive value that specifies the linear convergence error target. The default value is ten times greater than the default value on the TUNING keyword.			
		dimensionless	dimensionless	dimensionless	0.0001
2	XXXLCV	XXXLCV is a real positive values that sets the maximum linear convergence error. The default value is ten times greater than the default value on the TUNING keyword.			
		dimensionless	dimensionless	dimensionless	0.001
3	TRGDDP	TRGDDP a real positive value that stipulates the maximum pressure change during a Newton iteration that enables the solution to be accepted when the residual pressure is still outside its convergence criteria.			
		psia	barsa	atma	1.0
4	TRGDDS	TRGDDS a real positive value that sets the maximum saturation change during a Newton iteration that enables the solution to be accepted when the residual saturation is still outside its convergence criteria.			
		dimensionless	dimensionless	dimensionless	0.001

Notes:

1) There is no keyword terminating "/".

Table 12.69: TUNINGDP Keyword Description

Note that the TUNING keyword is stored on the restart files (see *RPTRST – Define Data to be Written to the RESTART File*) enabling the parameters to be utilized in a restart run without re-specifying the keyword.

Example

```
--
--       DEFAULT TUNINGDP PARAMETERS
--
TUNINGDP
/
```

The above example explicitly sets the default parameters.

12.3.232 TUNINGH – NUMERICAL TUNING CONTROL FOR HISTORY MATCH GRADIENT CALCULATIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

Defines the parameters used for controlling the commercial simulator's numerical convergence parameters. The keyword is similar to the TUNING keyword in the SCHEDULE section, but the defaults on this keyword are optimized for high throughput runs. See section *2.2 Running OPM Flow 2023-10 From The Command Line* on how to invoke various numerical schemes via the OPM Flow command line interface.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	GRGLCV	GRGLCV is a real positive value that specifies the linear convergence error target.			
		dimensionless	dimensionless	dimensionless	0.0001
2	GXXLCV	GXXLCV is a real positive values that sets the maximum linear convergence error.			
		dimensionless	dimensionless	dimensionless	0.001
3	GMSLCV	GMSLCV is a real positive value that specifies the linear convergence residual reduction.			
		dimensionless	dimensionless	dimensionless	1.0×10^{-20}
4	LGTMIN	LGTMIN is a positive integer less or equal to LGTMAX that sets the minimum number of linear iterations within a Newton iteration.			
		dimensionless	dimensionless	dimensionless	1
5	LGTMAX	LGTMAX is a positive integer greater or equal to LGTMIN that sets the maximum number of linear iterations within a Newton iteration.			
		dimensionless	dimensionless	dimensionless	25

Notes:

1) There is no keyword terminating "/".

Table 12.70: TUNINGH Keyword Description

Example

```
--
--       DEFAULT TUNINGH PARAMETERS
--
TUNINGH
/
```

The above example explicitly sets the default parameters.

12.3.233 TUNINGL - NUMERICAL TUNING CONTROL FOR ALL LGRS

Description

TUNINGL defines the parameters used for controlling the commercial simulator's numerical convergence parameters for all Local Grid Refinements (LGR"). The keyword is the same as the TUNING keyword in the SCHEDULE section that applies the tuning parameters to the global grid. See section *2.2 Running OPM Flow 2023-10 From The Command Line* on how to invoke various numerical schemes via the OPM Flow command line interface.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1-1	TSINIT	TSINT is a real positive value that defines the maximum length of the next time step. Note that whenever the keyword is used TSINIT is always set back to the default value of one, unless explicitly over written.			
		days	days	hours	1.0
1-2	TSMAXZ	TSMAXZ is a real positive value that defines the maximum length of the next time step following TSINIT.			
		days	days	hours	365.0
1-3	TSMINZ	TSMINZ is a real positive value that defines the minimum length of all time steps.			
		days	days	hours	0.1
1-4	TSMCHP	TSMCHP is a real positive values that sets the minimum length of all chopped time steps.			
		days	days	hours	0.15
1-5	TSFMAX	TSFMAX is a real positive value that specifies the maximum growth rate a time step can be increased by, subject to the maximum allowable time step size set by TSMAXZ. For example, if the current time step has converged at 10 days and TSFMAX is set to the default value, then the next time step will be 3.0 x 10 days, that is 30 days provided it is less than TSMAXZ.			
		dimensionless	dimensionless	dimensionless	3.0
1-6	TSFMIN	TSFMIN is a real positive value that specifies the minimum decay rate a time step can be decreased by, subject to the minimum allowable time step size set by TSMINZ. For example, if the current time step has not converged at 10 days and TSFMAX is set to the default value, then the next time step will be 0.3 x 10 days, that is the maximum of 0.3 days and TSMINZ.			
		dimensionless	dimensionless	dimensionless	0.3

No.	Name	Description			Default
		Field	Metric	Laboratory	
1-7	TSFCNV	TSFCNV real positive value that specifies the decay rate a time step can be decreased by after the number of target iterations has been exceeded.			
		dimensionless	dimensionless	dimensionless	0.1
1-8	TFDIFF	TFDIFFA is a real positive value that sets the time step growth factor of the time step after a convergence failure. For example, if the chopped current convergent time step is 10 days and TFDIFF is set to the default value, then the time step will be increased to 1.25 x 10 days, that is the minimum of 11.25 days and TSMAXZ.			
		dimensionless	dimensionless	dimensionless	1.25
1-9	THRURPT	THRURPT is a real positive value that specifies the maximum throughput ratio over a time step.			
		dimensionless	dimensionless	dimensionless	1.0×10^{20}
1-10	TMAXWC	TMAXWC is a real double precision value that defines maximum allowed time step after a well event; for example, when a well is opened or closed, etc.			
		days	days	hours	None
1-11	/	Record terminated by a "/"			Not Applicable
2-1	TRGTTE	TRGTTE is a real positive value that sets the time truncation error target.			
		dimensionless	dimensionless	dimensionless	0.1
2-2	TRGCNV	TRGCNV a real positive value that defines the non-linear convergence error.			
		dimensionless	dimensionless	dimensionless	0.001
2-3	TRGMBE	TRGMBE is a real positive value that specifies then target material balance error.			
		dimensionless	dimensionless	dimensionless	1.0×10^{-7}
2-4	TRGLCV	TRGLCV is a real positive value that specifies the linear convergence error target.			
		dimensionless	dimensionless	dimensionless	0.00001
2-5	XXXTTE	XXXTTE is a real positive value that sets the maximum time truncation error.			
		dimensionless	dimensionless	dimensionless	10.0
2-6	XXXCNV	XXXCNV is a real positive value that defines the maximum non-linear convergence error.			
		dimensionless	dimensionless	dimensionless	0.01

No.	Name	Description			Default
		Field	Metric	Laboratory	
2-7	XXXMBE	XXXMBE is a real positive value that specifies the maximum mass balance error, that is the tolerated mass balance error relative to total mass present.			
		dimensionless	dimensionless	dimensionless	1.0×10^{-6}
2-8	XXXLCV	XXXLCV is a real positive values that sets the maximum linear convergence error.			
		dimensionless	dimensionless	dimensionless	0.0001
2-9	XXXWFL	XXXWFL is a real positive values that fixes the maximum well flow convergence error.			
		dimensionless	dimensionless	dimensionless	0.001
2-10	TRGFIP	TRGFIP is a real positive value that stipulates the target fluid in-place error in Local Grid Refinements.			
		dimensionless	dimensionless	dimensionless	0.025
2-11	TRGSFT	TRGSFT is a real positive values that defines the target surfactant change when the Surfactant Model is active in the run.			
		dimensionless	dimensionless	dimensionless	None
2-12	THIONX	THIONX is a positive real value used to set the threshold for damping in the ion echange calculation for when the Brine Model is active in the run.			
		dimensionless	dimensionless	dimensionless	0.01
2-13	TRWGHT	TRWGHT is a positive integer that stipulates the implicitness for active tracer updates within the Newton iterations, and should be set to: 1 : The calculation is explicit, that is fully decoupled. 2 : The calculation is implicit, that is fully coupled.			
		dimensionless	dimensionless	dimensionless	1
2-14	/	Record terminated by a "/"			Not Applicable
3-1	NEWTMX	NEWTMX is a positive integer greater or equal to NEWTMN that stipulates the maximum number of Newton iterations for a time step.			
		dimensionless	dimensionless	dimensionless	12
3-2	NEWTMN	NEWTMN is a positive integer that is less or equal to NEWTMX that defines the minimum number of Newton iterations for a time step.			
		dimensionless	dimensionless	dimensionless	1
3-3	LITMAX	LITMAX is a positive integer greater or equal to LITMIN that sets the maximum number of linear iterations within a Newton iteration.			
		dimensionless	dimensionless	dimensionless	25

No.	Name	Description			Default
		Field	Metric	Laboratory	
3-4	LITMIN	LITMIN is a positive integer less or equal to LITMAX that sets the minimum number of linear iterations within a Newton iteration.			
		dimensionless	dimensionless	dimensionless	1
3-5	MXWSIT	MXWSIT is a positive integer that defines the maximum number of iterations within a well flow calculation.			
		dimensionless	dimensionless	dimensionless	8
3-6	MXWPIT	MXWPIT is a positive integer that stipulates the maximum number of iterations for solving the bottom-hole pressure for wells under tubing head pressure control within a well flow calculation.			
		dimensionless	dimensionless	dimensionless	8
3-7	DDPLIM	DDPLIM a real positive value that stipulates the maximum pressure change at the last Newton iteration.			
		psia	barsa	atma	1.0×10^{-6}
3-8	DDSLIM	DDSLIM a real positive value that sets the maximum saturation change at the last Newton iteration.			
		dimensionless	dimensionless	dimensionless	1.0×10^{-6}
3-9	TRGDPR	TRGDP is a real positive value that defines the target pressure change within a time step.			
		psia	barsa	atma	1.0×10^{-6}
3-10	XXXDPR	XXXDPR is a real positive value that stipulates the maximum tolerable pressure change within a time step.			
		psia	barsa	atma	1.0×10^{-6}
3-11	MNWRFP	MNWRFP is a positive integer greater than one and less than NEWTMX that defines the minimum number of Newton iterations before invoking the bisection algorithm for when the polymer phase is active in the model via the POLYMER keyword in the RUNSPEC section.			
		dimensionless	dimensionless	dimensionless	4
3-12	/	Record terminated by a "/"			Not Applicable

Notes:

1) The keyword consists of three records, with items 1-1 to 1-11 representing record one items, 2-1 to 2-14 record two and 3-1 to 3-12 representing record number three. Each of the records are terminated by a "/" and is explicitly shown in the above rows.
2) There is no keyword terminating "/".

Table 12.71: TUNINGL Keyword Description

Note that for record number two (items 2-1 to 2-13) the maximum values should always be greater than the associated target value; for example, XXXCNV should be greater than TRGCNV. Also note that the TUNINGS keyword is stored on the restart files (see *RPTRST – Define Data to be Written to the RESTART File*) enabling the parameters to be utilized in a restart run without re-specifying the keyword.

See also the TUNINGS keyword in the SCHEDULE section that sets the tuning parameters for individual LGRs.

Example

```
-
--       DEFAULT TUNINGL PARAMETERS
--
TUNINGL
/
/
/
```

The above example explicitly sets the default parameters for all LGRs.

12.3.234 TUNINGS - NUMERICAL TUNING CONTROL FOR INDIVIDUAL LGRS

Description

TUNINGS defines the parameters used for controlling the commercial simulator's numerical convergence parameters for individual Local Grid Refinements (LGR"). The keyword is similar to the TUNINGL keyword in the SCHEDULE section that applies the tuning parameters to the all LGRs, except for an additional first record that includes the LGR name.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

See section *2.2 Running OPM Flow 2023-10 From The Command Line* on how to invoke various numerical schemes via the OPM Flow command line interface.

No.	Name	Description			Default
		Field	Metric	Laboratory	
0-1	LGRNAME	A character string of up to eight characters in length that defines the LGR name for which the tuning data is being being defined.			None
0-2	/	Record terminated by a "/"			Not Applicable
1-1	TSINIT	TSINT is a real positive value that defines the maximum length of the next time step. Note that whenever the keyword is used TSINIT is always set back to the default value of one, unless explicitly over written.			
		days	days	hours	1.0
1-2	TSMAXZ	TSMAXZ is a real positive value that defines the maximum length of the next time step following TSINIT.			
		days	days	hours	365.0
1-3	TSMINZ	TSMINZ is a real positive value that defines the minimum length of all time steps.			
		days	days	hours	0.1
1-4	TSMCHP	TSMCHP is a real positive values that sets the minimum length of all chopped time steps.			
		days	days	hours	0.15
1-5	TSFMAX	TSFMAX is a real positive value that specifies the maximum growth rate a time step can be increased by, subject to the maximum allowable time step size set by TSMAXZ. For example, if the current time step has converged at 10 days and TSFMAX is set to the default value, then the next time step will be 3.0 x 10 days, that is 30 days provided it is less than TSMAXZ.			
		dimensionless	dimensionless	dimensionless	3.0

No.	Name	Description			Default
		Field	Metric	Laboratory	
1-6	TSFMIN	TSFMIN is a real positive value that specifies the minimum decay rate a time step can be decreased by, subject to the minimum allowable time step size set by TSMINZ. For example, if the current time step has not converged at 10 days and TSFMAX is set to the default value, then the next time step will be 0.3 x 10 days, that is the maximum of 0.3 days and TSMINZ.			
		dimensionless	dimensionless	dimensionless	0.3
1-7	TSFCNV	TSFCNV real positive value that specifies the decay rate a time step can be decreased by after the number of target iterations has been exceeded.			
		dimensionless	dimensionless	dimensionless	0.1
1-8	TFDIFF	TFDIFFA is a real positive value that sets the time step growth factor of the time step after a convergence failure. For example, if the chopped current convergent time step is 10 days and TFDIFF is set to the default value, then the time step will be increased to 1.25 x 10 days, that is the minimum of 11.25 days and TSMAXZ.			
		dimensionless	dimensionless	dimensionless	1.25
1-9	THRURPT	THRURPT is a real positive value that specifies the maximum throughput ratio over a time step.			
		dimensionless	dimensionless	dimensionless	1.0×10^{20}
1-10	TMAXWC	TMAXWC is a real double precision value that defines maximum allowed time step after a well event; for example, when a well is opened or closed, etc.			
		days	days	hours	None
1-11	/	Record terminated by a "/"			Not Applicable
2-1	TRGTTE	TRGTTE is a real positive value that sets the time truncation error target.			
		dimensionless	dimensionless	dimensionless	0.1
2-2	TRGCNV	TRGCNV a real positive value that defines the non-linear convergence error.			
		dimensionless	dimensionless	dimensionless	0.001
2-3	TRGMBE	TRGMBE is a real positive value that specifies then target material balance error.			
		dimensionless	dimensionless	dimensionless	1.0×10^{-7}
2-4	TRGLCV	TRGLCV is a real positive value that specifies the linear convergence error target.			
		dimensionless	dimensionless	dimensionless	0.00001

No.	Name	Description			Default
		Field	Metric	Laboratory	
2-5	XXXTTE	XXXTTE is a real positive value that sets the maximum time truncation error.			
		dimensionless	dimensionless	dimensionless	10.0
2-6	XXXCNV	XXXCNV is a real positive value that defines the maximum non-linear convergence error.			
		dimensionless	dimensionless	dimensionless	0.01
2-7	XXXMBE	XXXMBE is a real positive value that specifies the maximum mass balance error, that is the tolerated mass balance error relative to total mass present.			
		dimensionless	dimensionless	dimensionless	1.0×10^{-6}
2-8	XXXLCV	XXXLCV is a real positive values that sets the maximum linear convergence error.			
		dimensionless	dimensionless	dimensionless	0.0001
2-9	XXXWFL	XXXWFL is a real positive values that fixes the maximum well flow convergence error.			
		dimensionless	dimensionless	dimensionless	0.001
2-10	TRGFIP	TRGFIP is a real positive value that stipulates the target fluid in-place error in Local Grid Refinements.			
		dimensionless	dimensionless	dimensionless	0.025
2-11	TRGSFT	TRGSFT is a real positive values that defines the target surfactant change when the Surfactant Model is active in the run.			
		dimensionless	dimensionless	dimensionless	None
2-12	THIONX	THIONX is a positive real value used to set the threshold for damping in the ion echange calculation for when the Brine Model is active in the run.			
		dimensionless	dimensionless	dimensionless	0.01
2-13	TRWGHT	TRWGHT is a positive integer that stipulates the implicitness for active tracer updates within the Newton iterations, and should be set to: 1 : The calculation is explicit, that is fully decoupled. 2 : The calculation is implicit, that is fully coupled.			
		dimensionless	dimensionless	dimensionless	1
2-14	/	Record terminated by a "/"			Not Applicable
3-1	NEWTMX	NEWTMX is a positive integer greater or equal to NEWTMN that stipulates the maximum number of Newton iterations for a time step.			
		dimensionless	dimensionless	dimensionless	12

No.	Name	Description			Default
		Field	Metric	Laboratory	
3-2	NEWTMN	NEWTMN is a positive integer that is less or equal to NEWTMX that defines the minimum number of Newton iterations for a time step.			
		dimensionless	dimensionless	dimensionless	1
3-3	LITMAX	LITMAX is a positive integer greater or equal to LITMIN that sets the maximum number of linear iterations within a Newton iteration.			
		dimensionless	dimensionless	dimensionless	25
3-4	LITMIN	LITMIN is a positive integer less or equal to LITMAX that sets the minimum number of linear iterations within a Newton iteration.			
		dimensionless	dimensionless	dimensionless	1
3-5	MXWSIT	MXWSIT is a positive integer that defines the maximum number of iterations within a well flow calculation.			
		dimensionless	dimensionless	dimensionless	8
3-6	MXWPIT	MXWPIT is a positive integer that stipulates the maximum number of iterations for solving the bottom-hole pressure for wells under tubing head pressure control within a well flow calculation.			
		dimensionless	dimensionless	dimensionless	8
3-7	DDPLIM	DDPLIM a real positive value that stipulates the maximum pressure change at the last Newton iteration.			
		psia	barsa	atma	1.0×10^{-6}
3-8	DDSLIM	DDSLIM a real positive value that sets the maximum saturation change at the last Newton iteration.			
		dimensionless	dimensionless	dimensionless	1.0×10^{-6}
3-9	TRGDPR	TRGDP is a real positive value that defines the target pressure change within a time step.			
		psia	barsa	atma	1.0×10^{-6}
3-10	XXXDPR	XXXDPR is a real positive value that stipulates the maximum tolerable pressure change within a time step.			
		psia	barsa	atma	1.0×10^{-6}
3-11	MNWRFP	MNWRFP is a positive integer greater than one and less than NEWTMX that defines the minimum number of Newton iterations before invoking the bisection algorithm for when the polymer phase is active in the model via the POLYMER keyword in the RUNSPEC section.			
		dimensionless	dimensionless	dimensionless	4
3-12	/	Record terminated by a "/"			Not Applicable

No.	Name	Description			Default
		Field	Metric	Laboratory	
Notes: 1) The keyword consists of three records, with items 1-1 to 1-11 representing record one items, 2-1 to 2-14 record two and 3-1 to 3-12 representing record number three. Each of the records are terminated by a "/" and is explicitly shown in the above rows. 2) There is no keyword terminating "/".					

Table 12.72: TUNINGS Keyword Description

Note that for record number two (items 2-1 to 2-13) the maximum values should always be greater than the associated target value; for example, XXXCNV should be greater than TRGCNV. Also note that the TUNING keyword is stored on the restart files (see *RPTRST – Define Data to be Written to the RESTART File*) enabling the parameters to be utilized in a restart run without re-specifying the keyword.

See also the TUNINGL keyword in the SCHEDULE section that sets the tuning parameters for all LGRs.

Example

```
--
--       DEFAULT TUNINGS PARAMETERS
--
TUNINGS
OP01-LGR
/
/
/
```

The above example explicitly sets the default parameters for the LGR named OP01-LGR

12.3.235 UDQ - DECLARE USER DEFINE QUANTITIES ("UDQ")

Description

This keyword starts the definition of a UDQ section that stipulates the variables and operations used to access the User Defined Quantities features in OPM Flow. UDQ variables can be constants, SUMMARY variables, as defined in the SUMMARY section, or a formula using various mathematical functions together with constants and SUMMARY variables. Available operation commands include the ASSIGN, DEFINE UNITS and UPDATE that are sub-keywords to the UDQ section keyword. An UDQ definition section is terminated by a "/" on a single line.

User Defined Quantities and also be used as User Defined Arguments ("UDA") in the SCHEDULE section with various group, well, and connection keywords. In this case, the UDA variables are used to replace numerical values on these keywords by UDA variables that have been defined by the UDQ keyword. For example, if we wish to make the oil rate for certain wells to be a function of their GOR, then one can define the function using the UDQ keyword that results in a UDQ variable, WU_MXOIL say, and then use WU_MXOIL as a UDA variable on the WCONPROD keyword for the ORAT parameter. See the *ACTIONX – Define Action Conditions and Command Processing* in the SCHEDULE section for the list of keywords that can use UDA variables.

Although this keyword is read by OPM Flow and the ACTION and UDQ computational logic and calculations have been implemented, one should used caution using this facility as it may result in OPM Flow aborting.

No.	Name	Description	Default
UDQ		Define the start of UDQ Definition Section. This is then followed on a new line by any number of UDQ records that define the various operations to be performed using the ASSIGN, DEFINE UNITS and UPDATE sub-keywords for the OPERATOR.	
1	OPERATOR	OPERATOR is a character sting that that defines the type of operations to perform, and should be one of the following: 1) ASSIGN: This option assigns a value to a variable and sets the UPDATE status to OFF. 2) DEFINE: In this case a mathematical formula is defined and assigned to a variable. The variable is initialized with the formula and the UPDATE status is set to ON. 3) UNITS: This option sets the reporting units for a defined variable and has no effect on the calculations. The variable must already have been defined prior to using this option. 4) UPDATE: Stipulates when the defined variable should be re-calculated.	

No.	Name	Description	Default
2	VARIABLE	VARIABLE is a character string of length eight that stipulates the name of the user defined variable that will processed by the OPERATOR command. The first two characters of VARIABLE must be set based on the type of variable being defined, that is: 1) CU: For variables that are associated with connections, for example SUMMARY variable COFR (Connection Oil Flow Rate). 2) FU: For variables that are associated with field data, for example SUMMARY variable FOPR (Field Oil Production Rate). 3) GU: For variables that are associated with groups, for example SUMMARY variable GLPR (Group Liquid Production Rate). 4) RU: For variables that are associated with regions, for example SUMMARY variable RPR (Region Pressure). 5) SU: For variables that are associated with multi-segment wells, for example SUMMARY variable SOFR (Segment Oil Flow Rate). 6) WU: For variables that are associated with wells, for example SUMMARY variable WWCT (Well Water Cut). 7) AU: For variables that are associated with aquifers, for example SUMMARY variable AAQP (Analytical Aquifer Pressure). 8) BU: For variables that are associated with blocks, for example SUMMARY variable BPR (Block oil phase Pressure).	
3	EXPRESSION	The data type for EXPRESSION is based on the OPERATOR option above, namely if OPERATOR is set to: 1) ASSIGN: Then EXPRESSION should be a numerical value. 2) DEFINE: In this case a mathematical formula is defined and assigned to a variable. The variable is initialized with the formula and the UPDATE status is set to ON. 3) UNITS: Then EXPRESSION should be a character string enclosed in quotes it it contains blanks, with a maximum length of eight characters, that declares the units for VARIABLE that will be used for reporting. 4) UPDATE: In this case EXPRESSION can have the value ON to evaluate VARIABLE at all time steps, OFF to not evaluate VARIABLE, or NEXT to evaluate VARIABLE at the next time step.	
	/	Termination of a UDQ record. Note that multiple numbers of records can be entered within a UDQ section with each record terminated by a "/".	
/		Define the end of UDQ Definition Section	
Notes: 1) The keyword is terminated by a "/".			

Table 12.73: UDQ Keyword Description

Currently, the simulator only supports field and well variable names (FU and WU) type variables and simple mathematical formula consisting of opening and closing brackets (( )), and the plus, minus, multiply and divide operators, as illustrated in the examples below.

See also the UDADIMS, UDQDIMS and UDQPARAM keywords in the RUNSPEC section to define the dimensions for the UDQ keyword and associated variables.

Note

UDQ variables utilizing group and well variables, must have their associated groups and wells previously fully defined in the commercial simulator, otherwise an error will occur. For example, if a well's GOR is being used as part of a UDQ definition, then the well must be fully characterized prior to declaring the UDQ definition.

This restriction does not apply to OPM Flow; however, it should be considered if the same deck is to be run with both simulators.

User Defined Quantities and also be used as User Defined Arguments ("UDA") in the SCHEDULE section with various group, well, and connection keywords. In this case, the UDA variables are used to replace numerical values on these keywords by UDA variables that have been defined by the UDQ keyword. For example, if we wish to make the oil rate for certain wells to be a function of their water cut, then one can define the function using the UDQ keyword that results in a UDQ variable, WU_WCUT say, and then use WU_WCUT as a UDA variable on the WCONPROD keyword for the ORAT parameter. Table 12.74 list the keywords that can be used with UDA variables.

ACTIONX - User Defined Argument Keywords Schedule Section Keywords Status					
Number	Group Keywords	Well Keywords		Connection Keywords	Miscellaneous Keywords
1	GCONINJE	WALKALIN	WINJEDET	CECON	LINCOM
2	GCONPRI	WAPI	WINJTEMP	CPIFACT	
3	GCONPROD	WCONINJE	WPOLYMER	CPIFACTL	
4	GCONSALE	WCONPROD	WSALT		
5	GCONSUMP	WECON	WSOLVENT		
6	GECON	WECONCMF	WSURFACT		
7	GRUPFUEL	WELDRAW	WTADD		
8	GRUPSALE	WELLSTRE	WTMULT		
9	GTADD	WELTARG	WTRACER		
10	GTMULT	WFOAM			
		Multi-Segment Well Keywords			
1		WSEGTABL			

Notes:

1) Cells not colored show that the keyword has been tested and is functional within an ACTIONX block using a User Defined Argument.
2) Cells colored in gray indicate that the keyword has not been tested in OPM Flow.
3) Cells colored orange show keywords currently unavailable in OPM Flow because the underlying feature is not available; either because the keyword is a compositional keyword in the commercial simulator, or the keyword has not been implemented in OPM Flow.

Table 12.74:ACTIONX User Defined Argument Supported Keywords

Note

Note that after the terminating "/" for the ASSIGN operator normally any comments can be entered; however, if there is "/" within the comment field, as per:

```
ASSIGN FUNGLYLD 1.196   /    Condensate Yield (63.5/56.7)/(1.0 – 0.065)
```

then the simulator will abort. The work around is to manually place the comment characters "--" after the ASSIGN terminating "/", like so:

```
ASSIGN FUNGLYLD 1.196   / -- Condensate Yield (63.5/56.7)/(1.0 – 0.06)
```

Examples

The fist example shows how to define some constant field variables used for calculating facilities corrected condensate and Liquefied Petroleum Gas[319] ("LPG ") yields in a wet gas model:

```
--
-- DEFINE START OF USER DEFINED QUANTITY SECTION
--
UDQ
--
-- OPERATOR VARIABLE  EXPRESSION
--
ASSIGN      FUNGLYLD  1.100000         / -- Condensate  Yield (stb/Mscf)
ASSIGN      FUNGLSHK  0.000000         /    Condensate  Shrinkage Factor to Zero
ASSIGN      FULPGYLD  0.065775         / -- LPG Sep Gas Yield (stb/Mscf)
ASSIGN      FULPGSHK  0.080410         /    LPG         Shrinkage Factor
ASSIGN      FUFACSHK  0.000935         /    Facilities  Shrinkage Factor
ASSIGN      FUFULSHK  0.052924         /    Fuel        Utilization
ASSIGN      FUDELTA   1E-10            /    Value to avoid diving by zero errors

/  DEFINE END OF USER DEFINED QUANTITY SECTION
```

[319] *Liquefied Petroleum Gas or LPG consists mainly of propane, propylene, butane, and butylene in various mixtures. It is produced as a by-product of natural gas processing and petroleum refining. The components of LPG are gases at standard conditions.*

The next example is a continuation of this example by showing how one can calculate the adjusted field condensate and LPG rates. Note both examples could be merged into a single UDQ definition but have been stated separately for ease of reference.

```
--
-- DEFINE START OF USER DEFINED QUANTITY SECTION
--
UDQ
--
-- OPERATOR VARIABLE  EXPRESSION
--
DEFINE      FU_FNGLR  FGPR *(FOGR * FUNGLYLD)  / Calculate Condensate Rate Field
UPDATE      FU_FNGLR  ON                       /
UNITS       FU_FNGLR  STBD                     /

DEFINE      FU_FLPGR  FU_FWGPR * FULPGYLD      / Calculate LPG Rate Field
UPDATE      FU_FLPGR  ON                       /
UNITS       FU_FLPGR  STBD                     /

/  DEFINE END OF USER DEFINED QUANTITY SECTION
```

In the above the DEFINE operator is use to define the equations to calculate the corrected condensate (FU_FNGLR) and LPG rates (FU_FLPGR) with the UPDATE operator set to ON so that the rates are calculate at every time step, and finally, the UNITS operator is used to set the units of the calculated rates.

The final example show the use of the UDADIMS and UDQDIMS keywords in the RUNSPEC section, followed by the keywords in the SCHEDULE section that define a UDQ definition that uses the DEFINE operator to calculate adjusted well rates based on an expression. The final set of keywords show how the UDQ defined variables are employed on the WCONPROD keyword to control the production constraints for several wells.

```
                          RUNSPEC SECTION KEYWORDS
                          ------------------------
--
--       USER DEFINED ARGUMENT DIMENSIONS
--       NO.      NOT     TOTAL
--       ARGS     USED    UDQ
UDADIMS
         10       1*      10                                                  /
--
--       USER DEFINED ARGUMENT DIMENSIONS FACILITY
--       MAX      MAX     MAX     MAX     MAX     MAX    MAX    MAX    MAX   MAX     RAND
--       FUNCS    ITEMS   CONNS   FIELD   GROUP   REGS   SEGTM  WELL   AQUF  BLCKS   OPT
UDQDIMS
         50       25      0       50      50      0      0      0      0     0       N  /
```

And the SCHEDULE section part of the example is shown on the following page.

```
                              SCHEDULE SECTION KEYWORDS
                             --------------------------
--
-- DEFINE START OF USER DEFINED QUANTITY SECTION
--
UDQ
--
-- OPERATOR VARIABLE EXPRESSION
--
DEFINE      WUOPRL (WOPR OPL01 - 150) * 0.90 / OIL & LIQ CAPACITIES
DEFINE      WULPRL (WLPR OPL01 - 200) * 0.90 /  at GEFAC = 0.8995
DEFINE      WUOPRU (WOPR OPU01 - 250) * 0.80 /
DEFINE      WULPRU (WLPR OPU01 - 300) * 0.80 /
--
UNITS       WUOPRL SM3/DAY                   / DEFINE REPORTING UNITS
UNITS       WULPRL SM3/DAY                   / FOR UDQ VARIABLES
UNITS       WUOPRU SM3/DAY                   /
UNITS       WULPRU SM3/DAY                   /
/  DEFINE END OF USER DEFINED QUANTITY SECTION
--
--       WELL PRODUCTION WELL CONTROLS
--
-- WELL  OPEN/  CNTL   OIL    WAT    GAS    LIQ    RES    BHP   THP   VFP    VFP
-- NAME  SHUT   MODE   RATE   RATE   RATE   RATE   RATE   PRES  PRES  TABLE  ALFQ
WCONPROD
OP01     SHUT   GRUP   1*     1*     1*     1*     1*     200.0                /
OP02     SHUT   GRUP   1*     1*     1*     1*     1*     200.0                /
/
DATES
         1  FEB   2020  /
--
--       WELL PRODUCTION WELL CONTROLS
--
-- WELL  OPEN/  CNTL   OIL    WAT    GAS    LIQ    RES    BHP   THP   VFP    VFP
-- NAME  SHUT   MODE   RATE   RATE   RATE   RATE   RATE   PRES  PRES  TABLE  ALFQ
WCONPROD
OP01     OPEN   GRUP   WUOPRL 1*     1*     WULPRL 1*     60.0                 /
OP02     OPEN   GRUP   WUOPRL 1*     1*     WULPRL 1*     00.0                 /
/
DATES
         1  MAR   2020  /
         1  APR   2020  /
         1  MAY   2020  /
         1  JUN   2020  /
         1  JLY   2020  /
         1  AUG   2020  /
         1  SEP   2020  /
/
```

12.3.236 UDT - DECLARE USER DEFINE TABLES ("UDT")

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword starts the definition of a UDT table that defines a multi-dimensional table that can be used to assign User Defined Quantities ("UDQ") via the UDQ keyword in the SCHEDULE section.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

12.3.237 USECUPL – LOAD A RESERVOIR COUPLING FILE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The USECUPL keyword causes the simulator to read a Reservoir Coupling file that has been previously created in a master run using the DUMPCUPL keyword in the SCHEDULE section, for when reservoir coupling is invoked by the GRUPMAST and SLAVES keywords in the SCHEDULE section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.238 VAPPARS – OIL VAPORIZATION PARAMETERS

VAPPARS defines the rate of oil vaporization in the presence of undersaturated gas and the rate at which the remaining oil gets "heavier" via the reduction in the solution gas-oil ratio ("Rs"). This keyword should only be used if the OIL, GAS, DISGAS and VAPOIL keywords in the RUNSPEC section have been invoked to allow oil, gas, dissolved gas and vaporized oil to be present in the model.

In OPM Flow, VAPPARS can only be set once, and not changed multiple times in the SCHEDULE section.

See *VAPPARS – Oil Vaporization Parameters* in the SOLUTION section for a full description.

12.3.239 VFPCHK – DEFINE PRODUCTION VERTICAL FLOW PERFORMANCE BHP CHECK

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The VFPPROD keyword defines production Vertical Flow Performance ("VFP") tables that are used to determine the outflow or downstream pressure based on the inlet or upstream pressure and the phases flowing through the system. For a well this means the table relates the flowing bottom-hole pressure ("BHP") to the well's tubing head pressure ("THP") based on the oil, gas and water rates (and any artificial lift quantities like gas lift gas), or phases ratios, flowing up the wellbore.

The VFP tables are generated by external programs and may result in some inconsistencies. A common inconsistency is that the curves of BHP versus flow rate at adjacent THP values cross, that is the BHP is increasing with decreasing THP. This will result in numerical convergence problems and should therefore be avoided; which is why the simulator checks for this particular inconsistency. However, there are cases when the external software generating the VFP table outputs "valid" high BHP values, for example, when the flow exceeds the erosion velocity limits, or the flow is supersonic, indicating a no flow condition. The VPFCHK keyword sets the BHP check pressure (VFPCHK) for subsequent VFPPROD tables, so that crossing BHP values above VFPCHK will be ignored.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	VFPCHK	VFPCHK is a real positive value that defines the BHP pressure above which crossing VFP curves will be ignored. Setting VFPCHK to a large number like the default value number will cause all crossing curves to be checked. Also if the keyword is omitted from the input deck then the check is performed using the default value.			
		psia	barsa	atma	1.0^{10}
Notes: 1) The keyword is terminated by a "/".					

Table 12.75:VFPCHK Keyword Description

Note

One reason for external programs generating crossing VFP curves is that the curves have been generated with too much resolution. For example, if the GOR entries has been generated with values of 100, 150, 200, 250, 300, 350, 400, 450 and 500, then use a geometric spacing instead to generated the VFP table, that is: 100, 300, 900. This will enable the simulator to interpolate the curves consistently and avoid crossing VFP curves.

Example

Here the example sets the maximum BHP to be 1.0 x10^6 above which crossing VFP curves will be ignored.

```
--
--        DEFINE PRODUCTION VFP CHECK MAX BHP
--
--        MXBHP
VFPCHK
          1.0E6
/
```

12.3.240 VFPINJ – DEFINE INJECTION VERTICAL FLOW PERFORMANCE TABLES

Description

The VFPINJ keyword defines injection Vertical Flow Performance ("VFP") tables that are used to determine the outflow or downstream pressure based on the inlet or upstream pressure and the phases being injected into the system. For a well this means the table relates the flowing bottom-hole pressure ("BHP") to the well's tubing head pressure ("THP") based on the oil, gas or water injection rates. The table is also used to describe the pressure relationship when the network option is being used, although the Network option is not currently implemented in OPM Flow. In this case the table describes the pipeline pressure behavior from the HIGHER group (inlet node) to the LOWER group (outlet node) given the current flowing conditions (the group relationship is defined by the GRUPTREE keyword in SCHEDULE section).

The data for this keyword is generated by an external program and is normally included into the input deck using the INCLUDE keyword as described in section *CHAPTER 4: GLOBAL SECTION KEYWORDS*, as the data can be quite voluminous.

Each VFPINJ table must be entered with a separate VFPINJ keyword that consists of four records, with 1-1, 1-2 representing record one items (1) and (2) and 2-1, 2-2 representing record number two items (1) and (2) and so on in the "No." column in Table 12.76. Each record is terminated by a "/". The fourth record must be repeated to give BHP data as a function of FLO for all THP values.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1-1	VFPTAB	A positive integer greater than zero and less than or equal to the MXVFPTAB variable as defined on the VFPIDIMS keyword in the RUNSPEC section, that defines the vertical flow performance table number.			None
1-2	VFPREF	A real positive value that defines the reference depth used to generate this VFPINJ table data set. OPM Flow automatically corrects any difference between VFPREF and the BHPREF on the WELSPECS and WPAVDEP keywords in the SCHEDULE section, using the current hydrostatic head.			None
1-3	FLO	A defined character string that defines the injection phases, and should be set to one of the following character strings: 1) OIL: for injecting phase being oil. 2) GAS: for injecting phase being gas. 3) WAT: for injecting phase being water.			None
1-4	VFPTYPE	A defined character string that should be defaulted or set equal to THP.			THP
1-5	VFPUNITS	Units used for the BHP-DATA on this keyword. This variable is ignored by OPM Flow and should be defaulted with 1*.			
		FIELD	METRIC	LAB	1*
1-6	VFPVALUE	A defined character string that should be defaulted or set equal to BHP. This variable is ignored by OPM Flow and should be defaulted with 1*.			BHP

No.	Name	Description			Default
		Field	Metric	Laboratory	
1-7	/	Record terminated by a "/"			Not Applicable
2-1	FLO-DATA	A real positive monotonically increasing vector that defines the numerical values of the injection phase declared by the FLO variable. The number of entries must greater than two and less than or equal to MXMFLO as defined on the VFPIDIMS keyword in the RUNSPEC section.			
		Liquid: stb Gas: Mscf	Liquid: sm^3 Gas: sm^3	Liquid: scc Gas: scc	None
2-2	/	Record terminated by a "/"			Not Applicable
3-1	THP-DATA	A real positive monotonically increasing vector that defines the numerical values of the tubing head pressure values. The number of entries must be greater than two and less than or equal to MXMTHP as defined on the VFPIDIMS keyword in the RUNSPEC section.			
		psia	barsa	atma	None
3-2	/	Record terminated by a "/"			Not Applicable
4-1	NTHP	This data record consists of an integer value that defines the index of THP values entered via the THP-DATA records on this keyword. For example, if THP-DATA is equal to 1000, 2000, 3000 and 3500 and NTHP is equal to three then NTHP refers to third entry, that is THP equal to 3000.			
	BHP-DATA	NTHP is then followed by a real vector of BHP values for each FLO injection rate for the corresponding index value (NTHP) and is then terminated with a"/" The (4-1) record, which consists of both NTHP and BHP-DATA data, is then repeated, until all combinations of (NTHP and FLO) and the associate BHP data has been entered.			
		psia	barsa	atma	None
4-2	/	Each Index (NTHP, BHP-DATA) data set is terminated by a "/"			Not Applicable

Notes:

1) Each VFPINJ table must be entered with a separate VFINJ keyword that consists of four records, with items 1-1 to 1-7 representing record one items and 2-1 to 2-2 representing record number two items, etc., in the "No." column in this table.
2) Each of the records are terminated by a "/" and is explicitly shown in the above rows.
3) There is no keyword terminating "/".

Table 12.76:VFPINJ Keyword Description

See also the WELSPECS keyword to define wells and the WCONINJE keyword that is used to allocate the VFPINJ tables to specific wells.

Note that one VFPINJ table can be allocated to one or more wells, provided the wells in question have a similar trajectory and similar flow characteristics, for example vertical water injection wells injecting into the same reservoir.

The VFPPROD keyword is used to enter VFP tables for production wells or to describe the pressure relationship when the network option is being used. In this case the table describes the pipeline pressure behavior from the LOWER group (inlet node) to the HIGHER group (outlet node) given the current production conditions.

All the aforementioned keywords are described in the SCHEDULE section.

Examples

The following example shows the VFPINJ table for a water injection well and is taken from the Norne OPM Flow model.

```
VFPINJ

-- Table   Datum Depth   Rate Type
-- -----   -----------   ---------
      12       2718.07       'WAT' /


-- 'WAT' units - SM3/DAY
  500.0  1263.2  2026.3  2789.5  3552.6
 4315.8  5078.9  5842.1  6605.3  7368.4
 8131.6  8894.7  9657.9 10421.1 11184.2
11947.4 12710.5 13473.7 14236.8 15000.0 /

-- 'THP' units - BARSA
  21.01   63.24  105.46  147.68  189.90
 232.12  274.35  316.57  358.79  401.01 /

1  254.51  253.95  252.27  249.83  246.69
   242.88  238.42  233.32  227.59  221.22
   214.23  206.62  198.38  189.53  180.06
   169.97  159.26  147.95  136.00  123.46
/
2  297.02  296.49  294.82  292.39  289.26
   285.47  281.01  275.92  270.20  263.84
   256.87  249.28  241.05  232.22  222.76
   212.70  202.01  190.71  178.79  166.27
/
..... . ..... . .....
..... . ..... . .....

9  594.67  594.29  592.70  590.34  587.29
   583.57  579.16  574.17  568.55  562.25
   555.40  547.92  539.79  531.09  521.74
   511.82  501.25  490.13  478.34  466.01
/
10 637.19  636.83  635.26  632.91  629.86
   626.16  621.76  616.78  611.17  604.89
   598.05  590.59  582.47  573.79  564.45
   554.56  544.01  532.91  521.14  508.83
/
```

The example shows the first two and the last two records of the fourth kind, as the data is too voluminous to be included.

Note

The VFPTAB variable defines the table number of the VFPINJ data set; if more then one VFPINJ keyword is entered with the same VFPTAB number then the VFPINJ data set will be overwritten by the last VFPINJ keyword with the same VFPTAB number.

The same comment is also applicable to the VFPPROD keyword.

12.3.241 VFPPROD – DEFINE PRODUCTION VERTICAL FLOW PERFORMANCE TABLES

Description

The VFPPROD keyword defines production Vertical Flow Performance ("VFP") tables that are used to determine the outflow or downstream pressure based on the inlet or upstream pressure and the phases flowing through the system. For a well this means the table relates the flowing bottom-hole pressure ("BHP") to the well's tubing head pressure ("THP") based on the oil, gas and water rates (and any artificial lift quantities like gas lift gas), or phases ratios, flowing up the wellbore. The table is also used to describe the pressure relationship when the network option is being used. In this case the table describes the pipeline pressure behavior from the LOWER group (inlet node) to the HIGHER group (outlet node) given the current flowing conditions (the group relationship is defined by the GRUPTREE keyword in SCHEDULE section).

Each VFPPROD table must be entered with a separate VFPPROD keyword that consists of seven records, with 1-1, 1-2 representing record one items (1) and (2) and 2-1, 2-2 representing record number two items (1) and (2) in the "No." column in Table 12.77. Each record is terminated by a "/". The seventh record must be repeated to give BHP data as a function of FLO for all THP values.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1-1	VFPTAB	A positive integer greater than zero and less than or equal to the MXVFPTAB variable as defined on the VFPPDIMS keyword in the RUNSPEC section, that defines the vertical lift performance table number.			None
1-2	VFPREF	A real positive value that defines the reference depth used to generate this VFPPROD table data set. OPM Flow automatically corrects any difference between VFPREF and the BHPREF on the WELSPECS and WPAVEDEP keywords in the SCHEDULE section, using the current hydrostatic head.			None
1-3	FLO	A defined character string that defines the flowing phases, and should be set to one of the following character strings: 1) GAS: for flowing phase being the gas rate. 2) OIL: for flowing phase being the oil rate. 3) LIQ: for flowing phase being the liquid (oil plus water) rate.			None

No.	Name	Description			Default
		Field	Metric	Laboratory	
1-4	WFR	A defined character string that defines the flowing water fraction and should be set to one of the following character strings: 1) WOR: for the water fraction being the water-oil ratio $\frac{q_w}{q_o}$ and should be used if FLO is set to 'OIL' or 'LIQ' 2) WCT: for the water fraction being the water cut $\frac{q_w}{q_o+q_w}$ and should be used if FLO is set to 'OIL' or 'LIQ' 3) WGR: for the water fraction being the water-gas ratio $\frac{q_w}{q_g}$ and should be used if FLO is set to 'GAS'.			None
1-5	GFR	A defined character string that defines the flowing gas fraction and should be set to one of the following character strings: 1) GOR: for the gas fraction being the gas-oil ratio $\frac{q_g}{q_o}$ and should be used if FLO is set to 'OIL' or 'LIQ' 2) GLR: for the gas fraction being the gas-liquid ratio $\frac{q_g}{q_o+q_w}$ and should be used if FLO is set to 'OIL' or 'LIQ' 3) OGR: for the gas fraction being the oil-gas ratio $\frac{q_o}{q_g}$ and should be used if FLO is set to 'GAS'.			None
1-6	VFPTYPE	A defined character string that should be defaulted or set equal to THP.			THP

No.	Name	Description			Default
		Field	Metric	Laboratory	
1-7	ALQ	A defined character string that defines the artificial lift quantity and should be set to one of the following character strings: 1) GRAT: for the artificial lift quantity being the gas lift gas injection rate. 2) IGLR: for the artificial lift quantity being the gas lift gas, injection gas-liquid ratio. 3) TGLR: for the artificial lift quantity being the gas lift gas, injection total gas-liquid ratio. 4) COMP: for the artificial lift quantity being the compressor power, for a compressor. 5) PUMP: for the artificial lift quantity being the pump rating for a pump. 6) DENO: for oil surface density. 7) DENG: for gas surface density. 8) BEAN: for multi-segment wells choke parameter. 9) " ": for undefined, that is for no ALQ data. The ALQ parameter is just another variable used to interpolate the outflow pressure based on the inlet pressure, together with the phases flowing through the system. As such, ALQ can represent any parameter; however, the units should be consistent with that used to set the value. For example, if pump speed (Hz) is used to set a well's artificial lift quantity via the WCONPROD(ALQ-WELL) parameter in the SCHEDULE section, then the ALQ-DATA should represent pump speed in Hz. The default value of 1* is " " or undefined, that covers the case when the ALQ variable is not entered, except for when gas lift is employed in the model. When gas lift is active then the default value for ALQ is set to GRAT provided: 1) Gas lift optimization is active via the LIFTOPT keyword, or 2) The GLIFTLIM keyword in the SCHEDULE section is used to constrain the total amount of gas lift used within a group, or 3) Gas lift flows are included in the network pressure loss calculations as determined by the GRUPNET(OPTION2), or the NODEPROP(GASLIFT) parameters in the SCHEDULE section. In addition, if any of the above is true than ALQ must be set to GRAT.			1*
1-8	VFPUNITS	Units used for the VFP-DATA on this keyword. This variable is ignored by OPM Flow and should be defaulted with 1*.			
		FIELD	METRIC	LAB	1*
1-9	VFPVALUE	A defined character string that defines the type of data in the VFP-DATA vector. This should be set equal to BHP. if the data contains the bottom-hole pressure data, or TEMP if the data is the Tubing Head Temperature (THT) data. This variable is ignored by OPM Flow and should be defaulted with 1*.			BHP
1-10	/	Record terminated by a "/"			Not Applicable

No.	Name	Description			Default
		Field	Metric	Laboratory	
2-1	FLO-DATA	A real positive monotonically increasing vector that defines the numerical values of the flowing phase declared by the FLO variable. The number of entries must greater than two and less than or equal to MXMFLO as defined on the VFPPDIMS keyword in the RUNSPEC section.			
		Liquid: stb Gas: Mscf	Liquid: sm^3 Gas: sm^3	Liquid: scc Gas: scc	None
2-2	/	Record terminated by a "/"			Not Applicable
3-1	THP-DATA	A real positive monotonically increasing vector that defines the numerical values of the tubing head pressure values. The number of entries must greater than two and less than or equal to MXMTHP as defined on the VFPPDIMS keyword in the RUNSPEC section.			
		psia	barsa	atma	None
3-2	/	Record terminated by a "/"			Not Applicable
4-1	WFR-DATA	A real positive monotonically increasing vector that defines the numerical values of the flowing water fraction declared by the WFR variable. The number of entries must greater than two and less than or equal to MXMWFR as defined on the VFPPDIMS keyword in the RUNSPEC section.			
		WOR: dimensionless WCT: dimensionless WGR: stb/Mscf	dimensionless dimensionless dimensionless	dimensionless dimensionless dimensionless	None
4-2	/	Record terminated by a "/"			Not Applicable
5-1	GFR-DATA	A real positive monotonically increasing vector that defines the numerical values of the flowing gas fraction declared by the GFR variable. The number of entries must greater than two and less than or equal to MXMGFR as defined on the VFPPDIMS keyword in the RUNSPEC section.			
		GOR: Mscf/stb GLR: Mscf/stb OGR: stb/Mscf	dimensionless dimensionless dimensionless	dimensionless dimensionless dimensionless	None
5-2	/	Record terminated by a "/"			Not Applicable
6-1	ALQ-DATA	A real positive monotonically increasing vector that defines the numerical values of the artificial lift quantity declared by the ALQ variable. The number of entries must greater than two and less than or equal to MXMALQ as defined on the VFPPDIMS keyword in the RUNSPEC section.			None

No.	Name	Description			Default
		Field	Metric	Laboratory	
		GRAT: Mscf/d IGLR: Mscf/stb TGLR: Mscf/d DENO: lb/ft^3 DENG: lb/ft	sm^3/day dimensionless dimensionless kg/m^3 kg/m^3	scc/hour dimensionless dimensionless gm/cc gm/cc	
6-2	/	Record terminated by a "/"			Not Applicable
7-1	NTHP	This data record consists of a series of integer values that defines the index of THP, WFR, GFR, ALQ entered via the those records on this keyword. The first index, NTHP, is an integer value that defines the index of THP values entered via the THP-DATA records on this keyword. For example, if THP-DATA is equal to 100, 200, 300 and 350 and NTHP is equal to three then NTHP refers to third entry, that is THP equal to 300.			None
	NWFR	The second index, NWFR, is an integer value that defines the index of the water fraction values entered via the WFR-DATA records on this keyword. For example, if WFR-DATA is equal to 0.00, 0.25, 0.50 and 0.75 and NWFR is equal to two then NWFR refers to second entry, that is WFR equal to 0.25.			None
	NGFR	The third index, NGFR, is an integer value that defines the index of the gas fraction values entered via the GFR-DATA records on this keyword. For example, if GFR-DATA is equal to 100.0, 200.0, 500.0 and 750.0 and NGFR is equal to three then NGFR refers to third entry, that is GFR equal to 500.0.			None
	NALQ	The fourth and final index entry, NALQ, is an integer value that defines the index of artificial lift values via the ALQ-DATA records on this keyword. For example, if ALQ-DATA is equal to 50, 100, 200 and 300 and NALQ is equal to one then NALQ refers to first entry, that is ALQ equal to 50. The fourth index is then followed by the VFP-DATA, containing either the BHP or THT vector values.			None
	VFP-DATA	VFP-DATA is a real vector of either BHP or THT values for each FLO production rate for the corresponding index value (NTHP, NWFR, NGFR, NALQ) and is then terminated with a"/". The (7-1) record, which consists of the four indices and BHP (or THT) data, is then repeated until all combinations of (NTHP, NWFR, NGFR, NALQ) and the associate BHP (or THT) data has been entered. Note that the VFPVALUE parameter, determines the type of VFP-DATA, that is BHP for bottom-hole pressure, or THT for tubing head temperature.			
		psia	barsa	atma	None
7-2	/	Each Index (NTHP, NWFR, NGFR, NALQ.VFP-DATA) data set is terminated by a "/"			Not Applicable

No.	Name	Description			Default
		Field	Metric	Laboratory	
Notes: 1) Each VFPPROD table must be entered with a separate VFPPROD keyword that consists of seven records, with entries 1-1 to 1-10 representing record one items and 2-1 to 2-2 representing record number two items, etc., in the "No." column in this table. 2) Each of the records are terminated by a "/" and is explicitly shown in the above rows. 3) There is no keyword terminating "/".					

Table 12.77:VFPPROD Keyword Description

The data for this keyword is generated by an external program and is normally included into the input deck using the INCLUDE keyword as described in section *CHAPTER 4: GLOBAL SECTION KEYWORDS*, as the data can be quite voluminous.

Note that for equivalent two phase runs:

1) For example oil-water runs with only the OIL and WATER keywords in the RUNSPEC section, or runs that model dead oil[320] with a constant solution gas-oil ratio value defined by the RSCONST keyword in the PROPS section, then the FLO parameter in Table 12.77 must be set to either OIL or LIQ, WFR to either WCT or WOR, and GFR to GOR.

2) Gas-water models with only the GAS and WATER keywords in the RUNSPEC section, or models that only have dry gas[321] with a constant condensate-gas ratio defined by the RVCONST keyword in the PROPS section, then the FLO parameter in Table 12.77 must be set to GAS, WFR to WGR, and GFR to OGR.

Note

It is possible to have only the OIL and WATER keywords in the RUNSPEC section and to use gas lift for the wells, without declaring the GAS phase in the RUNSPEC section.

In this case, the FLO parameter in Table 12.77 must be set to either OIL or LIQ, WFR to either WCT or WOR, and GFR to GOR. In this case the ALQ parameter is optional, but if present must set to GRAT. If the ALQ and ALQ-DATA parameters are absent then the GFR-DATA will be used based on the flowing GOR plus the stipulated gas lift gas.

See also the WELSPECS keyword to define wells and the WCONPROD keyword that is used to allocate the VFPPROD tables to specific wells. Note that one VFPPROD table can be allocated to one or more wells, provided the wells in question have a similar trajectory and similar flow characteristics, for example vertical oil wells producing from the same reservoir, or different reservoirs with similar PVT properties.

The VFPINJ keyword is used to enter VFP tables for injection wells or to describe the pressure relationship when the network option is being used. In this case the table describes the pipeline pressure behavior from the HIGHER group (inlet node) to the LOWER group (outlet node) given the current injection conditions.

All the aforementioned keywords are described in the SCHEDULE section.

[320] *"Dead" oil is oil that it contains no dissolved gas or a relatively thick oil or residue that has lost its volatile components.*

[321] *Natural gas that occurs in the absence of condensate or liquid hydrocarbons, or gas that had condensable hydrocarbons removed, is called dry gas. It is primarily methane with some intermediates. The hydrocarbon mixture is solely gas in the reservoir and there is no liquid (condensate surface liquid) formed either in the reservoir or at surface. The term dry indicates that the gas does not contain heavier hydrocarbons to form liquids at the surface conditions. Dry gas typically has GOR's greater than 100,000 scf/stb or 18,000 Sm3/m3.*

Examples

The following example shows the VFPPROD table for a production gas well and is taken from the Norne OPM Flow model. Here WFR has been set to water-gas ratio and GFR has been set to the oil-gas ratio, and the ALQ value is defaulted.

```
VFPPROD

-- Table   Datum Depth   Rate Type   WFR Type   GFR Type
-- -----   -----------   ---------   --------   --------
       5       2623.39       'GAS'      'WGR'      'OGR' /


-- 'GAS' units - SM3/DAY
  50000.0  100000.0  200000.0  400000.0  800000.0
1200000.0 1600000.0 1999999.9 3000000.0 3999999.8
5000000.5 /

-- 'THP' units - BARSA
  10.00   20.00   40.00   80.00  120.00
 150.00  200.00  250.00 /

-- 'WGR' units - SM3/SM3
      0    1e-9    1e-6    1e-5  0.0001
  0.001    0.01     0.1 /

-- 'OGR' units - SM3/SM3
   1e-7    1e-6    1e-5  0.0001   0.001
   0.01 /

-- 'ALQ' units -
      0 /

 1  1  1  1   11.93   12.22   13.35   17.24   27.93
              39.83   52.06   64.38   95.20  125.89
             156.52
/
 1  1  2  1   11.93   12.22   13.35   17.24   27.94
              39.84   52.07   64.39   95.21  125.91
             156.55
/

..... . ..... . .....
..... . ..... . .....

 8  8  5  1  483.75  511.15  614.09 1044.78 2757.56
           5592.55 9528.36 14567.24 32005.79 56375.24
             87684
/
 8  8  6  1  487.68  516.24  624.74 1075.40 2860.16
           5803.92 9880.58 15093.76 33119.59 58297.57
             90639
/
```

The example shows the first two and the last two records of type seven, as the data is too voluminous to be included.

The next example below shows an example oil producing well VFPPROD, again taken from Norne OPM Flow model. Here WFR has been set to water cut and GFR has been set to the gas-oil ratio, and the ALQ value is defaulted.

```
VFPPROD

-- Table   Datum Depth   Rate Type   WFR Type   GFR Type   TAB Type
-- -----   -----------   ---------   --------   --------   --------
--    37       2641.02       'LIQ'      'WCT'      'GOR'           /

-- Prosper files are corrected from RKB to MSL depth. lmarr

-- Table   Datum Depth   Rate Type   WFR Type   GFR Type   TAB Type
-- -----   -----------   ---------   --------   --------   --------
      37       2617.02       'LIQ'      'WCT'      'GOR'           /

-- 'LIQ' units - SM3/DAY
  200.0   500.0  1000.0  1500.0  2000.0
 2500.0  3000.0  3500.0  4000.0  4500.0
 5000.0  5500.0  6000.0  6500.0  7000.0
 7500.0  8000.0 10000.0 14000.0 /

-- 'THP' units - BARSA
  21.01   51.01   61.01   81.01  101.01
 121.01  141.01  161.01  181.01  201.01 /

-- 'WCT' units - FRACTION
      0     0.1     0.2     0.3     0.4
    0.5     0.6     0.7     0.8       1 /

-- 'GOR' units - SM3/SM3
     90     100     150     200     500
   1000    2000 /

-- 'ALQ' units -
      0 /

 1  1  1  1  160.82  136.70  119.79  115.86  117.38
              121.16  126.08  131.56  137.48  143.74
              150.29  157.07  164.02  171.07  178.13
              185.11  192.09  220.38  280.86
/
 1  1  2  1  155.63  129.40  112.32  108.64  110.44
              114.74  120.15  126.09  132.47  139.05
              146.02  153.41  160.67  167.91  175.13
              182.34  189.55  218.81  281.02
/
..... . ..... . .....
..... . ..... . .....

10 10  6  1  439.30  437.95  437.53  437.79  438.39
              439.26  440.36  441.67  443.19  444.92
              446.85  448.99  451.32  453.85  456.58
              459.51  462.64  477.11  515.47
/
10 10  7  1  439.30  437.95  437.53  437.79  438.39
              439.26  440.36  441.67  443.19  444.92
              446.85  448.99  451.32  453.85  456.58
              459.51  462.64  477.11  515.47
/
```

The example shows the first two and the last two records of type seven, as the data is too voluminous to be included.

12.3.242 VFPTABL – DEFINE PRODUCTION VERTICAL FLOW PERFORMANCE ALQ INTERPOLATION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The VFPTABL keyword defines the interpolation method for production Vertical Flow Performance ("VFP") tables for the Artificial Lift Quantity ("ALQ"). Production VFP data is entered via the VFPPROD keyword in the SCHEDULE section. By default the simulator interpolates all the variables in the VFP tables using linear interpolation, including the ALQ quantity. However, if the ALQ values represent gas lift, then linear interpolation may not be insufficient, as the gradient change between the tabulated ALQ values may result in sudden changes. This is particularly important in gas lift optimization studies where the available gas lift gas is being allocated to a group of wells in order to maximize oil production rates. To overcome this issue the VFPTABL keyword allows the ALQ values to be interpolated using cubic spline interpolation, and results in a smother transition between the various ALQ entries.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	VFPTABL	VFPTABL is a defined positive integer that specifies the interpolation method to be used with the ALQ quantity in the VFP production tables, and should be set to one of the following: 1) Apply linear interpolation to all VFPPROD variables. 2) Apply linear interpolation to all VFPPROD variables, except for the ALQ variable, for which cubic spline interpolation should be used. If the keyword is absent from the input deck then linear interpolation will be used for all variables.			
		dimensionless	dimensionless	dimensionless	1
Notes: 1) The keyword is followed by a vector of numbers separated by a space and the keyword is terminated by a "/".					

Table 12.78:VFPTABL Keyword Description

Example

The example sets cubic spline interpolation for the ALQ quantity in the VFPPROD tables, with linear interpolation used for all the variables.

```
--
--       ALQ INTERPOLATION OPTION
--
--       OPTION
VFPTABL
         2
/
```

12.3.243 WAITBAL – WAIT ON NETWORK BALANCE BEFORE ALLOWING FURTHER ACTIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword sets the network balance option for all networks when networks are active in the model. Basically, the keyword either activates the PRORDER and GDRILPOT stipulated actions before or after the network has been balanced

The network option is normally used to ensure that the tubing head pressure ("THP") of a group of wells flowing into a common network node is consistent with a group's flow rates, that is each well's THP is flowing at the same THP and at the same time satisfying well and group targets and constraints. This is accomplished by calculating the well THP limits dynamically by balancing the flow rates and pressure losses in the network.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.244 WALKALIN – DEFINE WATER INJECTION ALKALINE CONCENTRATION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

WAKALIN keyword defines the water injection alkaline concentration for water injection wells for when the surfactant and/or polymer models have been activated by the SURFACT, SURFACTW, or the POLYMER keywords in the RUNSPEC section, combined with the ALKALINE keyword which is also in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.245 WALQCALC – DEFINE WELL VFP SURFACE ALQ PHASE DENSITY

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The WALQCALC keyword defines the well VFP surface ALQ phase density use in the VFP table lookup and interpolation to be gas surface density, oil surface density, or neither. Note that the user should ensure that generated VFP tables have been generated consistent with the setting on this keyword.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.246 WAPI – DEFINE OIL WELL INJECTION API GRAVITY

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword defines an oil injection well's API gravity for when API tracking has been made active via the API keyword in the RUNSPEC section. The American Petroleum Institute (API) classifies oils based on an API gravity (γAPI), or degrees API (oAPI), the relationship between relative density (γo) of oil and API gravity (γAPI) is given by:

$$\gamma_{API} = \frac{141.5}{\gamma_o} - 131.5 \quad \textbf{(12.33)}$$

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.247 WARN – ACTIVATE WARNING MESSAGES

Turns on warning messages to be printed to the print file (*.PRT); note that this keyword is activated by default and can subsequently be switched off by the NOWARN activation keyword. The warning messages may be turned on and off using keywords WARN and NOWARN. OPM Flow always prints error messages.

See *WARN – Activate Warning Messages* in the GLOBAL section for a full description.

12.3.248 WBHGLR – DEFINE WELL BOTTOM-HOLE GLR CONSTRAINT

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, WBHGLR, defines a well's bottom-hole Gas Liquid Ratio ("GLR") constraint, where the GLR is the is the ratio of the "free" gas rate and liquid rate at bottom-hole conditions. The reference depth for bottom-hole conditions is given by the BHPREF variable on the WELSPECS keyword in the SCHEDULE section.

Normally this type of well control is applied to pumping wells to avoid the well "pumping off", that is when the liquid column above the pump is low, resulting in an increase in gas intake and an associated loss in pump efficiency.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.249 WBOREVOL – DEFINE EFFECTIVE WELLBORE STORAGE VOLUME

Description

The WBOREVOL defines a well's effective wellbore storage volume. The primary purpose of the keyword is to enable matching of the wellbore storage effects in well tests and the corresponding pressure response observed in the test. Normally, as part of well test interpretation, the pressure, permeability, effective wellbore storage, etc., are derived from the analytical interpretation of the test. This keyword therefore allows the engineer to enter the analytical derived effective wellbore storage.

Wellbore storage, in terms of well testing, is an important variable when the well is shut-in at the surface, as the well continues to flow down-hole until the fluids obtain equilibrium. Most well tests are now conducted using specialized tools that shut-in the well down-hole, thus eliminating, or mostly eliminating, wellbore storage effects.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.250 WCALCVAL – DEFINE GAS WELL CALORIFIC VALUE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword defines a gas well's calorific value for when the Gas Calorific Value option has been activated by specifying a target calorific value for a group via the GCONCAL keyword in the SCHEDULE section. If this option is invoked then the gas calorific value must be set either by this keyword for a well by well allocation of the calorific value, or by using the Tracer Tracking option (activated by the TRACER keyword in the RUNSPEC section) combined with CALTRAC keyword in the SCHEDULE section that defines the tracer for the calorific value.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.251 WCONHIST – DEFINE WELL HISTORICAL PRODUCTION RATES AND PRESSURES

Description

The WCONHIST keyword defines production rates and pressures for wells that have been declared history matching wells by the use of this keyword. History matching wells are handled differently then ordinary wells that use the WCONPROD keyword for controlling their production targets and constraints. However, the wells still need to be defined like ordinary production wells using the WELSPECS keyword in the SCHEDULE section.

Note that although wells can be allocated to a group when they are specified by the WELSPECS keyword, history matching wells cannot operate under group control. Field and group reporting is still consistent for all wells allocated to a group, but history matching wells cannot be under group control.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WELNAME	A character string of up to eight characters in length that defines the well name for which the wells observed production rates and pressures are being defined. Note that the well name (WELNAME) must have been declared previously using the WELSPECS keyword in the SCHEDULE section, otherwise an error may occur.			None
2	STATUS	A defined character string that declares the status of the well. STATUS should be set to one of the following character strings: 1) OPEN: the well is open to flow and will attempt to produce the required production volumes. 2) STOP: the well is "stopped" at the surface and will not produce any fluids to surface; however, if there any open connections then flow may occur within the wellbore and between the open connections depending on a connection's potential with respect to all the other connections. Inter-connection flow (cross flow) can be prevented by setting the XFLOW variable on the WELSPECS keyword to NO. In this case the well's behavior will be similar to the SHUT option described below. 3) SHUT: the well is shut at the surface and downhole, this results in no flow at the surface and no cross flow downhole. Note a well's STATUS should always be set either STOP or SHUT if the well's production is to be set to zero. Just setting a well's production rate to zero means that the well is open to flow with a zero rate.			OPEN

No.	Name	Description			Default
		Field	Metric	Laboratory	
3	TARGET	A defined character string that sets the observed target production phase for the well, all the other phases are calculated unconstrained and used for reporting only. The simulator will attempt to meet the TARGET based on the phase rate stated in items (4) to (6) and (10) on this keyword. TARGET should be set to one of the following character strings: 1) ORAT: the target is set to the surface oil production rate as defined by item (4). 2) WRAT: the target is set to the surface water production rate as defined by item (5). 3) GRAT: the target is set to the surface gas production rate as defined by item (6). 4) LRAT: the target is set to the surface liquid (oil plus water) production rate and is calculated by the simulator using (4) and (5). 5) RESV: the target is set to the in situ reservoir volume rate and is calculated by the simulator using items (4), (5) and (6). 6) BHP: the target rate is set to the bottom-hole pressure as defined by item (10). Note the TARGET control mode may be reset using the WHISTCTL keyword in the SCHEDULE section, from the time the WHISTCTL is invoked, thus avoiding changing the control mode on all subsequent WCONHIST keywords.			None
4	ORAT	A real positive value that defines the observed surface oil production rate target or constraint.			
		stb/d 0.0	sm^3/day 0.0	scc/hour 0.0	Defined
5	WRAT	A real positive value that defines the observed surface water production rate target or constraint.			
		stb/d 0.0	sm^3/day 0.0	scc/hour 0.0	Defined
6	GRAT	A real positive value that defines the observed surface gas production rate target or constraint			
		Mscf/d 0.0	sm^3/day 0.0	scc/hour 0.0	Defined
7	VFPTAB	A positive integer greater than or equal to zero that defines the vertical lift performance tables to be used for calculating the tubing head pressure for the well. If a non-zero value is entered then the vertical lift performance tables must be entered via the VFPPROD keyword in the SCHEDULE section and allocated to the well via this item. The default value of zero implies no vertical lift performance table initially. If this value is then reset to be greater than zero then the table will be used to calculate the well's tubing head pressure. Subsequently, the default is to use the previously declared table number.			0

No.	Name	Description			Default
		Field	Metric	Laboratory	
8	ALQ-WELL	A real positive value that defines the artificial lift quantity to be used in conjunction with the VFPPROD assigned to the well via that keyword's VFPTAB variable. VFPTAB vertical lift performance table and the artificial lift quantity ALQ-WELL are used with the well fluid rates to calculate the well's tubing head pressures values from the bottom-hole pressure. Note that the units for ALQ-WELL is dependent on the associated variable on the VFPPROD keyword.			None
9	THP	A real positive value that defines the observed tubing head pressure. This parameter is only used for comparing the actual tubing head pressure given here with those calculated by the simulator, that is history matching wells can only be controlled by either the surface injection rate or their bottom-hole pressure.			
		psia 0.0	barsa 0.0	atma 0.0	Defined
10	BHP	A real positive value that defines the observed bottom-hole pressure.			
		psia 0.0	barsa 0.0	atma 0.0	Defined
11	WGRA	A real positive value that defines the observed wet gas rate in the commercial compositional simulator. The parameter is ignored by OPM Flow and should be defaulted or set to the default value of zero.			0.0
12	NGL	A real positive value that defines the observed Natural Gas Liquid ("NGL") rate in the commercial compositional simulator. The parameter is ignored by OPM Flow and should be defaulted or set to the default value of zero.			0.0

Notes:

1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".

Table 12.79: WCONHIST Keyword Description

See also the WHISTCTL keyword that can be used to reset the TARGET phase, the GCONPROD and GCONINJE keywords to define a group's production and injection targets and constraints, and the WCONPROD keyword to define a production well's targets and constraints. All the aforementioned keywords are described in the SCHEDULE section.

Note

One can use TARGET set to RESV in the initial history matching runs to get a "reasonable" pressure match, this ensures that the total reservoir withdrawals are correct, although the individual phase withdrawals will not match. Once a reasonable pressure match is achieved for the reservoir then one can reset TARGET to the sales phase, OIL or GAS, and continue with the matching of all the phases.

In oil reservoirs some engineers prefer to use LIQ rather than OIL as the TARGET phase, although one should consider that as the water phase has no commercial value, the measurement accuracy is significantly less than the oil sales phase.

History matching wells are converted to ordinary wells by restating a well's control mode using either the WCONPROD or WELTARG keywords in the SCHEDULE section.

Examples

The following example below shows the observed production rates for the OP01 oil producer for the first quarter of 2000.

```
-- ------------------------------------------------------------------------
-- 01 JAN 2000 START OF SCHEDULE SECTION
-- ------------------------------------------------------------------------
--
--       WELL HISTORICAL PRODUCTION CONTROLS
--
-- WELL  OPEN/  CNTL   OIL    WAT    GAS    VFP    VFP   THP   BHP
-- NAME  SHUT   MODE   RATE   RATE   RATE   TABLE  ALFQ  PRES  PRES
WCONHIST
OP01     OPEN   ORAT  15.5E3  100.0  1550   10      1*   900.0 1*     /
/
DATES
01 FEB 2000 /
/
--
--       WELL HISTORICAL PRODUCTION CONTROLS
--
-- WELL  OPEN/  CNTL   OIL    WAT    GAS    VFP    VFP   THP   BHP
-- NAME  SHUT   MODE   RATE   RATE   RATE   TABLE  ALFQ  PRES  PRES
WCONHIST
OP01     OPEN   ORAT  15.2E3  150.0  1520   1*      1*   875.0 3250.0 /
/
DATES
01 MAR 2000 /
/
--
--       WELL HISTORICAL PRODUCTION CONTROLS
--
-- WELL  OPEN/  CNTL   OIL    WAT    GAS    VFP    VFP   THP   BHP
-- NAME  SHUT   MODE   RATE   RATE   RATE   TABLE  ALFQ  PRES  PRES
WCONHIST
OP01     OPEN   ORAT  15.0E3  200.0  1500   1*      1*   850.0 1*     /
/
```

From January 1, 2000 well OP01 is open and is on oil rate control, and produces 15,500 stb/d oil, with the observed rates of 100 stb/d of water and 15.5 MMscf/d of gas. The well uses VFPPROD vertical lift table number 10 so that OPM Flow can calculate the tubing head pressures based on the fluids produced and the calculated pressures in the simulator.

The next example illustrates how to convert OP01 from a history match well to a normal production well at the start for the forecast run at August 1, 2017 using the WELTARG keyword.

```
DATES
01 AUG 2017 /
/
--
--       WELL PRODUCTION AND INJECTION TARGETS
--
--  WELL WELL   TARGET
--  NAME TARG   VALUE
WELTARG
OP01     THP    1*                                                  /
/
```

Here by defaulting the bottom-hole pressure via 1* OPM Flow automatically applies the last bottom-hole pressure from the previous time step as the "constraining phase" together with the last historical rates as

constraints. This ensures a smooth transition between history and prediction without having to resort to unreasonable changes to the model. This option is currently not implemented in OPM Flow but is expected to be incorporated in a future release._

12.3.252 WCONINJ – WELL INJECTION TARGETS AND CONSTRAINTS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The WCONINJ is a legacy keyword that is no longer used in the commercial simulator and is not supported by OPM Flow. Instead well injection targets and constraints should be defined using the WCONINJE keyword in the SCHEDULE section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.253 WCONINJE – WELL INJECTION TARGETS AND CONSTRAINTS

Description

The WCONINJE keyword defines injection targets and constraints for wells that have previously been defined by the WELSPECS keyword in the SCHEDULE section. Note that wells can be allocated to a group when they are specified by the WELSPECS keyword. Wells defined to be under group control will have their injection rates controlled by the group to which they belong, in addition to any well constraints defined for the wells using this keyword.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WELNAME	A character string of up to eight characters in length that defines the well name for which the well injection targets and constraints data are being defined. Note that the well name (WELNAME) must have been declared previously using the WELSPECS keyword in the SCHEDULE section, otherwise an error may occur.			None
2	TYPE	A defined character string that defines the type of injection well. TYPE should be set to one of the following character strings: 1) GAS: for a gas injection well. 2) OIL: for an oil injection well. 3) WAT: for a water injection well.			None
3	STATUS	A defined character string that declares the status of the well. STATUS should be set to one of the following character strings: 1) OPEN: the well is open for injection and will attempt to inject the required injection volumes. 2) STOP: the well is "stopped" at the surface and will not inject any fluids; however, if there any open connections then flow may occur within the wellbore and between the open connections depending on a connection's potential with respect to all the other connections. Inter-connection flow (cross flow) can be prevented by setting the XFLOW variable on the WELSPECS keyword to NO. In this case the well's behavior will be similar to the SHUT option described below. 3) SHUT: the well is shut at the surface and downhole, this results in no injection and no cross flow downhole. 4) AUTO: the well is initially SHUT, but may be opened automatically if an economic limit is violated. This option is currently not supported by OPM Flow. Note a well's STATUS should always be set either STOP or SHUT if the well's injection is to be set to zero. Just setting a well's injection rate to zero means that the well is open for injection with a zero rate, this will cause numerical issues especially for wells under THP control.			OPEN

No.	Name	Description			Default
		Field	Metric	Laboratory	
4	TARGET	A defined character string that sets the target injection control mode for the well. TARGET should be set to one of the following character strings: 1) RATE: the injection phase will be controlled by the surface fluid rate for the given well type as defined by the TYPE variable. For example, if TYPE has been set to WAT then this would mean the surface water injection rate as defined by item (5). 2) RESV: the injection phase will be controlled by the in situ reservoir volume fluid rate for the given well type as defined by the TYPE variable. For example, if TYPE has been set to GAS then this would mean the gas reservoir volume injection rate as defined by item (6). 3) BHP: the target rate is set according to the bottom-hole pressure as defined by item (7). 4) THP: the target rate is set according to the tubing head pressure as defined by item (8). If this option is selected then the vertical lift performance tables must be entered via the VFPINJ keyword in the SCHEDULE section and allocated to the well via item (9). 5) GRUP: the well is under group control and injects its share of the group's target as set using the GCONINJE keyword in the SCHEDULE section.			None
5	RATE	A real positive value that defines the maximum surface injection rate target or constraint.			
		Liquid stb/d Gas Mscf/d	Liquid sm^3/day Gas sm3/day	Liquid scc/hour Gas scc/hour	None
6	RESV	A real positive value that defines the maximum reservoir volume injection rate target or constraint.			
		rb/d	rm^3/day	rcc/hour	None
7	BHP	A real positive value that defines the maximum bottom-hole pressure target or constraint. Note the default value basically means unlimited injection or no constraint and should therefore be avoided as the BHP will result in unrealistic well potentials as well as optimistic injection forecasts for the well.			
		psia 10,0000	barsa 6,895	atma 6,803	Defined
8	THP	A real positive value that defines the maximum tubing head pressure target or constraint.			None
		psia	barsa	atma	

No.	Name	Description			Default
		Field	Metric	Laboratory	
9	VFPTAB	A positive integer greater than or equal to zero that defines the vertical lift performance tables to be used for calculating the tubing head pressure for the well. If a non-zero value is entered then the vertical lift performance tables must be entered via the VFPINJ keyword in the SCHEDULE section and allocated to the well via this item. The default value of zero implies no vertical lift performance tables and in this case TARGET cannot be set to THP and in addition item (10) should be defaulted or set to zero.			0
10	RSRVINJ	The dissolved gas-oil ratio in the injected oil, or the vaporized oil-gas ratio in the injected gas.			
		Gas Injection: stb/Mscf Oil Injection: Mscf/stb	Gas Injection: sm^3/sm^3 Oil Injection: sm^3/sm^3	Gas Injection: scc/scc Oil Injection: scc/scc	0.0
11	RSSTEAM	Thermal/Temperature gas-steam ratio for steam-gas injectors. The parameter is ignored by OPM Flow and should be defaulted or set to the default value of zero.			0.0
12	OILFRAC	Surface oil fraction in a multi-phase injector. The parameter is ignored by OPM Flow and should be defaulted or set to the default value of zero.			0.0
13	WATFRAC	Surface water fraction in a multi-phase injector. The parameter is ignored by OPM Flow and should be defaulted or set to the default value of zero.			0.0
14	GASFRAC	Surface gas fraction in a multi-phase injector. The parameter is ignored by OPM Flow and should be defaulted or set to the default value of zero.			0.0
15	OILSTEAM	Surface oil volume to steam volume ratio in a steam-oil injector. The parameter is ignored by OPM Flow and should be defaulted or set to the default value of zero.			0.0

Notes:

1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".

Table 12.80: WCONINJE Keyword Description

See also the GCONPROD and the GCONINJE keywords to define a group's production and injection targets and constraints, and the WCONPROD keyword to define a production well's targets and constraints. All the aforementioned keywords are described in the SCHEDULE section.

Example

The following example defines the injection targets and constraints for one gas injection well and one water injection well as follows:

```
--
--       WELL INJECTION CONTROLS
--
-- WELL  FLUID  OPEN/  CNTL  SURF   RESV   BHP   THP   VFP
-- NAME  TYPE   SHUT   MODE  RATE   RATE   PRSES PRES  TABLE
WCONINJE
GI01     GAS    OPEN   GRUP  50E3   1*     1*     1*    1*  /
WI01     WAT    OPEN   RATE  25E3   1*     5000.  1*    1*  /
/
```

Well GI01 is a gas injection well directly under group control constrained by a maximum surface gas injection rate of 50 MMscf/d and well WI01 is an open water injection well with a surface water injection rate target of 25,000 stb/d, subject to a maximum bottom-hole pressure constraint 5,000 psia.

12.3.254 WCONINJH – WELL HISTORICAL OBSERVED INJECTION RATES AND PRESSURES

Description

The WCONINJH keyword defines injection rates and pressures for wells that have been declared history matching wells by the use of this keyword. History matching wells are handled differently then ordinary wells that use the WCONINJE keyword for controlling their injection targets and constraints. However, the wells still need to be defined like ordinary injection wells using the WELSPECS keyword in the SCHEDULE section.

Note that although wells can be allocated to a group when they are specified by the WELSPECS keyword, history matching wells cannot operate under group control. Field and group reporting is still consistent for all wells allocated to a group, but history matching wells cannot be under group control.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WELNAME	A character string of up to eight characters in length that defines the well name for which the wells observed injection rates and pressures are being defined. Note that the well name (WELNAME) must have been declared previously using the WELSPECS keyword in the SCHEDULE section, otherwise an error may occur.			None
2	TYPE	A defined character string that defines the type of injection well. TYPE should be set to one of the following character strings: 1) GAS: for a gas injection well. 2) OIL: for an oil injection well. 3) WAT: for a water injection well.			None
3	STATUS	A defined character string that declares the status of the well. STATUS should be set to one of the following character strings: 1) OPEN: the well is open for injection and will attempt to inject the observed injection volumes. 2) STOP: the well is "stopped" at the surface and will not inject fluids; however, if there any open connections then flow may occur within the wellbore and between the open connections depending on a connection's potential with respect to all the other connections. Inter-connection flow (cross flow) can be prevented by setting the XFLOW variable on the WELSPECS keyword to NO. In this case the well's behavior will be similar to the SHUT option described below. 3) SHUT: the well is shut at the surface and downhole, this results in no injection and no cross flow downhole. Note a well's STATUS should always be set either STOP or SHUT if the well's injection is to be set to zero. Just setting a well's injection rate to zero means that the well is open to flow with a zero injection rate, this may cause numerical issues.			OPEN

No.	Name	Description			Default
		Field	Metric	Laboratory	
4	RATE	A real positive value that defines the observed surface injection rate.			
		Liquid stb/d Gas Mscf/d	Liquid sm^3/day Gas sm3/day	Liquid scc/hour Gas scc/hour	0.0
5	BHP	A real positive value that defines the observed bottom-hole pressure.			
		psia	barsa	atma	0.0
6	THP	A real positive value that defines the observed tubing head pressure. This parameter is only used for comparing the actual tubing head pressure given here with those calculated by the simulator, that is history matching wells can only be controlled by either the surface injection rate or their bottom-hole pressure.			
		psia	barsa	atma	None
7	VFPTAB	A positive integer greater than or equal to zero that defines the vertical lift performance tables to be used for calculating the tubing head pressure for the well. If a non-zero value is entered then the vertical lift performance tables must be entered via the VFPINJ keyword in the SCHEDULE section and allocated to the well via this item. The default value of zero implies no vertical lift performance table initially. If this value is then reset to be greater than zero then the table will be used to calculate the well's tubing head pressure. Subsequently, the default is to use the previously declared table number.			0
8	RSRVINJ	Dissolved gas fraction in injected oil or vaporized oil fraction in injected gas. The parameter is ignored by OPM Flow and should be defaulted or set to the default value of zero.			0.0
9	OILFRAC	Surface oil fraction in a multi-phase injector. The parameter is ignored by OPM Flow and should be defaulted or set to the default value of zero.			0.0
10	WATFRAC	Surface water fraction in a multi-phase injector. The parameter is ignored by OPM Flow and should be defaulted or set to the default value of zero.			0.0
11	GASFRAC	Surface gas fraction in a multi-phase injector. The parameter is ignored by OPM Flow and should be defaulted or set to the default value of zero.			0.0
12	TARGET	A defined character string that sets the target injection control mode for the well. TARGET should be set to one of the following character strings: 1) RATE: the injection well will be controlled by the surface injection rate for the given well type as defined by the TYPE variable. For example, if TYPE has been set to WAT then this would mean the surface water injection rate as defined by item (4). 2) BHP: the injection well will be controlled by the bottom-hole pressure as defined by item (5).			RATE

No.	Name	Description			Default
		Field	Metric	Laboratory	
Notes: 1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".					

Table 12.81: WCONINJH Keyword Description

This keyword should be repeated at various time steps to fully describe the historical injection performance of the wells. For example, as most production and injection data is reconciled on a monthly basis, then monthly time steps covering the injection history of the wells should be used with WCONINJH keyword entered on a monthly basis.

History matching well are converted to ordinary wells by restating a well's control mode using either the WCONINJE or WELTARG keywords in the SCHEDULE section.

Example

The following example below shows the observed gas rates for the GI01 gas injector for the first quarter of 2000.

```
-- ------------------------------------------------------------------------------
-- 01 JAN 2000 START OF SCHEDULE SECTION
-- ------------------------------------------------------------------------------
--
--       WELL HISTORICAL INJECTION CONTROLS
--
-- WELL  FLUID  OPEN/  SURF   BHP   THP   VFP    NOT   CNTL
-- NAME  TYPE   SHUT   RATE   PRES  PRES  TABLE  USED  MODE
WCONINJH
GI01     GAS    OPEN  15.5E3  1*    5462   12    4*    1*  /
/
DATES
01 FEB 2000 /
/
--
--       WELL HISTORICAL INJECTION CONTROLS
--
-- WELL  FLUID  OPEN/  SURF   BHP   THP   VFP    NOT   CNTL
-- NAME  TYPE   SHUT   RATE   PRES  PRES  TABLE  USED  MODE
WCONINJH
GI01     GAS    OPEN  15.9E3  1*    5468   1*    4*    1*  /
/
DATES
01 MAR 2000 /
/
```

```
--
--       WELL HISTORICAL INJECTION CONTROLS
--
-- WELL  FLUID  OPEN/  SURF   BHP   THP   VFP    NOT   CNTL
-- NAME  TYPE   SHUT   RATE   PRES  PRES  TABLE  USED  MODE
WCONINJH
GI01     GAS    OPEN  17.2E3  1*    5489   1*    4*    1*  /
/
```

Well GI01is declared as a gas injection well under gas rate control as TARGET variable is defaulted to rate control by using 1* (the last entry on the record). In addition, the well uses vertical lift table VFPINJ number 12 (as shown at January 1, 2000) to calculate the tubing head pressures for the well. Note that it is not necessary to declare the VFPINJ table number if it remains the same for subsequent time steps and thus the default 1* is used to indicate the last entry should be used.

12.3.255 WCONINJP – DEFINE WELL INJECTION TARGETS AND CONSTRAINTS FOR PATTERN FLOOD WELLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, WCONINJP, defines well injection targets and constraints for pattern flood wells. The keyword is similar to the WCONINJE keyword in the SCHEDULE section except that the injection control is applied to a group of wells defined by the first record of this keyword, combined with a second record that defines the wells in the pattern and their contribution to the pattern.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.256 WCONPROD – DEFINE WELL PRODUCTION TARGETS AND CONSTRAINTS

Description

The WCONPROD keyword defines production targets and constraints for wells that have previously been defined by the WELSPECS keyword in the SCHEDULE section. Note that wells can be allocated to a group when they are specified by the WELSPECS keyword. Wells defined to be under group control will have their production rates controlled by the group to which they belong, in addition to any well constraints defined for the wells using this keyword.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WELNAME	A character string of up to eight characters in length that defines the well name for which the well production targets and constraints data are being defined. Note that the well name (WELNAME) must have been declared previously using the WELSPECS keyword in the SCHEDULE section, otherwise an error may occur.			None
2	STATUS	A defined character string that declares the status of the well. STATUS should be set to one of the following character strings: 1) OPEN: the well is open to flow and will attempt to produce the required production volumes. 2) STOP: the well is "stopped" at the surface and will not produce any fluids to surface; however, if there any open connections then flow may occur within the wellbore and between the open connections depending on a connection's potential with respect to all the other connections. Inter-connection flow (cross flow) can be prevented by setting the XFLOW variable on the WELSPECS keyword to NO. In this case the well's behavior will be similar to the SHUT option described below. 3) SHUT: the well is shut at the surface and downhole, this results in no flow at the surface and no cross flow downhole. 4) AUTO: the well is initially SHUT, but may be opened automatically if an economic limit is violated. This option is currently not supported by OPM Flow. Note a well's STATUS should always be set either STOP or SHUT if the well's production is to be set to zero. Just setting a well's production rate to zero means that the well is open to flow with a zero rate, this will cause numerical issues especially for wells under THP control.			OPEN

No.	Name	Description			Default
		Field	Metric	Laboratory	
3	TARGET	A defined character string that sets the target production phase for the well, all the other phases will therefore act as constraints. The simulator will attempt to meet the TARGET based on the phase rate stated in items (4) to (10) on this keyword. TARGET should be set to one of the following character strings: 1) ORAT: the target is set to the surface oil production rate as defined by item (4). 2) WRAT: the target is set to the surface water production rate as defined by item (5). 3) GRAT: the target is set to the surface gas production rate as defined by item (6). 4) LRAT: the target is set to the surface liquid (oil plus water) production rate as defined by item (7). 5) RESV: the target is set to the in situ reservoir volume rate as defined by item (8). 6) BHP: the target rate is set to the bottom-hole pressure as defined by item (9). 7) THP: the target rate is set to the tubing head pressure as defined by item (10). If this option is selected then the vertical lift performance tables must be entered via the VFPPROD keyword in the SCHEDULE section and allocated to the well via item (11). 8) GRUP: the well is under group control and produces its share of the group's target as set using the GCONPROD keyword in the SCHEDULE section. Here the default tokens of 1* or '' may be used, and both default tokens behave in the same manner; however, the actual default value used when TARGET is defaulted is dependent on the data entered on this keyword as described below: 1) If the well's group parameters have not been defined via the GCONPROD keyword, in the SCHEDULE section, then TARGET is set to the first non-defaulted hydrocarbon rate (ORAT, GRAT, LRAT, or RESV). If all the rates are defaulted and a value for BHP is entered, then TARGET is set to BHP control. Similarly, if BHP is also defaulted, and THP has been entered, then TARGET is set to THP control. Finally, if all parameters are defaulted, then the default value for BHP, one atmosphere, will be used with BHP control. 2) If, and only if, the well's group parameters have been defined via the GCONPROD keyword, then TARGET is set equal to GRUP, and the remaining parameters on the WCONPROD keyword are used as well constraints. Note the default value of one atmosphere should be avoided as the BHP will result in unrealistic well potentials as well as optimistic production forecasts for the well.			1*
4	ORAT	A real positive value that defines the maximum surface oil production rate target or constraint.			
		stb/day	sm^3/day	scc/hour	None
5	WRAT	A real positive value that defines the maximum surface water production rate target or constraint.			None

No.	Name	Description			Default
		Field	Metric	Laboratory	
		stb/day	sm^3/day	scc/hour	
6	GRAT	A real positive value that defines the maximum surface gas production rate target or constraint			
		Mscf/day	sm^3/day	scc/hour	None
7	LRAT	A real positive value that defines the maximum surface liquid (oil plus water) production rate target or constraint.			
		stb/day	sm^3/day	scc/hour	None
8	RESV	A real positive value that defines the maximum reservoir volume production rate target or constraint.			
		rtb/day	rm^3/day	rcc/hour	None
9	BHP	A real positive value that defines the minimum bottom-hole pressure target or constraint. Note the default value of one atmosphere should be avoided as the BHP will result in unrealistic well potentials as well as optimistic production forecasts for the well.			
		psia 14.70	barsa 1.01325.	atma 1.0	Defined
10	THP	A real positive value that defines the minimum tubing head pressure target or constraint. Note the default value of zero should be avoided if the well's control TARGET has been set to THP, as this will result in optimistic production forecasts for a well, since a well must flow against a back pressure imposed by the surface facilities.			
		psia 0.0	barsa 0.0	atma 0.0	Defined
11	VFPTAB	A positive integer greater than or equal to zero that defines the vertical lift performance tables to be used for calculating the tubing head pressure for the well. If a non-zero value is entered then the vertical lift performance tables must be entered via the VFPPROD keyword in the SCHEDULE section and allocated to the well via this item. The default value of zero implies no vertical lift performance tables and in this case TARGET cannot be set to THP and in addition item (10) should be defaulted or set to zero.			0
12	ALQ-WELL	A real positive value that defines the artificial lift quantity to be used in conjunction with the VFPPROD assigned to the well via the VFPTAB variable. VFPTAB vertical lift performance table and the artificial lift quantity ALQ-WELL are used with the well fluid rates to calculate the well's tubing head pressures values from the bottom-hole pressure. Note that the units for ALQ-WELL are dependent on the associated variable on the VFPPROD keyword.			0.0

No.	Name	Description			Default
		Field	Metric	Laboratory	
13	WGASRATE	Wet gas production rate used in the commercial compositional simulator. Not used and should be defaulted with 1*.			1*
14	MOLARATE	Total molar rate used in the commercial compositional simulator. Not used and should be defaulted with 1*.			1*
15	STEAMRAT	Thermal/Temperature steam rate (Cold Water Equivalent) for steam producers used in the commercial compositional simulator. Not used and should be defaulted with 1*.			1*
16	DELTAP	Thermal/Temperature delta pressure offset for steam producers used in the commercial compositional simulator. Not used and should be defaulted with 1*.			1*
17	DELTAT	Thermal/Temperature delta temperature offset for steam producer used in the commercial compositional simulator. Not used and should be defaulted with 1*.			1*
18	CALRATE	Calorific production rate used in the commercial compositional simulator. Not used and should be defaulted with 1*.			1*
19	COMBPROC	Linearly combined procedure for when exceeding COMBRATE, used in the commercial compositional simulator. Not used and should be defaulted with 1*.			1*
20	NGL	A real positive value that defines the observed Natural Gas Liquid ("NGL") rate in the commercial compositional simulator. Not used and should be defaulted with 1*.			1*

Notes:

1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".

Table 12.82: WCONPROD Keyword Description

See also the GCONPROD and GCONINJE keywords to define a group's production and injection targets and constraints, and the WCONINJE keyword to define an injection well's targets and constraints. All the aforementioned keywords are described in the SCHEDULE section.

Example

The following example defines the production targets and constraints for five wells as follows:

```
--
--        WELL PRODUCTION WELL CONTROLS
--
-- WELL   OPEN/   CNTL   OIL    WAT    GAS    LIQ     RES     BHP     THP    VFP     VFP
-- NAME   SHUT    MODE   RATE   RATE   RATE   RATE    RATE    PRES    PRES   TABLE   ALFQ
WCONPROD
OP01      OPEN    GRUP   5E3    1*      1*     1*     1*      500.0                          /
OP02      OPEN    GRUP  10E3    1*      1*     1*     1*      200.0   500.0   2      0.0     /
OP03      OPEN    GRUP  15E3    1*      1*     1*     1*      200.0   500.0   3     10.0     /
OP04      OPEN    ORAT  20E3    1*      1*     1*     1*      500.0                          /
OP05      SHUT    GRUP  20E3    1*      1*     1*     1*      500.0                          /
/
```

Well OP01 is open and is on group control, subject to a maximum oil rate constraint of 5,000 stb/d and a minimum bottom-hole pressure of 500 psia. OP02 is also open and on group control but it's maximum oil rate constraint has been set 10,000 stb/d, and is subject to a minimum bottom-hole pressure limit of 200 psia and a minimum tubing head pressure limit of 500 psia using VFPPROD vertical lift table number two. Well OP03 is very similar to OP02, but with a 15,000 stb/d maximum oil constraint and using VFPPROD vertical lift table number three with an artificial lift parameter of 10. The next well is not on group control. Well OP04 is open and has an oil rate target of 20,000 stb/d, subject to a minimum bottom-hole pressure of 500 psia. Finally, well OP05 is shut and will not be brought back on production despite being put under group control, as the well has been declared shut.

The next example defines the production targets and constraints for five wells, of which well OP01 is under group control as the well's group target and constraints have been set with the GCONPROD keyword, and wells OP02 to OP05 belong to groups that have not had their group constraints set by GCONPROD.

```
--
--        GROUP PRODUCTION CONTROLS
--
-- GRUP   CNTL   OIL     WAT     GAS     LIQ     CNTL   GRUP   GUIDE   GUIDE   CNTL
-- NAME   MODE   RATE    RATE    RATE    RATE    OPT    CNTL   RATE    DEF     WAT
GCONPROD
GRP01     SAT    25E3    1*      1*      1*      1*      1*     1*      1*      1*       /
/
--
--        WELL SPECIFICATION DATA
--
-- WELL   GROUP      LOCATION   BHP     PHASE   DRAIN   INFLOW   OPEN   CROSS   PVT
-- NAME   NAME         I     J   DEPTH   FLUID   AREA    EQUANS   SHUT   FLOW    TABLE
WELSPECS
OP01      SAT         14    13    1*      OIL     1*      P-P      SHUT    NO      1*      /
0P02      PLATFORM    64    80    1*      OIL     1*      GPP      SHUT    NO      1*      /
OP03      PLATFORM    24   110    1*      OIL     1*      STD      SHUT    NO      1*      /
OP04      PLATFORM    24   110    1*      OIL     1*      STD      SHUT    NO      1*      /
OP05      PLATFORM    24   110    1*      OIL     1*      STD      SHUT    NO      1*      /
/
```

```
--
--       WELL PRODUCTION WELL CONTROLS
--
-- WELL  OPEN/  CNTL  OIL   WAT   GAS   LIQ    RES    BHP    THP   VFP    VFP
-- NAME  SHUT   MODE  RATE  RATE  RATE  RATE   RATE   PRES   PRES  TABLE  ALFQ
WCONPROD
OP01     OPEN   1*   15E3    1*    1*    1*     1*     500.0                  /
OP02     OPEN   1*    1*    5E3*  15E6   1*     1*     200.0  500.0  2      0.0 /
OP03     OPEN   1*    1*     1*    1*    1*     1*     1*     500.0  3     10.0 /
OP04     OPEN   ''    1*     1*    1*    1*     1*     1*     1*     1*     1*  /
OP05     SHUT   1*    1*     1*    1*    1*     1*     1*     1*     1*     1*  /
/
```

Here, well OP01's control mode, TARGET, will be set to group control (GRUP) and the well's production parameters will all act as constraints. For well OP02, the well's control mode is set to GRAT, and the other production parameters all act as constraints. Well OP03 control mode is set to THP and if the well's BHP value has been previously defined then that value will be used as a constraint; otherwise the default value of one atmosphere will be used instead. Finally, wells OP04 and OP05 will have their control mode set to BHP control. Note in this case, OP04's status has been set to OPEN; thus the well will produce at a target BHP of one atmosphere, which is unrealistic.

12.3.257 WCUTBACK – DEFINE WELL CUTBACK LIMITS AND PARAMETERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, WCUTBACK, defines a well's cutback limits and parameters for both production and injection wells. See also the GCUTBACK keyword in the SCHEDULE section that provides similar functionality for groups.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.258 WCUTBACT – DEFINE WELL TRACER CUTBACK LIMITS AND PARAMETERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, WCUTBACT, defines a production well's cutback limits and parameters based on the named produced tracer from the well. See also the GCUTBACT keyword in the SCHEDULE section that provides similar functionality for groups.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.259 WCYCLE – DEFINE AUTOMATIC WELL OPENING AND CLOSING CYCLING PARAMETERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The WCYCLE keyword defines automatic well opening and closing cycling parameters used to model "huff-and puff" cyclic steam injection in heavy oil reservoirs or Water-Alternating-Gas ("WAG") processes in enhanced oil recovery modeling. The keyword defines specific time periods for automatically cycling wells on and off. For example in a WAG scheme the water injection wells would have one set of cycling parameters and the gas injection wells another, such that only one type of well is active at a time.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.260 WDFAC – DEFINE GAS FLOW DEPENDENT SKIN FACTOR

Description

The WDFAC keyword defines a gas well's connection D-factor, which is normally derived from well tests or calculated analytically based on the coefficient of inertial resistance, usually known as β, in Forchheimer's flow equation[322, 323] and [324].

See also the WDFACCOR keyword in the SCHEDULE section that uses Dake's[325] correlation to calculate the D-factor.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

[322] *Geertsma, J., 1974. Estimating the Coefficient of Inertial Resistance in Fluid Flow Through Porous Media. Soc.Pet.Eng.J., October: 445-450.*

[323] *Gewers, C.W.W. and Nichol, L.R., 1969. Gas Turbulence Factor in a Microvugular Carbonate. J.Can.Pet.Tech., April.*

[324] *Wong, S.W., 1970. Effects of Liquid Saturation on Turbulence Factors for Gas Liquid Systems. J.Can.Pet.Tech., October*

[325] *Dake, L.P. Fundamentals of Reservoir Engineering, Amsterdam, The Netherlands, Elsevier Science BV (1978) Chapter 8.6, pages 252-257.*

12.3.261 WDFACCOR – GAS FLOW DEPENDENT SKIN FACTOR (CORRELATION)

Description

WDFACCOR keyword defines the parameters to calculate a gas well's connection D-factor based on a correlation for the coefficient of inertial resistance, usually known as β, in Forchheimer's flow equation[326,327,328] and [329]. This keyword uses Dake's correlation to calculate the D-factor.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

[326] *Dake, L.P. Fundamentals of Reservoir Engineering, Amsterdam, The Netherlands, Elsevier Science BV (1978) Chapter 8.6, pages 252-257.*

[327] *Geertsma, J., 1974. Estimating the Coefficient of Inertial Resistance in Fluid Flow Through Porous Media. Soc.Pet.Eng.J., October: 445-450.*

[328] *Gewers, C.W.W. and Nichol, L.R., 1969. Gas Turbulence Factor in a Microvugular Carbonate. J.Can.Pet.Tech., April.*

[329] *Wong, S.W., 1970. Effects of Liquid Saturation on Turbulence Factors for Gas Liquid Systems. J.Can.Pet.Tech., October*

12.3.262 WDRILPRI – ADD WELLS TO THE DRILLING PRIORITY DRILLING QUEUE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, WDRILPRI, adds wells to the Drilling Priority Drilling Queue and defines the well priority and drilling unit number or batch queue sequence for the well. The batch queue sequence number enables all wells with the same sequence number to drilled at the same time.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.263 WDRILRES – ACTIVATE PREVENTION OF MULTI-COMPLETIONS IN THE SAME CELL FOR QUEUED WELLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The WDRILRES keyword activates the prevention of multi-completions being completed in the same cell for wells in a drilling queue. Setting this option stops any well defined as a queued well via the QDRILL and WDRILLPRI keywords in the SCHEDULE section, or any wells set to automatic opening by setting the STATUS variable to AUTO on the WCONPROD keyword in the RUNSPEC section, from opening if there is an already existing active well connection to a cell.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.264 WDRILTIM – DEFINE DRILLING PARAMETERS FOR AUTOMATIC DRILLING OF NEW WELLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

WDRILTIM defines the automatic drilling parameters used to describe the numbers of days taken to drill a well, the drilling status of the well, and status of other wells when drilling an automatically drilled well.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.265 WECON – WELL ECONOMIC CRITERIA FOR PRODUCTION WELLS

Description

The WECON keyword defines the economic criteria for production wells that have previously been defined by the WELSPECS and WCONPROD keywords in the SCHEDULE section.

Note that wells can be allocated to a group when they are specified by the WELSPECS keyword and groups can also have economic controls. Wells under group control are therefore subject to the economic criteria set via the GCONPROD and GECON keywords in the SCHEDULE section and the controls specified by the WECON keyword. Note that GECON is not supported by OPM Flow in the current release.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WELNAME	A character string of up to eight characters in length that defines the well name for which the well economic criteria data is being defined. Note that the well name (WELNAME) must have been declared previously using the WELSPECS keyword in the SCHEDULE section, otherwise an error may occur.			None
2	ORAT	A real positive value that defines the minimum economic surface oil production rate, below which an economic action will take place, as outlined below: 1) If there are any remaining connections in the well with the STATUS variable set to AUTO on the COMPDAT keyword in the SCHEDULE section, then one of these connections (or completion) will be opened. 2) If there are no remaining connections in the well with the STATUS variable set to AUTO on the COMPDAT keyword, then the well will be shut or stopped as requested by item (9) of the WELSPECS keyword. Only option (2) is supported by OPM Flow as STATUS equals AUTO on the COMPDAT keyword is currently not supported by the simulator. Hence, the well be either shut or stopped. A value less than or equal to zero switches off this criterion.			
		stb/d	sm^3/day	scc/hour	0.0

No.	Name	Description			Default
		Field	Metric	Laboratory	
3	GAS	A real positive value that defines the minimum economic surface gas production rate, below which an economic action will take place, as outlined below: 1) If there are any remaining connections in the well with the STATUS variable set to AUTO on the COMPDAT keyword in the SCHEDULE section, then one of these connections (or completion) will be opened. 2) If there are no remaining connections in the well with the STATUS variable set to AUTO on the COMPDAT keyword, then the well will be shut or stopped as requested by item (9) of the WELSPECS keyword. Only option (2) is supported by OPM Flow as STATUS equals AUTO on the COMPDAT keyword is currently not supported by the simulator. Hence, the well be either shut or stopped. A value less than or equal to zero switches off this criterion.			
		Mscf/d	sm^3/day	scc/hour	0.0
4	WCUT	A real positive value that defines the maximum economic surface water cut, above which an economic action will take place. Water cut is defined as: $f_w = \frac{q_w}{q_w + q_o}$, and the various actions that are available if the water cut limit is exceeded are described in item (7). A value less than or equal to zero switches off this criterion.			
		dimensionless	dimensionless	dimensionless	0.0
5	GOR	A real positive value that defines the maximum economic surface gas-oil ratio, above which an economic action will take place, as defined by item (7). A value less than or equal to zero switches off this criterion.			
		Mscf/stb	sm^3/sm^3	scc/scc	0.0
6	WGR	A real positive value that defines the maximum economic surface water-gas ratio, above which an economic action will take place, as defined by item (7). A value less than or equal to zero switches off this criterion. Note that this feature is currently not supported in OPM Flow and should be defaulted.			
		stb/Mscf	sm^3/sm^3	scc/scc	0.0

No.	Name	Description			Default
		Field	Metric	Laboratory	
7	ACTION	A defined character string that defines the action to be taken if the economic WCUT, GOR, or WGR limits are violated. ACTION should be set to one of the following character strings: 1) NONE: no action is taken. 2) CON: close the worst offending connection. If connections have been grouped as completions then the worst offending completion will be closed. 3) +CON: close the worst offending connection and all below it. If connections have been grouped as completions then the worst offending completion and all below it will be closed. 4) WELL: shut or stop the well as per the AUTO variable on the 5) WELSPECS keyword. The corrective action takes places at the end of the time step in which the constraint is violated. Only ACTION equal to CON and NONE are are currently supported by OPM Flow.			NONE
8	END	A defined character string that defines if the simulation should terminate if the well is shut or stopped. END should be set to one of the following character strings: 1) NO: no action is taken and the run continues. 2) YES: terminate the run at the next report time step. Only the default value of NO is supported in OPM Flow.			NO
9	WELOPEN	A character string of up to eight characters in length that defines the well name for the well which will be opened when WELNAME has been automatically closed/shut by the simulator. Wells closed manually do not invoke this opton. The well name (WELOPEN) must have been declared previously using the WELSPECS keyword in the SCHEDULE section, otherwise an error may occur. Note that this feature is currently not supported in OPM Flow and should be defaulted.			None

No.	Name	Description			Default
		Field	Metric	Laboratory	
10	ELTOPT	A defined character string that defines the type of quantity used to test the economic limit, and should be set to one of the following character strings: 1) POTN: The Economic Limit Test ("ELT") is based on a well's potential rate. A well's potential rate is calculated using only the flowing bottom-hole pressure and tubing head pressure constraints, all other constraints are ignored, including group constraints. However, if the WELDRAW keyword in the SCHEDULE section has been used to limit a well's pressure draw down, then this will also be considered in the potential calculation, as it influences the bottom-hole pressure constraint. 2) RATE: The economic limit is based a well's surface production rate as oppose to the potential rate, this is the default behavior. Wells under group control via the GCONPROD keyword in the SCHEDULE section will be subject to the ELT, except for when the group's production target rate is set to zero. Wells under group control via the GCONPRI keyword in the SCHEDULE section will be subject to the ELT, except for when the group has curtailed a well's production. Both options are supported in OPM Flow.			RATE
11	WCUT2	A real positive value that defines the secondary maximum economic surface water cut, above which an economic action will take place. A value less than or equal to zero switches off this criterion. Note that this feature is currently not supported in OPM Flow and should be defaulted.			0.0
12	ACTION2	A defined character string that defines the action to be taken if the secondary economic WCUT2 limits is violated. Note that this feature is currently not supported in OPM Flow and should be defaulted with 1*.			1*
13	GLR	A real positive value that defines the maximum economic surface gas-liquid ratio, above which an economic action will take place, as defined by item (7). A value less than or equal to zero switches off this criterion. Note that this feature is currently not supported in OPM Flow and should be defaulted.			None
14	LRAT	A real positive value that defines the minimum liquid rate, below which the well (WELNAME) is shut-in. Note that this feature is currently not supported in OPM Flow and should be defaulted.			0.0
15	TEMP	A real positive value that defines the maximum economic temperature in the commercial compositional Thermal/Temperature model for a well. Note that this feature is not supported in OPM Flow and should be defaulted.			None

No.	Name	Description			Default
		Field	Metric	Laboratory	
16	RESV	A real positive value that defines the minimum reservoir volume rate, below which the well (WELNAME) is shut-in. Note that this feature is currently not supported in OPM Flow and should be defaulted.			0.0

Notes:

1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".

Table 12.83: WECON Keyword Description

See also the WELSPECS keyword to define a wells shut-in or stop options, GCONPROD for group controls, and GECON for setting a group's economic criteria. All the aforementioned keywords are described in the SCHEDULE section.

Example

The following example defines one oil well and one gas well using the WELSPECS keyword, together with their economic criteria.

```
--
-- WELL SPECIFICATION DATA
--
-- WELL      GROUP       LOCATION   BHP     PHASE   DRAIN   INFLOW   SHUT   CROSS   PRESS
-- NAME      NAME         I     J    DEPTH   FLUID   AREA    EQUA.    IN     FLOW    TABLE
WELSPECS
GP01       PLATFORM      14    13   1*       GAS    1*      GPP      SHUT    NO     1*   /
OP01       PLATFORM      28    96   1*       OIL    1*      STD      SHUT    NO     1*   /
/
--
--        WELL ECONOMIC CRITERIA FOR PRODUCTION WELLS
-- WELL   MIN     MIN     MAX     MAX     MAX     CNTL     END
-- NAME   ORAT    GRAT    WCUT    GOR     WGR     MODE     RUN
WECON
GP01      1*      5.0E3   1*      1*      1*     'WELL'   'NO'                        /
OP01      500     1*      0.95    15E3    1*     'WELL'   'YES'                       /
/
```

Well GP01 has a minimum economic gas rate of 5 MMscf/d and will shut-in if the gas rate falls below this rate, but the simulation will continue even if this occurs. Well OP02 as a minimum economic oil rate of 500 stb/d, a maximum water cut limit of 95%, and a maximum GOR of 15 MMscf/d, if any any of these limits are violated the well will be shut-in and the run terminated at the next reporting time step.

12.3.266 WECONINJ – WELL ECONOMIC CRITERIA FOR INJECTION WELLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The WECONINJ keyword defines economic criteria for injection wells that have previously been defined by the WELSPECS and WCONINJE keywords in the SCHEDULE section.

Note that wells can be allocated to a group when they are specified by the WELSPECS keyword and groups can also have economic controls. Wells under group control are therefore subject to the economic criteria set via the GCONINJE keyword in the SCHEDULE section and the controls specified by this keyword.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WELNAME	A character string of up to eight characters in length that defines the well name for which the well economic injection criteria data is being defined. Note that the well name (WELNAME) must have been declared previously using the WELSPECS keyword in the SCHEDULE section, otherwise an error may occur.			None
2	MINVALUE	A real positive value that defines the minimum economic injection value, below which an economic action will take place, as defined by the AUTO parameter on the WELSPECS keyword (SHUT or STOP). Note that TYPE determines if the minimum value is applied to the well's actual injection rate or the well's potential. A value less than or equal to zero switches off this criterion.			
		Liquid stb/d Gas Mscf/d	Liquid sm^3/day Gas sm3/day	Liquid scc/hour Gas scc/hour	0.0
3	TYPE	A defined character string that determines if MINVALUE is applied to a well's actual rate or potential, and should be set to one of the following: 1) RATE: In this case the MINVALUE is applied to a well's actual rate. 2) POTN: Here, MINVALUE is applied to the well's potential with only the BHP and THP constraints applied. The default value is RATE.			RATE

Notes:

1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".

Table 12.84: WECONINJ Keyword Description

See also the WELSPECS keyword to define a wells shut-in or stop options and GCONINJE for group controls in the SCHEDULE section.

Example

The following example defines the economic injection parameters for all gas and water injection wells.

```
--
--       WELL ECONOMIC LIMIT DATA FOR INJECTION WELLS
--
-- WELL  MIN    RATE
-- NAME  VALUE  POTN
WECONINJ
GI*      2.0E3  RATE                                                    /
WI*      5.0E3  POTN                                                    /
/
```

Here all the gas injection wells have a minimum economic gas injection rate of 2 MMscf/d and the water injection wells have a minimum water potential rate of 5,000 stb/d. The AUTO parameter on the WELSPECS keyword will determine if the wells will be shut-in or stopped.

12.3.267 WECONT – WELL ECONOMIC TRACER CRITERIA FOR PRODUCTION WELLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The WECONT keyword defines the tracer economic criteria for production wells that have previously been defined by the WELSPECS and WCONPROD keywords in the SCHEDULE section, for tracers define by the TRACER keyword in the PROPS section.

Note that wells can be allocated to a group when they are specified by the WELSPECS keyword and groups can also have economic controls. Wells under group control are therefore subject to the economic criteria set via the GECONT keyword in the SCHEDULE section and the controls specified by this keyword. Note that GECONT is not supported by OPM Flow in the current release

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1-1	WELNAME	A character string of up to eight characters in length that defines the well name for which the well target and constraints are being defined.			None
1-2	ACTION	A defined character string that defines the action to be taken if the economic WCUT, GOR, or WGR limits are violated. ACTION should be set to one of the following character strings: 1) NONE: no action is taken. 2) CON: close the worst offending connection in the worst offending. If connections have been welled as completions then the worst offending completion will be closed. 3) +CON: close the worst offending connection and all below it in the worst offending well. If connections have been welled as completions then the worst offending completion and all below it in the worst offending well will be closed. 4) WELL: shut or stop the well as per the AUTO variable on the WELSPECS keyword. 5) PLUG: plug back the worst offending well based on the plug back length and options defined on the WPLUG keyword in the SCHEDULE. The corrective action takes places at the end of the time step in which the constraint is violated.			None
1-3	END	A defined character string that defines if the simulation should terminate if the well is shut or stopped. END should be set to one of the following character strings: 1) NO: no action is taken and the run continues. 2) YES: terminate the run at the next report time step.			NO
1-4	WELL	A character string of up to eight characters in length that defines the well name of a fully defined well that will be "opened" when the well WELNAME is shut-in or stopped.			None

No.	Name	Description			Default
		Field	Metric	Laboratory	
1-5	/	Record one terminated by a "/"			Not Applicable
2-1	NAME	A three letter character string defining the tracer's name. Note it is best to void names beginning with the letters F, S, and T as these names may create naming issues in post-processing software.			None
2-2	MXTOTAL	A real positive value that defines the maximum total (free plus solution) tracer rate. Tracer units are per those defined by the carrying fluid, oil, gas, water, etc.			None
2-3	MXFREET	A real positive value that defines the maximum total (free plus solution) tracer concentration. Tracer units are per those defined by the carrying fluid, oil, gas, water, etc.			None
2-4	MXFREEQ	A real positive value that defines the maximum free tracer rate. Tracer units are per those defined by the carrying fluid, oil, gas, water, etc.			None
2-5	MXCONC	A real positive value that defines the maximum free tracer concentration. Tracer units are per those defined by the carrying fluid, oil, gas, water, etc.			None
2-6	MXSOLNQ	A real positive value that defines the maximum solution rate. Tracer units are per those defined by the carrying fluid, oil, gas, water, etc.			None
2-7	MXSOLNC	A real positive value that defines the maximum solution concentration. Tracer units are per those defined by the carrying fluid, oil, gas, water, etc.			
2-8	/	Record two terminated by a "/"			Not Applicable
3-1	/	Well terminated by a "/"			Not Applicable

Notes:

1) WECONT keyword consists of two records, with entries 1-1 to 1-5 representing record one items and 2-1 to 2-8 representing record number two items, in the "No." column in this table. A maximum of three type two records can be entered following a type one record.
2) Each type one and type two records are terminate by a "/" as indicated in the table, and a well data set is terminated by a further "/", after which additional well data sets can be entered stating with a record of type one followed by type two.
3) The keyword the keyword should be terminated by an additional "/" after the well data set termination "/" character.

Table 12.85: WECONT Keyword Description

See also the WELSPECS keyword to define a wells shut-in or stop options, and WECON for setting a well's economic criteria. Both the aforementioned keywords are described in the SCHEDULE section.

Example

The following example defines the tracer economic criteria for the field and two wells, OP01 and OP02.

```
--
--       WELL TRACER ECONOMIC CRITERIA FOR PRODUCTION WELLS
--
-- WELL  WORK    END    MAX
-- NAME  OVER    RUN    WELLS
WECONT
OP01     +CON   'YES'   1*                                        / START OF WELL
--
--       TRACER  TRACER  TRACER  TRACER  TRACER  TRACER  TRACER
--       NAME    TOTAL   TOTAL   FREE    FREE    SOLN    SOLN
--               RATE    CONCEN  RATE    CONCEN  RATE    CONCEN
         PLY     800.0                                      /
         BRI     800.0                                      /
                                                            /
OP02     +CON   'YES'   1*                                        / START OF WELL
--
--       TRACER  TRACER  TRACER  TRACER  TRACER  TRACER  TRACER
--       NAME    TOTAL   TOTAL   FREE    FREE    SOLN    SOLN
--               RATE    CONCEN  RATE    CONCEN  RATE    CONCEN
         PLY     800.0                                      /
         BRI     800.0                                      / END OF WELL
                                                            / END OF KEYWORD
```

If the economic limits are violated then the worst offending connection and all below it in the worst offending well will be closed, If connections have been grouped as completions then the worst offending completion and all below it in the worst offending well will be closed

12.3.268 WEFAC – DEFINE WELL EFFICIENCY

Description

Defines a well's efficiency or up-time as opposed to setting the efficiency at the group level. Note that wells are allocated to a group when they are specified by the WELSPECS keyword and groups can also have efficiency factors.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WELNAME	A character string of up to eight characters in length that defines the well name for which the well efficient factor is being defined. Note that the well name (WELNAME) must have been declared previously using the WELSPECS keyword in the SCHEDULE section, otherwise an error may occur.			None
2	FACTOR	A real positive value greater than zero and less than or equal to one that defines the efficiency factor for the well. If a well's down time is 5% then FACTOR should be set to 0.95 (1.0 – 0.05). Noted that well pressures and rates are calculated at their full flowing conditions but subject to any operating constraints, that is without the well efficiency being applied (FACTOR), in order to represent the actual flowing conditions in the field. The effective rates and volumes are calculated by applying FACTOR when summing individual well rates to their group level and up, including summing to the top most group FIELD. In terms of a well's cumulative production, FACTOR is applied to the well rate times the time interval for the time step. This ensures that correct effective volume is withdrawn (or injected) from (or to) the reservoir. This approach means that wells are effectively arbitrarily offline for a small period during a time step, as oppose to all wells going offline concurrently. And thus the group and field rates and volumes are the effective rates and volumes for the field.			
		dimensionless	dimensionless	dimensionless	1.0
3	WELNETWK	A defined character string that determines if the WELNAME efficiency factor should be transferred to the equivalent Extended Network Model node, and should be set to either: 1) NO: The well's equivalent Extended Network Model node flow rates are not reduced by the efficiency factor FACTOR. 2) YES: The well's equivalent Extended Network Model node flow rates are reduced by the efficiency factor FACTOR This option is only applicable for the Extended Network Model, as in the Standard Network Model groups flow rates are always used in the calculation of pressure drops. Only the default value of YES is supported by OPM Flow.			YES

Notes:

1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".

Table 12.86: WEFAC Keyword Description

See also the GEFAC keyword in the SCHEDULE section to set the efficiency at the group level, as opposed to applying the efficiency to individual wells.

Note

One can also apply plant efficiencies through the GEFAC keyword in the SCHEDULE section. If all the wells in a group are flowing through a facility that has an overall efficiency factor, then it is more appropriate to apply the efficiency factor at the group level. This of course does not preclude applying additional well efficiencies to individual wells. For example, subsea wells (wet trees) may have additional down time compared to platform wells (dry trees) even though both sets of well are flowing through the same platform. Another example would be gas lift wells and wells using electrical submersible pumps, as their artificial lift mechanisms.

Example

```
--
--        WELL EFFICIENCY FACTORS
--
-- WELL   EFF     NETWK
-- NAME   FACT    OPTN
WEFAC
'GP*   '  0.950                                                    /
'OP*   '  0.862                                                    /
/
```

In the above example the all the gas wells are are defined as having a well efficiency factor (up time) of 0.950 and all the oil wells have a lower efficiency factor of 0.862.

12.3.269 WELCNTL – MODIFY WELL CONTROL AND TARGETS

Description

The WELCNTL keyword modifies a well's target control and value, both rates and pressures, for previously defined wells without having to define all the variables on the well control keywords: WCONPROD, WCONHIST, WCONINJE, or WCONINJH keywords. Variables not changed by the WELCNTL keyword remain the same as those previously entered via the well control keywords or previously entered WELCNTL keywords. Note that the well must still be initially be fully defined using the WCONPROD or WCONINJE keywords. All the aforementioned keywords are described in the SCHEDULE section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WELNAME	A character string of up to eight characters in length that defines the well name for which the well production rates and pressures data are being redefined. Note that the well name (WELNAME) must have been declared previously using the WELSPECS and WCONPROD (or WCONINJE) keywords in the SCHEDULE section, otherwise an error may occur.			None
2	TARGET	A defined character string that sets the item to be changed for the well the value of the item is set by item (3). 1) ORAT: reset the surface oil production rate value as defined by item (3). 2) WRAT: reset the surface water production rate value as defined by item (3). 3) GRAT: reset the surface gas production rate value as defined by item (3). 4) LRAT: reset the surface liquid (oil plus water) production rate value as defined by (3). 5) RESV: reset he in situ reservoir volume rate value as defined by (3). 6) BHP: reset the bottom-hole pressure value as defined by item (3). 7) THP: reset the tubing head pressure value for the well as defined by item (3). 8) VFP: reset the vertical lift performance table number as defined by (3). 9) LIFT: reset the artificial lift quantity for use with vertical lift performance tables. 10) GUID: reset the guide rate value for wells operating under group control. Note TARGET redefines the target controlled for a well and the control value on item (4). For example, if a well is operating on ORAT control, as defined by the previously entered WCONPROD keyword, entering TARGET equal to LRAT with a value, sets the TARGET to liquid rate with the given value. That is the well will be targeting a liquid rate not the previously requested oil ratel. Use the WELTARG keyword in the SCHEDULE section to change the target and constraint values for a well.			None

No.	Name	Description			Default
		Field	Metric	Laboratory	
3	VALUE	A real positive value that defines the value of the variable declared by TARGET			
	Liquid	stb/d	sm^3/day	scc/hour	
	Gas	Mscf/d	sm3/day	scc/hour	
	Res Vol	rb/d	rm^3/day	rcc/hour	
	Pressure	psia	barsa	atma	
	VFP	dimensionless	dimensionless	dimensionless	
	LIFT	same as VFPPROD or VFPINJ	same as VFPPROD or VFPINJ	same as VFPPROD or VFPINJ	None

Notes:

1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".

Table 12.87: WELCNTL Keyword Description

If a well is currently a history matching well, then WELCNTL can be used to change the well to a standard well. See also the WELTARG keyword, in the SCHEDULE section that can be used to reset a well's target and constraints of both rates and pressures.

Example

The following example below shows the oil rates for the OP01 oil producer at the start of the schedule section (January 1, 2000).

```
-- ------------------------------------------------------------------------
-- 01 JAN 2000 START OF SCHEDULE SECTION
-- ------------------------------------------------------------------------
--
--       WELL PRODUCTION WELL CONTROLS
--
-- WELL  OPEN/  CNTL   OIL    WAT    GAS    LIQ    RES    BHP    THP   VFP    VFP
-- NAME  SHUT   MODE   RATE   RATE   RATE   RATE   RATE   PRES   PRES  TABLE  ALFQ
WCONPROD
OP01     OPEN   ORAT   3000   1*     1*     1*     1*     750.0  500.  9      1* /
/
DATES
01 FEB 2000 /
/
--
--       WELL CONTROL MODE AND OPERATING TARGET
--
--  WELL WELL   TARGET
--  NAME CNTL   VALUE
WELCNTL
OP01     LRAT   5000                                        /
/
```

From January 1, 2000 to February 1, 2000 well OP01 is open and is on oil rate control and has a target oil rate of 3,000 stb/d and uses VFPPROD vertical lift table number 9 with a minimum tubing head pressure constraint of 500 psia. After February 1, 2000 the well is changed to liquid control with a target rate of 5,000 stb/d of liquid and all the other parameters remain unchanged.

12.3.270 WELDEBUG – DEFINE THE WELL DEBUG DATA TO BE PRINTED TO FILE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword defines the well debug data to be written to the debug file (*.DBG). This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.271 WELDRAW – DEFINE MAXIMUM DRAW DOWN FOR PRODUCTION WELLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, WELDRAW, defines the maximum draw down for production wells The keyword may be useful in wells that are subject to fines or sand production to limit the draw down between the sand face and the well in order to limit or avoid sand production.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.272 WELEVNT – DEFINE WELL WPWEM SUMMARY MNEMONIC OUTPUT VALUE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, WELEVNT, defines an integer value to be assigned to an individual well's WPWEM summary variable that is written to the SUMMARY file. The value is set to zero after the current time step.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.273 WELLSTRE – DEFINE INJECTION STREAM COMPOSITION

Description

The WELLSTRE keyword defines a well stream together with the compositional component mole factions, associated with the well stream, as such it should have the same number of entries as that declared via the COMPS keyword in the RUNSPEC section, and the NCOMPS keyword in the PROPS section. Once a gas well stream has been defined, it can be used with either the WINJGAS or GINJGAS keywords in the SCHEDULE section, to set the injected gas composition. Similarly, if an oil well stream has been defined by WELLSTRE, then the well stream can be used with the WINJOIL keyword in the SCHEDULE section, to specify the injected oil composition.

The keyword should only be used if the CO2STORE and GASWAT keywords in the RUNSPEC section have also be activated for the gas-water two component model.

Note

This is an OPM Flow keyword used with OPM Flow's CO2STORE and GASWAT keywords in the RUNSPEC section, and should not be confused with the more general version of the WELLSTRE keyword used in the commercial compositional simulator.

Secondly, although OPM Flow parses the keyword, the simulator currently ignores the data for this keyword.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	STREAM	STREAM is a character string of up to eight characters in length, representing the well stream name. The WELLDIMS(MXSTRMS) parameter in the RUNSPEC section determines the maximum number of well streams allowed in the model.			None
2	ZCOMP	A row vector, with each item representing a compositional component mole fraction for a given component. In addition, the sum of the compositional component mole fractions must sum to one, otherwise an error will occur. Note that the number ZCOMP values, should be the same as that entered via the NCOMPS keyword in the PROPS section, and the COMPS keyword in the RUNSPEC section. However, for a given ZCOMP component, the mole fraction may be defaulted with 1*, in which case the mole fraction is set to zero. Secondly, ZCOMP may be terminated early, in this case the undefined mole fractions will be set to zero. Finally, only the default value of two components are currently supported by OPM Flow.			
		mole fraction	mole fraction	mole fraction	None

Notes:

1) The keyword is followed by up to MXSTRMS records as declared on the WELLDIMS keyword in the RUNSPEC section.
2) Each record is terminated by a "/" and the keyword should be terminated by a "/".

Table 12.88: WELLSTRE Keyword Description

Example

The following example defines how to specify a two component formulation, together with defining the names of the composition components, to be used with the CO2STORE and GASWAT options.

```
-- ==================================================================================
--
-- PROPS SECTION
--
-- ==================================================================================
PROPS             --
--       CONFIRM NUMBER OF COMPOSITIONAL COMPONENTS (OPM FLOW KEYWORD)
--
NCOMPS
         2                                                                        /
--
--       DEFINE COMPOSITIONAL COMPONENTS NAMES (OPM FLOW KEYWORD)
--
CNAMES
         'CO2'
         'H2O'                                                                   /
```

The second part of the example, defines the well stream for the above two component CO_2 water system.

```
-- ==================================================================================
--
-- SCHEDULE SECTION
--
-- ==================================================================================
SCHEDULE
--
--       WELL STREAM INJECTION COMPOSITION (OPM FLOW Keyword)
--
-- WELL     -- WELL STREAM COMPOSITIONAL COMPONENT       --
-- STREAM   --           MOLE FRACTIONS                  --
WELLSTRE
'C02STREAM'    1.000   0.000                               /
/
```

Here the well stream consists of 100% CO_2 and zero water.

12.3.274 WELMOVEL – MOVE GLOBAL WELL INTO AN LGR

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, WELMOVEL, moves a previously defined gloabal well into a previously declared Local Grid Refinement ("LGR"), in a RESTART run. The keyword should only be used in RESTART runs.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.275 WELOPEN – DEFINE WELL AND WELL CONNECTIONS FLOWING STATUS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The WELOPEN keyword defines the status of wells and well connections, and is used to open and shut previously defined well and well connections without having to re-specify all the data on the well control keywords: WCONPROD, WCONHIST, WCONINJE, or WCONINJH keywords. Note that the well must still be initially be fully defined using the WCONPROD or WCONINJE keywords. All the aforementioned keywords are described in the SCHEDULE section

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WELNAME	A character string of up to eight characters in length that defines the well name for which the well and well connection status data are being defined. Note that the well name (WELNAME) must have been declared previously using the WELSPECS keyword in the SCHEDULE section, otherwise an error may occur.			None
2	STATUS	A character string of length four that defines the well and a well's connections' operational status, STATUS should be set to one of the following character strings: 1) OPEN: the connections are open to flow. 2) SHUT: the connections are closed to flow (shut-in). 3) AUTO: the connection are initially closed, but may be opened automatically if an economic limit is violated.			OPEN
3	I	An integer less than or equal to NX that defines the connection location in the I-direction.			1*
4	J	An integer less than or equal to NY that defines the connection location in the J-direction.			1*
5	K	An integer less than or equal to NZ that defines the connection location in the K-direction.			1*
6	K1	An integer less than or equal to NZ that defines the UPPER completion location in the K-direction. Connections are lumped into completions via the COMPLUMP keyword, and K1 refers to the first completion number, as defined by the COMPLUMP keyword, and all the connections contained within the K1 completion.			1*
7	K2	An integer less than or equal to NZ that defines the LOWER completion location in the K-direction. Connections are lumped into completions via the COMPLUMP keyword, and K2 refers to the last completion number, as defined by the COMPLUMP keyword, and all the connections contained within the K2 completion.			1*
Notes: 1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".					

Table 12.89: WELOPEN Keyword Description

If variables I, J K, K1 and K2 are all set to a negative number or defaulted with 1* then STATUS is applied to the well and the well connections remain unchanged.

If variables I, J K, K1 and K2 are all set to zero or a positive value then STATUS is applied to the defined connections and the well status remains unchanged. The defined connections are those values with the I, J, K specified locations and connections within the completion number range specified by K1 and K2.

See also the COMPDAT keyword to define a well's connections, the COMPLUMP keyword to group well connections into well completions, the WCONPROD and WCONINJE keywords to define a well's production and injections targets and constraints. All the aforementioned keywords are described in the SCHEDULE section.

Examples

The following example defines two vertical oil wells using the WELSPECS keyword and their associated connection data.

```
-- WELL SPECIFICATION DATA
--
-- WELL     GROUP      LOCATION  BHP    PHASE  DRAIN  INFLOW  SHUT  CROSS  PRESS
-- NAME     NAME        I    J   DEPTH  FLUID  AREA   EQUA.   IN    FLOW   TABLE
WELSPECS
OP01      PLATFORM     14   13   1*      OIL   1*     STD     OPEN   NO    1*  /
OP02      PLATFORM     28   96   1*      OIL   1*     STD     OPEN   NO    1*  /
/
--
--       WELL PRODUCTION WELL CONTROLS
--
-- WELL  OPEN/  CNTL   OIL    WAT    GAS   LIQ    RES    BHP    THP   VFP    VFP
-- NAME  SHUT   MODE   RATE   RATE   RATE  RATE   RATE   PRES   PRES  TABLE  ALFQ
WCONPROD
'*'      SHUT   GRUP   1*     1*     1*    1*     1*     200.0                /
/
--       WELL CONNECTION DATA
--
-- WELL  --- LOCATION ---  OPEN   SAT   CONN   WELL   KH    SKIN   D     DIR
-- NAME   II  JJ  K1  K2   SHUT   TAB   FACT   DIA    FACT  FACT   FACT  PEN
COMPDAT
OP01      1*  1*   1  10   SHUT   1*    1*    0.708  1*    0.0    1*    'Z' /
OP01      1*  1*  15  30   SHUT   1*    1*    0.708  1*    0.0    1*    'Z' /
OP01      1*  1*  35  90   SHUT   1*    1*    0.708  1*    0.0    1*    'Z' /
OP02      1*  1*   1  10   SHUT   1*    1*    0.708  1*    0.0    1*    'Z' /
/
```

```
--
--       DEFINE WELL AND WELL CONNECTIONS FLOWING STATUS
--
--  WELL WELL    --LOCATION--  COMPLETION
--  NAME STAT      I   J    K  FIRST LAST
WELOPEN
OP01     OPEN                                    /
OP01     OPEN      0   0    1                    /
OP01     OPEN      0   0    2                    /
OP01     OPEN      0   0    3                    /
OP01     OPEN      0   0    4                    /
OP02     OPEN                                    /
OP02     OPEN      0   0    0                    /
/
```

In this example the first record of each well WELOPEN keyword changes the well status from shut (as per the WCONPROD keyword) to open. Then for well OP01 well connections one to four are opened for flow and all the connections for well OP02 are opened, that is connections one to ten.

The next example shows the use of the COMPLUMP keyword to group the well connections into well completions for wells OP01, OP02 and OP03, and then use the WELOPEN keyword to open the well and the well completions.

```
--
--       ASSIGN WELL CONNECTIONS TO COMPLETIONS
--
-- WELL   --- LOCATION ---   COMPL
-- NAME    II  JJ  K1  K2    NO.
COMPLUMP
OP01       0   0   1  10     1                    / COMPLETION NO. 01
OP01       0   0  15  30     2                    / COMPLETION NO. 02
OP01       0   0  35  90     3                    / COMPLETION NO. 03
OP02       0   0  15  30     2                    / COMPLETION NO. 02
OP02       0   0  35  90     3                    / COMPLETION NO. 03
OP03       0   0   1  10     1                    / COMPLETION NO. 01
OP03       0   0  35  90     3                    / COMPLETION NO. 03
/
--
--       DEFINE WELL AND WELL CONNECTIONS FLOWING STATUS
--
--  WELL WELL    --LOCATION--  COMPLETION
--  NAME STAT      I   J    K  FIRST LAST
WELOPEN
OP01     OPEN                                    /
OP01     OPEN      0   0    0     3     3        /
OP02     OPEN                                    /
OP02     OPEN      0   0    0     2     2        /
OP03     OPEN                                    /
OP03     OPEN      0   0    0     0     0        /
/
```

Again, the first record of each well WELOPEN keyword changes the well status from shut (as per the WCONPROD keyword) to open. Then for well OP01 well completion number three is opened (connections 35 to 90), completion two (connections 15 to 30) for well OP02 and completion numbers one to three (all the connections) for well OP03. Note that there is no completion number two for OP03 so connections 15 to 30 will not be opened.

Note the last completion number for well OP03 was named completion number three, but it could have been named number two as well. The reason why it was named number three instead of two was because it was assumed (for the example) that layers 35 to 90 represent a particular reservoir, and therefore allowing for the tracking of completions for individual reservoirs, as shown in the example.

This example shows how one can open all the wells and well completions for a given reservoir.

```
--
--       DEFINE WELL AND WELL CONNECTIONS FLOWING STATUS
--
--  WELL WELL    --LOCATION--  COMPLETION
--  NAME STAT      I   J    K  FIRST LAST
WELOPEN
'*'       OPEN                                        /
'*'       OPEN      0   0    0     3     3           /
OP02      SHUT      0   0    0     0     0           /
OP02      OPEN      0   0    0     2     2           /
/
```

In this case well OP01, OP02 and OP03 are opened via completion number three – which we do not want. Hence, all the connection for OP02 are shut, and then connections associated with completion two are opened instead for this well.

12.3.276 WELOPENL – DEFINE WELL AND WELL CONNECTIONS FLOWING STATUS (LGR)

Description

The WELOPENL keyword defines the status of wells and well connection in Local Grid Refinement Grids ("LGR") and is used to open and shut previously defined well and well connections without having to re-specify all the data on the well control keywords: WCONPROD, WCONHIST, WCONINJE, or WCONINJH keywords. Note that the well must still be initially be fully defined using the WCONPROD or WCONINJE keywords. All the aforementioned keywords are described in the SCHEDULE section

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WELNAME	A character string of up to eight characters in length that defines the well name for which the well and well connection status data is being defined. Note that the well name (WELNAME) must have been declared previously using the WELSPECS keyword in the SCHEDULE section, otherwise an error may occur.			None
2	LGRNAME	A character string of up to eight characters in length that defines the LGR name for which the well LGR connection data are being defined. Note that the well name (LGRNAME) must have been declared previously using the WELSPECL keyword in the SCHEDULE section, otherwise an error may occur. If defaulted with 1* the LGR on the WELSPECL keyword will be utilized.			Defined
3	STATUS	A character string of length four that defines the well and a well's connections' operational status, STATUS should be set to one of the following character strings: 1) OPEN: the connections are open to flow. 2) SHUT: the connections are closed to flow (shut-in). 3) AUTO: the connection are initially closed, but may be opened automatically if an economic limit is violated.			OPEN
4	I	An integer less than or equal to NX that defines the connection location in the I-direction.			1*
5	J	An integer less than or equal to NY that defines the connection location in the J-direction.			1*
6	K	An integer less than or equal to NZ that defines the connection location in the K-direction.			1*
7	K1	An integer less than or equal to NZ that defines the UPPER completion location in the K-direction. Connections are lumped into completions via the COMPLUMP keyword, and K1 refers to the first completion number, as defined by the COMPLUMP keyword, and all the connections contained within the K1 completion.			1*

No.	Name	Description			Default
		Field	Metric	Laboratory	
8	K2	An integer less than or equal to NZ that defines the LOWER completion location in the K-direction. Connections are lumped into completions via the COMPLUMP keyword, and K2 refers to the last completion number, as defined by the COMPLUMP keyword, and all the connections contained within the K2 completion.			1*

Notes:

1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".

Table 12.90: WELOPENL Keyword Description

If variables I, J K, K1 and K2 are all set to a negative number or defaulted with 1* then STATUS is applied to the well and the well connection remain unchanged.

If variables I, J K, K1 and K2 are all set to zero or a positive value then STATUS is applied to the defined connections and the well status remains unchanged. The defined connections are those values with the I, J, K specified locations and connections within the completion number range specified by K1 and K2.

See also the COMPDAT keyword to define a well's connections, the COMPLUMP keyword to group well connections into well completions, the WCONPROD and WCONINJE keywords to define a well's production and injections targets and constraints. All the aforementioned keywords are described in the SCHEDULE section.

Example

The following example shows the use of the COMPLMPL keyword to group the well connections into well completions for well OP01 and then use the WELOPEN keyword to open the well and the well connections.

```
--
--        ASSIGN WELL LGR CONNECTIONS TO COMPLETIONS
--
--        WELL    LGR        ---LOCATION---    COMPL
--        NAME    NAME       II   JJ   K1   K2   NO.
COMPLMPL
          OP01    LGR1       26   58    1    3    1              /
          OP01    LGR1       26   58    4   10    2              /
          OP01    LGR1       26   58   11   12    3              /
/
```

```
--
--       WELL PRODUCTION STATUS FOR LGR WELLS
--
--  WELL    LGR      WELL    --LOCATION--  COMPLETION
--  NAME    NAME     STAT      I   J    K  FIRST LAST
WELOPENL
OP01       LGR1     OPEN                                    /
OP01       LGR1     OPEN      0   0    0     1     2       /
OP01       LGR2     SHUT      0   0    0     3     3       /
/
```

The first record of the WELOPENL keyword changes the well status from shut (as per the WCONPROD keyword) to open, in case it has been shut-in. Then well completion number one and two are opened (connections 1 to 10), and completion number three shut-in (connections 11 to 12).

12.3.277 WELPI – DEFINE WELL PRODUCTIVITY AND INJECTIVITY INDICES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The WELPI keyword is used to define a well's productivity or injectivity index at the time the keyword is activated. Productivity and injectivity indices are a function of both bottom-hole pressure and mobility and thus will vary in time as the bottom-hole pressure and fluids produce by the well are changing. Thus, the values enter on this keyword for a given well will override any previously calculated values, or values previously entered, using this keyword. This keyword should only be invoked after a well's connection factors have been declared via the COMPDAT keyword in the SCHEDULE section.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WELNAME	A character string of up to eight characters in length that defines the well name for which the well productivity index is being defined. Note that the well name (WELNAME) must have been declared previously using the WELSPECS keyword in the SCHEDULE section, otherwise an error may occur.			None
2	WELPI	A real positive value that defines the productivity or injectivity of a well at the time the keyword is activated in the input deck. Subsequent use of the keyword for a given well will override all previously entered data, including the original productivity index calculated from the well's COMPDAT connection data. Note that: 1) Only the connections currently opened will have their connection factors re-scaled to honor the productivity and injectivity index. Connections not opened will remain unchanged. 2) Additional, previously defined connections added to the well at a later time, will not be influenced by a previous WELPI keyword. It may therefore be necessary to add another WELPI keyword at the time the new connections are opened, if desired. 3) However, re-defining connections via the COMPDAT keyword resets the calculated productivity and injectivity index for those re-defined connections in the well. Subsequent usage of the keyword for the same well may lead to unintended values; use the WPI series of variables in the SUMMARY section to output and verify that the values are consistent with the desired results.			
		Liquid stb/d/psia Gas Mscf/d/psia	Liquid sm^3/day/bars Gas sm3/day/bars	Liquid scc/hour/atm Gas scc/hour/atm	1.0
Notes: 1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".					

Table 12.91: WELPI Keyword Description

Note that only a well's currently opened connection factors, as entered via the COMPDAT keyword, are re-scaled to match the required productivity and injectivity index. See section *Well Productivity* on well productivity for further information on well productivity.

Note

One should not modify the productivity and injectivity index of gas wells that use:

1) Russell Goodrich[330] pressure square inflow equation,
2) the general dry gas pseudo pressure inflow equation, or
3) the generalized gas pseudo pressure inflow equation, as declared via the INFLOW property on the WELSPECS keyword in the SCHEDULE section, or
4) those gas wells that use the non-Darcy D factor coefficient, as declared by the DFACT property on the COMPDAT and WDFAC keywords in the SCHEDULE section.

See also the WPIMULT keyword in the SCHEDULE section to set the productivity and injectivity index for a well by scaling the current index by a scaling factor.

Example

```
--
--       DEFINE WELL PRODUCTIVITY/INJECTIVITY INDEX
--
-- WELL  PI
-- NAME  MULT
WELPI
OP01     1.250                                          /
OP02     2.750                                          /
OP03     1.100                                          /
/
```

In the above example the oil wells are are defined as having a well productivity indices of 1.250, 2.750 and 1.100 for the OP01, OP02 and OP03 wells, respectively.

[330] *Russell, D.G., Goodrich, J.H., Perry, G.E and Bruskotter, J.F "Methods of Predicting Gas Well Performance", Transactions of the ASME, Journal of Petroleum Technology (1966) 99-108.*

12.3.278 WELPRI – ASSIGN WELL PRIORITY

Description

The WELPRI keyword is used to re-assign a priority number to a well for when the PRIORITY keyword has been used in the SCHEDULE section. The PRIORITY keyword activates the Well Priority option and defines the coefficients in the well priority equation; WELPRI keyword can be used to over write these calculated priority numbers

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.279 WELSEGS – DEFINE MULTI-SEGMENT WELLS AND THEIR SEGMENT STRUCTURE

Description

The WELSEGS keyword defines a well to be a multi-segment well and defines the well's segment structure. Note that the well must have been previously define by the WELSPECS keyword in the SCHEDULE section and that the WELSEGS keyword should be repeated for each multi-segment well in the model.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1-1	WELNAME	A character string of up to eight characters in length that defines the well name for which a multi-segment well is being defined. Note that the well name (WELNAME) must have been declared previously using the WELSPECS keyword in the SCHEDULE section, otherwise an error may occur.			None
1-2	BHPREF	A real value that must be entered that defines the reference depth for reporting the bottom-hole pressure for the multi-segment well. Ideally this value should be set to the nodal point of the top segment. If the keyword is entered multiple times for the same well, due to for example the well configuration changing through time, then it is only necessary to enter this data the first time the keyword is used for a well.			
		feet	m	cm	None
1-3	TUBDZ	TUBDZ is a real positive value that defines the length of the tubing from the tubing head or wellhead at the surface to the nodal point of the top segment (BHPREF). Tubing pressures from BHREF up to the tubing length of TUBDZ to the surface are not calculated by the multi-segment well option as these are taken into account by the VFP tables allocated to well and entered via the VFPROD and VPINJ keywords in the SCHEDULE section. If TUBDZ is set to zero or defaulted then the tubing length is measured from the nodal point of the top segment, that is BHPREF.			
		feet	m	cm	0
1-4	WBORVOL	WBORVOL is a real positive value that defines the effective wellbore volume for the top segment, that is from the tubing head or wellhead at the surface to the nodal point of the top segment (BHPREF). The default value of 1.0×10^{-5} results in minimal wellbore storage.			
		ft3	m3	cm3	1.0E-5
1-5	TUBOPT	TUBOPT is a character string that defines the type of length and depth data entered for DEPTH1 and DEPTH2 on the second record and should be set to one of the following: 1) INC: Incremental values, that is the length of each segment. 2) ABS: Absolute values, that is the depth of each segment. There is no default value for TUBOPT one of the above options must be explicitly defined.			None

No.	Name	Description			Default
		Field	Metric	Laboratory	
I-6	PRESOPT	PRESOPT is a character string that defines the pressure drop calculation used for each well segment and should be set to one of the following: 1) HFA: Sets the pressure calculation to include the hydrostatic, friction and acceleration terms. 2) HF-: Sets the pressure calculation to include the hydrostatic and friction terms only. 3) H--: Sets the pressure calculation to include the hydrostatic pressure drop term only. The default value for PRESOPT of HFA sets the pressure calculation to include the hydrostatic, friction and acceleration terms.			HFA
I-7	FLOWOPT	FLOWOPT is a character string that defines the type of multi-phase calculation used for each well segment and should be set to one of the following: 1) HO: Sets the multi-phase calculation to the homogeneous model, that is all phases flow at the same velocity. 2) DF-: Sets the multi-phase calculation to the Drift Flux Slip model. OPM Flow only supports the default value of HO.			HO
I-8	XCORD	A real positive values equal to or greater than zero that defines the coordinate in the x-direction of the nodal point of the top segment that is used for display purposes only. Currently this option is not supported by OPM Flow.			
		feet	m	cm	None
I-9	YCORD	A real positive value equal to or greater than zero that defines the coordinate in the y-direction of the nodal point of the top segment that is used for display purposes only. Currently this option is not supported by OPM Flow.			
		feet	m	cm	None
I-10	XAREATH	XAREATH is real positive value equal to or greater than zero that defines the cross sectional area of the pipe wall used in thermal conductivity calculations for when the temperature calculation is activated by the TEMP keyword in the RUNSPEC section. Currently this option is not supported by OPM Flow.			
		ft^2	m^2	cm^2	None
I-11	VHEATCAP	VHEATCAP is real positive value equal to or greater than zero that defines the volumetric heat capacity of the pipe wall used in thermal conductivity calculations for when the temperature calculation is activated by the TEMP keyword in the RUNSPEC section. Currently this option is not supported by OPM Flow.			
		Btu/ft/day/°R	kJ/m/day/K	J/cm/hr/K	None

No.	Name	Description			Default
		Field	Metric	Laboratory	
1-12	THCON	THCON is real positive value equal to or greater than zero that defines the thermal conductivity of the pipe wall used in thermal conductivity calculations for when the temperature calculation is activated by the TEMP keyword in the RUNSPEC section. Currently this option is not supported by OPM Flow.			
		Btu/ft/day/°R	kJ/m/day/K	J/cm/hr/K	None
1-13	/	Record terminated by a "/"			Not Applicable
2-1	ISEG1	A positive integer greater than or equal to two and less than or equal to MXSEGS on WSEGDIMS keyword in the RUNSPEC section that defines the start of a segment			None
2-2	ISEG2	A positive integer greater than or equal to two and less than or equal to ISEG1 on this record and MXSEGS on the WSEGDIMS keyword in the RUNSPEC section that defines the end of a segment.			None
2-3	IBRANCH	A positive integer greater than or equal to one and less than or equal to MXBRAN on WSEGDIMS keyword in the RUNSPEC section that defines the branch number of a segment. All segments on the main stem must have IBRANCH set to one and lateral branches should have values between two and MXSEGS on the WSEGDIMS keyword in the RUNSPEC section.			None
2-4	ISEG3	A positive integer greater than or equal to two and less than or equal to MXSEGS on the WSEGDIMS keyword in the RUNSPEC section that defines the outlet segment.			None
2-5	DEPTH1	DEPTH1 is a real positive value that: 1) If TUBOPT is set to INC then DEPTH1 is the total length of the tubing for this segment. 2) If TUBOPT is set to ABS then DEPTH is the length of the tubing from the tubing head or wellhead at the surface to the last segment in the range.			
		feet	m	cm	None
2-6	DEPTH2	DEPTH2 is a real positive value that: 1) If TUBOPT is set to INC then DEPTH2 is the total incremental depth change of the tubing for this segment. 2) If TUBOPT is set to ABS then DEPTH defines the depth of the tubing at the last nodal point of this segment. in this range.			
		feet	m	cm	None
2-7	ID	A real positive value that defines the tubing internal diameter of the segment for the well.			
		feet	m	cm	None

No.	Name	Description			Default
		Field	Metric	Laboratory	
2-8	EPSILON	A real positive value that defines the tubing absolute roughness of the segment for the well.			
		feet	m	cm	None
2-9	XAREA	XAREA is real positive value equal to or greater than zero that defines the cross sectional area for fluid flow. Currently this option is not supported by OPM Flow.			
		ft^2	m^2	cm^2	None
2-10	VOLSEG	VOLSEG is a real positive value that defines the effective segment volume for the this segment. Currently this option is not supported by OPM Flow.			
		ft3	m3	cm3	None
2-11	XCORDS	A real positive values equal to or greater than zero that defines the coordinate in the x-direction of the nodal point of this segment that is used for display purposes only. Currently this option is not supported by OPM Flow.			
		feet	m	cm	None
12-2	YCORDS	A real positive value equal to or greater than zero that defines the coordinate in the y-direction of the nodal point of this top segment that is used for display purposes only. Currently this option is not supported by OPM Flow.			
		feet	m	cm	None
2-13	XAREAS	XAREAS is real positive value equal to or greater than zero that defines the cross sectional area of the pipe wall for this segment, that is used in thermal conductivity calculations for when the temperature calculation is activated by the TEMP keyword in the RUNSPEC section. Currently this option is not supported by OPM Flow.			
		ft^2	m^2	cm^2	None
2-14	VHEATSEG	VHEATCAP is real positive value equal to or greater than zero that defines the volumetric heat capacity of the pipe wall for this segment, that is used in thermal conductivity calculations for when the temperature calculation is activated by the TEMP keyword in the RUNSPEC section. Currently this option is not supported by OPM Flow.			
		Btu/ft/day/°R	kJ/m/day/K	J/cm/hr/K	None
2.15	THCSEG	THCON is real positive value equal to or greater than zero that defines the thermal conductivity of the pipe wall for this segment, that is used in thermal conductivity calculations for when the temperature calculation is activated by the TEMP keyword in the RUNSPEC section. Currently this option is not supported by OPM Flow.			
		Btu/ft/day/°R	kJ/m/day/K	J/cm/hr/K	None

No.	Name	Description			Default
		Field	Metric	Laboratory	
2-16	/	Record terminated by a "/"			Not Applicable

Notes:

1) Each multi-segment wells must be defined by a separate WELSEGS keyword that consists of two records, with entries 1-1 to 1-12 representing record one items and 2-1 to 2-15 representing record number two items in the "No." column in this table.
2) Record number two of the keyword, items 2-1 to 2-15, is followed by up to MXCONS records as declared on the WSEGDIMS keyword in the RUNSPEC section.
3) Each of the records are terminated by a "/" and is explicitly shown in the above rows and the keyword should be terminated by a "/".

Table 12.92: WELSEGS Keyword Description

The total number of wells should be defined via the WELLSDIMS keyword and the number of multi-segment wells should be declared on the WSEGDIMS keyword, both keywords are in the RUNSPEC section.

See also the WELSPECS keyword to define wells, the COMPDAT keyword to define the well completions for both ordinary wells and multi-segment wells, and the COMPSEGS keyword to define a multi-segment well segment completions. All the aforementioned keywords are described in the SCHEDULE section.

Example

The following example defines one producing well segment oil well (OP01) using the WELSPECS, WELSEGS COMPDAT and COMPSEGS keywords, and one water injection well (WI01) using the WELSPECS and COMPDAT keywords.

```
--
--       WELL SPECIFICATION DATA
--
-- WELL  GROUP     LOCATION  BHP    PHASE  DRAIN  INFLOW  OPEN  CROSS  PVT
-- NAME  NAME        I    J  DEPTH  FLUID  AREA   EQUANS  SHUT  FLOW   TABLE
WELSPECS
OP01     PLATFORM   10   10   1*     OIL                                    /
WI01     PLATFORM    1    1   1*     WATER                                  /
/
--
--       WELL CONNECTION DATA
--
-- WELL  --- LOCATION ---  OPEN   SAT   CONN   WELL   KH    SKIN   D     DIR
-- NAME   II  JJ  K1  K2   SHUT   TAB   FACT   DIA    FACT  FACT   FACT  PEN
COMPDAT
OP01      10  10   1   1   OPEN   1*    200.   0.5                          /
OP01      10  10   2   2   OPEN   1*    200.   0.5                          /
OP01      10  10   3   3   OPEN   1*    200.   0.4                          /
OP01      10  10   4   4   OPEN   1*    200.   0.4                          /
OP01      10  10   5   5   OPEN   1*    200.   0.4                          /
OP01      10  10   6   6   OPEN   1*    200.   0.4                          /

OP01       9  10   2   2   OPEN   1*    200.   0.4                          /
OP01       8  10   2   2   OPEN   1*    200.   0.4                          /
OP01       7  10   2   2   OPEN   1*    200.   0.4                          /
OP01       6  10   2   2   OPEN   1*    200.   0.4                          /
OP01       5  10   2   2   OPEN   1*    200.   0.4                          /
```

```
OP01      10   9   3   3   OPEN   1*    200.   0.4                          /
OP01      10   8   3   3   OPEN   1*    200.   0.4                          /
OP01      10   7   3   3   OPEN   1*    200.   0.4                          /
OP01      10   6   3   3   OPEN   1*    200.   0.4                          /
OP01      10   5   3   3   OPEN   1*    200.   0.4                          /

OP01       9  10   5   5   OPEN   1*    200.   0.4                          /
OP01       8  10   5   5   OPEN   1*    200.   0.4                          /
OP01       7  10   5   5   OPEN   1*    200.   0.4                          /
OP01       6  10   5   5   OPEN   1*    200.   0.4                          /
OP01       5  10   5   5   OPEN   1*    200.   0.4                          /

OP01      10   9   6   6   OPEN   1*    200.   0.4                          /
OP01      10   8   6   6   OPEN   1*    200.   0.4                          /
OP01      10   7   6   6   OPEN   1*    200.   0.4                          /
OP01      10   6   6   6   OPEN   1*    200.   0.4                          /
OP01      10   5   6   6   OPEN   1*    200.   0.4                          /

WI01       1   1   7   9   OPEN   1*    200.   0.5                          /
/
--
--       WELL SEGMENT SPECIFICATION DATA
--
-- WELL  NODAL       LEN     WELL    DEPH    PRESS   FLOW
-- NAME  DEPTH       TUBING  VOLM    OPTN    CALC    MODEL
WELSEGS
OP01     2512.5     2512.5  1.0E-5  ABS     HFA     HO                       /
--
--       SEG   SEG   BRAN   SEG     TUBING  NODAL   TUBE   TUBE     XSEC  VOL
--       ISTR  IEND  NO     NO      LENGTH  DEPTH   ID     ROUGH    AREA  SEG
           2     2    1      1      2537.5  2534.5  0.3    0.00010          /
           3     3    1      2      2562.5  2560.5  0.3    0.00010          /
           4     4    1      3      2587.5  2593.5  0.3    0.00010          /
           5     5    1      4      2612.5  2614.5  0.3    0.00010          /
           6     6    1      5      2637.5  2635.5  0.3    0.00010          /

           7     7    2      2      2737.5  2538.5  0.2    0.00010          /
           8     8    2      7      2937.5  2537.5  0.2    0.00010          /
           9     9    2      8      3137.5  2539.5  0.2    0.00010          /
          10    10    2      9      3337.5  2535.5  0.2    0.00010          /
          11    11    2     10      3537.5  2536.5  0.2    0.00010          /

          12    12    3      3      2762.5  2563.5  0.2    0.00010          /
          13    13    3     12      2962.5  2562.5  0.1    0.00010          /
          14    14    3     13      3162.5  2562.5  0.1    0.00010          /
          15    15    3     14      3362.5  2564.5  0.1    0.00010          /
          16    16    3     15      3562.5  2562.5  0.1    0.00010          /

          17    17    4      5      2812.5  2613.5  0.2    0.00010          /
          18    18    4     17      3012.5  2612.5  0.1    0.00010          /
          19    19    4     18      3212.5  2612.5  0.1    0.00010          /
          20    20    4     19      3412.5  2612.5  0.1    0.00010          /
          21    21    4     20      3612.5  2613.5  0.1    0.00010          /

          22    22    5      6      2837.5  2634.5  0.2    0.00010          /
          23    23    5     22      3037.5  2637.5  0.2    0.00010          /
          24    24    5     23      3237.5  2638.5  0.2    0.00010          /
          25    25    5     24      3437.5  2639.5  0.1    0.00010          /
          26    26    5     25      3637.5  2639.5  0.1    0.00010          /
/
--
--       COMPLETION SEGMENT SPECIFICATION DATA
--
```

```
-- WELL
-- NAME
COMPSEGS
OP01                                                                        /
--
--       --LOCATION--   BRAN   TUBING   NODAL    DIR   LOC     MID     COMP     ISEG
--        II   JJ   K1    NO     LENGTH   DEPTH    PEN   I,J,K   PERFS   LENGTH   NO.
          10   10    1     1     2512.5   2525.0                                  /
          10   10    2     1     2525.0   2550.0                                  /
          10   10    3     1     2550.0   2575.0                                  /
          10   10    4     1     2575.0   2600.0                                  /
          10   10    5     1     2600.0   2625.0                                  /
          10   10    6     1     2625.0   2650.0                                  /

           9   10    2     2     2637.5   2837.5                                  /
           8   10    2     2     2837.5   3037.5                                  /
           7   10    2     2     3037.5   3237.5                                  /
           6   10    2     2     3237.5   3437.5                                  /
           5   10    2     2     3437.5   3637.5                                  /

          10    9    3     3     2662.5   2862.5                                  /
          10    8    3     3     2862.5   3062.5                                  /
          10    7    3     3     3062.5   3262.5                                  /
          10    6    3     3     3262.5   3462.5                                  /
          10    5    3     3     3462.5   3662.5                                  /

           9   10    5     4     2712.5   2912.5                                  /
           8   10    5     4     2912.5   3112.5                                  /
           7   10    5     4     3112.5   3312.5                                  /
           6   10    5     4     3312.5   3512.5                                  /
           5   10    5     4     3512.5   3712.5                                  /

          10    9    6     5     2737.5   2937.5                                  /
          10    8    6     5     2937.5   3137.5                                  /
          10    7    6     5     3137.5   3337.5                                  /
          10    6    6     5     3337.5   3537.5                                  /
          10    5    6     5     3537.5   3737.5                                  /
/
```

Note the use of both the COMPDAT and COMPSEGS keywords to fully define a multi-segment well's completion.

Finally, Figure 12.15 depicts the resulting well configuration for both wells, with the conventional water injection well shown in blue and the multi-segment oil producer shown in green.

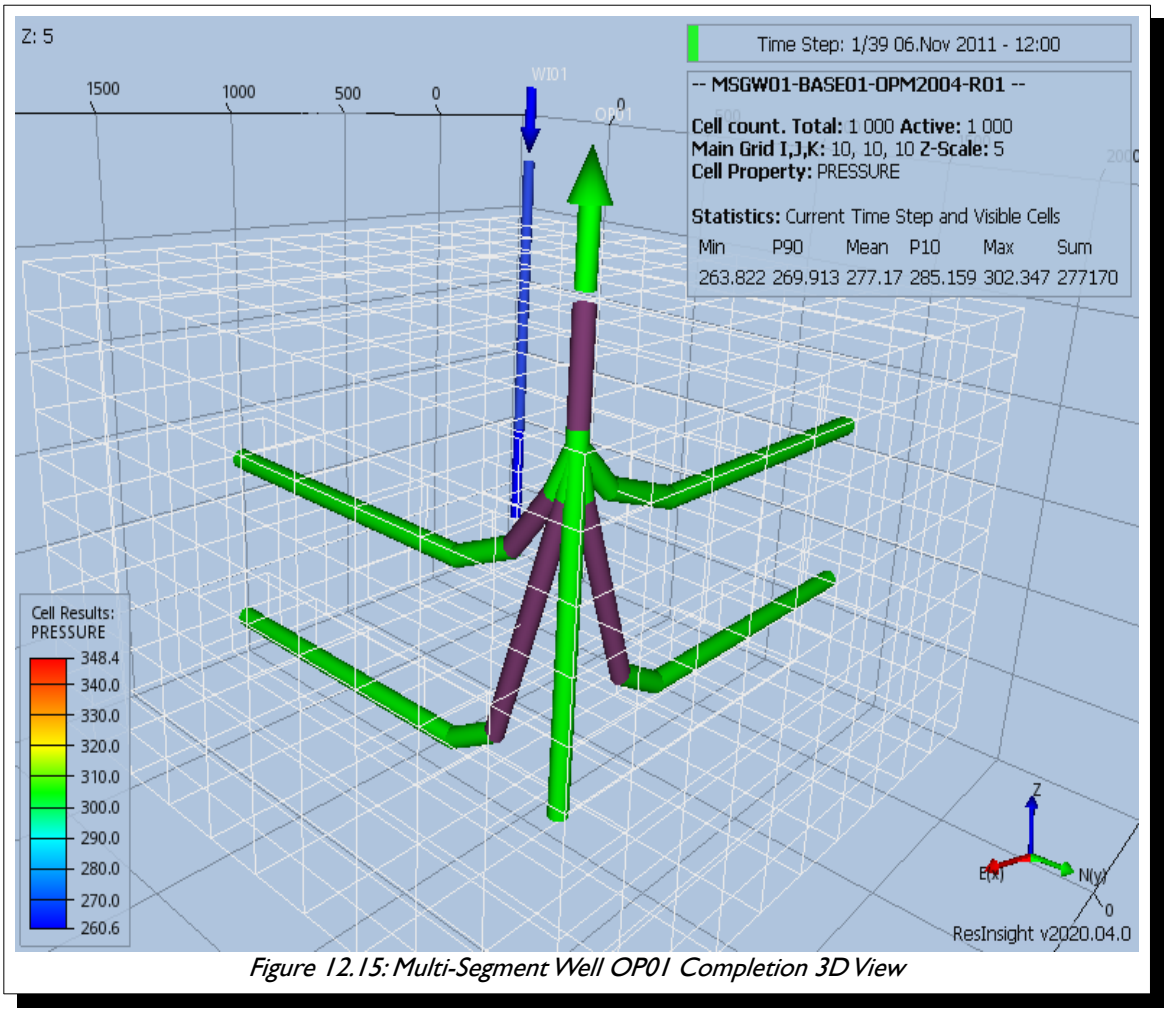


Figure 12.15: Multi-Segment Well OP01 Completion 3D View

12.3.280 WELSOMIN – DEFINE WELL CONNECTION MINIMUM OIL SATURATION FOR OPENING

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

WELSOMIN defines a minimum oil saturation for a well connection above which the connection will be opened automatically. If the grid block connection is below WELSOMIN then connection will not be automatically opened. Automatic opening of connection is controlled by the STATUS parameter on the COMPDAT keyword in the SCHEDULE section. Note that if the COMPLUMP keyword in the SCHEDULE section has been used to lump connections into completions then WELSOMIN is compared to the average oil saturation of the completion.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.281 WELSPECL – DEFINE WELL SPECIFICATIONS FOR LOCAL GRID REFINEMENTS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The WELSPECL keyword defines the general well specification data for all well types and must be used for all wells contained within a Local Grid Refinement ("LGR") instead of the WELSPECS keyword. WELSPECL must declare wells first before any other LGR well specification keywords are used in the input file. The keyword declares the name of well, the group the well belongs to, the LGR the well is incorporated into, the wellhead location and other key parameters.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WELNAME	A character string of up to eight characters in length that defines the well name for which the well specification data is being defined.			None
2	GRPNAME	A character string of up to eight characters in length that defines the group name for which the well is assigned to. The group named FIELD is the top most group and thus GRPNAME cannot be set to FIELD, although this is allowed in the commercial compositional simulator but not the commercial black-oil simulator. Note that the group hierarchy should be defined by the GRUPTREE keyword when there is more than one level of groups, otherwise all the groups will sit directly under the FIELD group in the group tree hierarchy. Secondly, groups defined by the GRUPTREE keyword cannot contain other groups and wells; that is, groups must either contain other groups or wells but not both. If necessary, wells can be re-allocated to a different group by re-entering a well's WELSPECS data together with a new value for GRPNAME.			None
3	LGRNAME	A character string of up to eight characters in length that defines the name of the local grid refinement for which the well is assigned to.			None
4	I	A positive integer greater than or equal to zero and less than or equal to NX on the CARFIN keyword for Cartesian grids, that defines the wellhead location for a vertical or deviated well, or the heel for a horizontal well in the I-direction within the LGR. For radial LGRs this parameter should be set to one.			None
5	J	A positive integer greater than or equal to zero and less than or equal to NY on the CARFIN keyword for Cartesian grids, that defines the wellhead location for a vertical or deviated well, or the heel for a horizontal well in the J-direction within the LGR. For radial LGRs this parameter should be set to one.			None

No.	Name	Description			Default
		Field	Metric	Laboratory	
6	BHPREF	A real value that defines the reference depth for reporting the bottom-hole pressure for the well. Ideally this value should be set to the midpoint of the perforations as defined by the COMPDATL keyword in the SCHEDULE section. If defaulted by 1* or set to a value less than or equal to zero, then the mid-point of shallowest connection defined by the COMPDATL keyword will be used.			Mid-point of shallowest connection defined by the COMPDATL keyword
		feet	m	cm	
7	TYPE	A defined character that defines the "main" phase for the well, and should be set to one of the following character strings: 1) GAS: for a gas well. 2) OIL: for an oil well. 3) WAT: for a water injection well. 4) LIQ: for an oil well when the liquid productivity index is required for the well. This parameter defines the phase used to calculate a well's productivity or injectivity index and the type of well, or a well's connection, to close when a group's production constraints, as defined on the GCONPROD keyword in the SCHEDULE section, have been violated. For example, if the well is declared as an oil well, then excessive gas and water connections will be subject to closure.			None
8	DRADIUS	A real value that defines the well drainage radius for the well used to calculate a well's productivity or injectivity index. A default of zero results in the pressure equivalent radius of the grid blocks containing the well connections are used.			
		feet	m	cm	0.0

No.	Name	Description			Default
		Field	Metric	Laboratory	
9	INFLOW	A defined character string that defines the inflow equation to be used for the well in calculating the well's flow rates. INFLOW should be set to one of the following character strings: 1) STD: the standard inflow equation will be used. This is normally used for wells that are primary oil or water wells. 2) NO: an alias for STD. 3) R-G: the Russell Goodrich[331] pressure square inflow equation will used. This option can be used for dry gas wells. 4) YES: an alias for R-G. 5) P-P: the general dry gas pseudo pressure inflow equation will be used. Normally used for dry gas wells. 6) GPP: the generalized gas pseudo pressure inflow equation used with wet gas wells, that is condensate gas wells. This inflow equation is based on the formulation of Whitson et al.[332] For oil and water wells the INFLOW should be set to STD, why for dry gas wells INFLOW can be set to either R-G or P-P; however, the P-P option is preferred for dry gas wells due to the more rigorous treatment of gas flow. For wet gas wells, that is gas condensate wells, INFLOW should be set to GPP. Only INFLOW equal to STD and NO are currently implemented in OPM Flow.			STD
10	AUTO	A defined character string that defines the automatic action to be taken if the economic WCUT, GOR, or WGR limits are violated and the well is to cease production. AUTO should be set to one of the following character strings: 1) STOP: the well is "stopped" at the surface and will not produce any fluids to surface; however, if there any open connections then flow may occur within the wellbore and between the open connections depending on a connection's potential with respect to all the other connections. Inter-connection flow (cross flow) can be prevented by setting the XFLOW variable to NO. In this case the well's behavior will be similar to the SHUT option described below. 2) SHUT: the well is shut at the surface and downhole, this results in no flow at the surface and no cross flow downhole. The corrective action takes places at the end of the time step in which the constraint is violated.			SHUT

[331] *Russell, D.G., Goodrich, J.H., Perry, G.E and Bruskotter, J.F "Methods of Predicting Gas Well Performance", Transactions of the ASME, Journal of Petroleum Technology (1966) 99-108.*

[332] *Whitson, C. H. and Fevang, Ø. "Generalised Pseudopressure Well Treatment in Reservoir Simulation," Presented at the IBC Technical Services Conference on Optimisation of Gas Condensate Fields, Aberdeen, UK (June 26-27, 1997).*

No.	Name	Description			Default
		Field	Metric	Laboratory	
11	XFLOW	A defined character string that defines the if cross flow should occur within the wellbore, and should be set to either: 1) YES: to allow cross flow in the wellbore through well connections. 2) NO: to disallow cross flow within the wellbore, even if the flow potentials in the well connections would allow such flow to occur. In some cases numerical issues can occur if this variable is set to YES, and resetting it to NO may resolve the issue; however the results may not represent the physical process in this case.			YES
12	PVTNUM	A positive integer greater than or equal to zero that defines the PVT table used to calculate the wellbore fluid properties that define the relationship between reservoir and surface volume rates. The default value of zero sets PVTNUM to be the PVT table of the deepest connection in the well.			0
13	DENOPT	A defined character string that sets the type of density calculation used in calculating the wellbore hydrostatic head, and should be set to one of the following character strings: 1) SEG: sets the hydrostatic head density calculation to segmented. In this cases the density is calculated between neighboring well connections and the volumes flowing from the connections. This is the more accurate calculation if the fluid properties flowing from the well connections are variable. The density calculation itself is explicit, i.e. uses the flowing volumes of the last time step. 2) AVG: sets the hydrostatic head density calculation to the average density calculation. Here the density is considered uniform across a given reservoir and is dependent on total inflow rates of each phase and the well's bottom-hole pressure The default option of 1* invokes the SEG option and is the only option implemented in OPM Flow.			SEG
14	FIPNUM	An integer value defines the FIPNUM region used to determine the reservoir conditions in calculating the well's reservoir volumes. If set to a negative integer value then the FIPNUM region of the deepest connection in the well will be used. If set to zero, the default value, then the average properties for the field will be used. If set to an integer value greater than zero, then the FIPNUM indicated by this value will be used.			0
15		Not used.			
16		Not used.			
17		Not used.			
18		Not used.			

No.	Name	Description			Default
		Field	Metric	Laboratory	
Notes: 1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/". 2) Note that the total number of individual wells declared by the WELSPECS cannot exceed the valued entered via the MXWELS variable on the WELLDIMS keyword in the RUNSPEC section.					

Table 12.93: WELSPECL Keyword Description

See also the COMPDATL keyword to define a well's connections in a LGR, the WCONPROD and WCONINJE keywords to define a well's production and injections targets and constraints. All the aforementioned keywords are described in the SCHEDULE section.

Example

The following example defines three wells using the WELSPECL keyword

```
--
-- WELL SPECIFICATION DATA FOR LGR WELLS
--
-- WELL   GROUP    LGR    LOCATION  BHP    PHASE  DRAIN  INFLOW  SHUT  CROSS  PVT
-- NAME   NAME     NAME     I    J   DEPTH  FLUID  AREA   EQUA.   IN    FLOW  TABLE
WELSPECL
GI01      PLATFORM LGR01  14   13   1*     GAS    1*     P-P     SHUT  NO     1* /
GP01      PLATFORM LGR01  64   80   1*     GAS    1*     GPP     SHUT  NO     1* /
OP01      PLATFORM LGR02  24   10   1*     OIL    1*     STD     SHUT  NO     1* /
/
```

Here, well GI01 and GP01 are in the same LGR named LGR01 and OP01 is in a separate LGR named LGR02. GI01 is a dry gas injection well that uses the dry gas pseudo inflow equation, GP01 is a gas condensate well that uses the generalized gas pseudo pressure inflow equation, and finally, OP01 is an oil well that uses the standard inflow equation. All wells: will be shut if they are required to cease production, all wells disallow cross flow, and the hydrostatic head calculation is defaulted to the segment option for all wells.

12.3.282 WELSPECS – DEFINE WELL SPECIFICATIONS

Description

The WELSPECS keyword defines the general well specification data for all well types, and must be used for all wells before any other well specification keywords are used in the input file. The keyword declares the well, the name of the well, the group the well initial belongs to, the wellhead location and other key parameters.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WELNAME	A character string of up to eight characters in length that defines the well name for which the well specification data are being defined.			None
2	GRPNAME	A character string of up to eight characters in length that defines the group name for which the well is assigned to. The group named FIELD is the top most group. GRPNAME can be set to FIELD although this is discouraged and a warning message will be issued. This is allowed in the commercial compositional simulator but not the commercial black-oil simulator. The FIELD group cannot contain both groups and wells. Note that the group hierarchy should be defined by the GRUPTREE keyword when there is more than one level of groups, otherwise all the groups will sit directly under the FIELD group in the group tree hierarchy. Secondly, groups defined by the GRUPTREE keyword cannot contain other groups and wells; that is, groups must either contain other groups or wells but not both. If necessary, wells can be re-allocated to a different group by re-entering a well's WELSPECS data together with a new value for GRPNAME.			None
3	I	A positive integer greater than or equal to zero and less than or equal to NX that defines the wellhead location for a vertical or deviated well, or the heel for a horizontal well in the I-direction. For wells being specified with the COMPTRAJ and WELTRAJ keywords in SCHEDULE section, that allow for an alternative manner to define the well connections to the simulation grid blocks, this parameter should be defaulted with 1*. Since the simulator will calculate the wellhead location from the trajectory data on the WELTRAJ keyword. Note that the COMPTRAJ and WELTRAJ keywords are OPM Flow specific keywords, and will cause an error in the commercial simulator.			None
4	J	A positive integer greater than or equal to zero and less than or equal to NY that defines the wellhead location for a vertical or deviated well, or the heel for a horizontal well in the J-direction. For wells being specified with the COMPTRAJ and WELTRAJ keywords in SCHEDULE section, that allows for an alternative manner to define the well connections to the simulation grid blocks, this parameter should be defaulted with 1*. Since the simulator will calculate the wellhead location from the trajectory data on the WELTRAJ keyword. Note that the COMPTRAJ and WELTRAJ keywords are OPM Flow specific keywords, and will cause an error in the commercial simulator.			None

No.	Name	Description			Default
		Field	Metric	Laboratory	
5	BHPREF	A real value that defines the reference depth for reporting the bottom-hole pressure for the well. Ideally this value should be set to the midpoint of the perforations as defined by the COMPDAT keyword in the SCHEDULE section. If defaulted by 1* or set to a value less than or equal to zero, then the mid-point of shallowest connection defined by the COMPDAT keyword will be used.			Mid-point of shallowest connection defined by the COMPDAT keyword
		feet	m	cm	
6	TYPE	A defined character string that defines the "main" phase for the well, and should be set to one of the following character strings: 1) GAS: for a gas well. 2) OIL: for an oil well. 3) WAT: for a water injection well. 4) LIQ: for an oil well when the liquid productivity index is required for the well. This parameter defines the phase used to calculate a well's productivity or injectivity index and the type of well, or a well's connection, to close when a group's production constraints, as defined on the GCONPROD keyword in the SCHEDULE section, have been violated. For example, if the well is declared as an oil well, then excessive gas and water connections will be subject to closure. Note OPM Flow only currently supports options one to three, that is option four (LIQ) is not supported. For producing wells this mostly matters if one plots the WPI summary vector (productivity index for well's preferred phase). In the current treatment WPI will not have contributions from the water phase if the declared preferred phase is LIQ. For injecting wells WELSPECS's preferred phase does not matter, since the preferred phase is (typically) reset to the injected phase via the WCONINJE and WCONINJH keywords.			None
7	DRADIUS	A real value that defines the well drainage radius for the well used to calculate a well's productivity or injectivity index. A default of zero results in the pressure equivalent radius of the grid blocks containing the well connections are used.			
		feet	m	cm	0.0

No.	Name	Description			Default
		Field	Metric	Laboratory	
8	INFLOW	A defined character string that defines the inflow equation to be used for the well in calculating the well's flow rates. INFLOW should be set to one of the following character strings: 1) STD: the standard inflow equation will be used. This is normally used for wells that are primary oil or water wells. 2) NO: an alias for STD. 3) R-G: the Russell Goodrich[333] pressure square inflow equation will used. This option can be used for dry gas wells. 4) YES: an alias for R-G. 5) P-P: the general dry gas pseudo pressure inflow equation will be used. Normally used for dry gas wells. 6) GPP: the generalized gas pseudo pressure inflow equation used with wet gas wells, that is condensate gas wells. This inflow equation is based on the formulation of Whitson et al.[334] For oil and water wells the INFLOW should be set to STD, whereas for dry gas wells INFLOW can be set to either R-G or P-P; however, the P-P option is preferred for dry gas wells due to the more rigorous treatment of gas flow. For wet gas wells, that is gas condensate wells, INFLOW should be set to GPP. Only INFLOW equal to STD and NO are currently implemented in OPM Flow.			STD
9	AUTO	A defined character string that defines the automatic action to be taken if the economic WCUT, GOR, or WGR limits are violated and the well is to cease production. AUTO should be set to one of the following character strings: 1) STOP: the well is "stopped" at the surface and will not produce any fluids to surface; however, if there any open connections then flow may occur within the wellbore and between the open connections depending on a connection's potential with respect to all the other connections. Inter-connection flow (cross flow) can be prevented by setting the XFLOW variable to NO. In this case the well's behavior will be similar to the SHUT option described below. 2) SHUT: the well is shut at the surface and downhole, this results in no flow at the surface and no cross flow downhole. The corrective action takes places at the end of the time step in which the constraint is violated.			SHUT

[333] *Russell, D.G., Goodrich, J.H., Perry, G.E and Bruskotter, J.F "Methods of Predicting Gas Well Performance", Transactions of the ASME, Journal of Petroleum Technology (1966) 99-108.*

[334] *Whitson, C. H. and Fevang, Ø. "Generalised Pseudopressure Well Treatment in Reservoir Simulation," Presented at the IBC Technical Services Conference on Optimisation of Gas Condensate Fields, Aberdeen, UK (June 26-27, 1997).*

No.	Name	Description			Default
		Field	Metric	Laboratory	
10	XFLOW	A defined character string that defines the if cross flow should occur within the wellbore, and should be set to either: 1) YES: to allow cross flow in the wellbore through well connections. 2) NO: to disallow cross flow within the wellbore, even if the flow potentials in the well connections would allow such flow to occur. In some cases numerical issues can occur if this variable is set to YES, and resetting it to NO may resolve the issue; however, the results may not represent the physical down hole process in this case.			YES
11	PVTNUM	A positive integer greater than or equal to zero that defines the PVT table used to calculate the wellbore fluid properties that define the relationship between reservoir and surface volume rates. The default value of zero sets PVTNUM to be the PVT table of the deepest connection in the well.			0
12	DENOPT	A defined character string that sets the type of density calculation used in calculating the wellbore hydrostatic head, and should be set to one of the following character strings: 1) SEG: sets the hydrostatic head density calculation to segmented. In this cases the density is calculated between neighboring well connections and the volumes flowing from the connections. This is the more accurate calculation if the fluid properties flowing from the well connections are variable. The density calculation itself is explicit, i.e. uses the flowing volumes of the last time step. 2) AVG: sets the hydrostatic head density calculation to the average density calculation. Here the density is considered uniform across a given reservoir and is dependent on total inflow rates of each phase and the well's bottom-hole pressure The default option of 1* invokes the SEG option and is the only option implemented in OPM Flow.			SEG
13	FIPNUM	An integer value defines the FIPNUM region used to determine the reservoir conditions in calculating the well's reservoir volumes and is determined by: 1) If set to a negative integer value then the FIPNUM region of the deepest connection in the well will be used. 2) If set to zero, the default value, then the average properties for the field will be used. 3) If set to an integer value greater than zero, then the FIPNUM indicated by this value will be used.			0
14	STRMLIN1	Not used and should be defaulted with 1*.			1*
15	STRMLIN2	Not used and should be defaulted with 1*.			1*
16	TYPECOMP	Commercial compositional simulator well type model option that is not used and should be defaulted with either STD or 1*.			STD

No.	Name	Description			Default
		Field	Metric	Laboratory	
17	POLYTAB	A positive integer greater than or equal to zero that defines the polymer mixing table, as defined by the PLMIXPAR and PLYMAX keywords, to be used in calculating the well's well bore properties. The default value of zero means the table allocated via the PLMIXNUM array for the deepest connection in the well bore is utilized. Only the default value of zero is supported by OPM Flow.			0

Notes:

1) The keyword is followed by any number records with each record terminated by a "/" and the keyword should be terminated by a "/".

2) Note that the total number of individual wells declared by the WELSPECS cannot exceed the valued entered via the MXWELS variable on the WELLDIMS keyword in the RUNSPEC section.

Table 12.94: WELSPECS Keyword Description

See also the COMPDAT keyword to define a well's connections, the WCONPROD and WCONINJE keywords to define a well's production and injection targets and constraints. All the aforementioned keywords are described in the SCHEDULE section.

Example

The following example defines three wells using the WELSPECS keyword

```
--
--        WELL SPECIFICATION DATA
--
-- WELL   GROUP      LOCATION   BHP     PHASE   DRAIN   INFLOW   OPEN   CROSS   PVT
-- NAME   NAME         I     J   DEPTH   FLUID   AREA    EQUANS   SHUT   FLOW    TABLE
WELSPECS
GI01      PLATFORM    14    13    1*      GAS     1*      P-P      SHUT    NO      1*      /
GP01      PLATFORM    64    80    1*      GAS     1*      GPP      SHUT    NO      1*      /
OP01      PLATFORM    24   110    1*      OIL     1*      STD      SHUT    NO      1*      /
/
```

Here, well GI01 is a dry gas injection well that uses the dry gas pseudo inflow equation, GP01 is a gas condensate well that uses the generalized gas pseudo pressure inflow equation, and finally, OP01 is an oil well that uses the standard inflow equation. All wells will be shut if they are required to cease production, all wells disallow cross flow, and the hydrostatic head calculation is defaulted to the segment option for all wells.

If the same three wells are using the COMPTRAJ and WELTRAJ keywords to specify the connections to the simulation grid, then the WELSPECS keyword should be;

```
--
--        WELL SPECIFICATION DATA
--
-- WELL   GROUP      LOCATION   BHP     PHASE   DRAIN   INFLOW   OPEN   CROSS   PVT
-- NAME   NAME         I     J   DEPTH   FLUID   AREA    EQUANS   SHUT   FLOW    TABLE
WELSPECS
GI01      PLATFORM    1*    1*    1*      GAS     1*      P-P      SHUT    NO      1*      /
GP01      PLATFORM    1*    1*    1*      GAS     1*      GPP      SHUT    NO      1*      /
OP01      PLATFORM    1*    1*    1*      OIL     1*      STD      SHUT    NO      1*      /
/
```

Notice how the well location parameters have been defaulted with 1* in this case.

12.3.283 WELTARG – MODIFY WELL TARGET AND CONSTRAINT VALUES

Description

The WELTARG keyword modifies the target and constraints values of both rates and pressures for previously defined wells without having to define all the variables on the well control keywords: WCONPROD, WCONHIST, WCONINJE, or WCONINJH. Variables not changed by the WELTARG keyword remain the same as those previously entered via the well control keywords or previously entered WELTARG keywords. Note that the well must still be initially be fully defined using the WCONPROD or WCONINJE keywords. All the aforementioned keywords are described in the SCHEDULE section.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WELNAME	A character string of up to eight characters in length that defines the well name for which the well production rates and pressures data are being redefined. Note that the well name (WELNAME) must have been declared previously using the WELSPECS and WCONPROD (or WCONINJE) keywords in the SCHEDULE section, otherwise an error may occur.			None
2	TARGET	A defined character string that sets the item to be changed for the well the value of the item is set by item (3). 1) ORAT: reset the surface oil production rate value as defined by item (3). 2) WRAT: reset the surface water production rate value as defined by item (3). 3) GRAT: reset the surface gas production rate value as defined by item (3). 4) LRAT: reset the surface liquid (oil plus water) production rate value as defined by (3). 5) CRAT: reset the linearly combined calculated rate value as defined by (3). This option is not supported. 6) RESV: reset the in situ reservoir volume rate value as defined by (3). 7) BHP: reset the bottom-hole pressure value as defined by item (3). 8) THP: reset the tubing head pressure value for the well as defined by item (3). 9) VFP: reset the vertical lift performance table number as defined by (3). 10) LIFT: reset the artificial lift quantity for use with vertical lift performance tables. 11) GUID: reset the guide rate value for wells operating under group control. The commercial compositional simulator options: WGRA, NGL, CVAL, REIN, STRA, SATP and SATT are not applicable. Note that TARGET only defines the variable to be changed, it does not change how a well is controlled. For example, if a well is operating on ORAT control, as defined by the previously entered WCONPROD keyword, entering TARGET equal to LRAT with a value, changes the liquid constraint but the well still remains on ORAT control. Use the WELCNTL keyword in the SCHEDULE section to change the control mode of a well.			None

No.	Name	Description			Default
		Field	Metric	Laboratory	
3	VALUE	A real positive value that defines the value of the variable declared by TARGET			
	Liquid	stb/d	sm^3/day	scc/hour	
	Gas	Mscf/d	sm3/day	scc/hour	
	Res Vol	rb/d	rm^3/day	rcc/hour	
	Pressure	psia	barsa	atma	
	VFP	dimensionless	dimensionless	dimensionless	
	LIFT	same as VFPPROD or VFPINJ	same as VFPPROD or VFPINJ	same as VFPPROD or VFPINJ	None

Notes:

1) The keyword is followed by any number records with each record terminated by a "/" and the keyword should be terminated by a "/".

Table 12.95: WELTARG Keyword Description

If a well is currently a history matching well, then WELTARG should only be used to change a well's bottom-hole pressure limit, vertical flow performance table number or the artificial lift quantity.

See also the WELCNTL keyword, in the SCHEDULE section that can be used to reset the control mode, as well as a well's target and constraints of both rates and pressures.

Example

The following example below shows the oil rates for the OP01 oil producer at the start of the schedule section (January 1, 2000).

```
-- ------------------------------------------------------------------------
-- 01 JAN 2000 START OF SCHEDULE SECTION
-- ------------------------------------------------------------------------
--
--       WELL PRODUCTION WELL CONTROLS
--
-- WELL  OPEN/  CNTL   OIL    WAT    GAS   LIQ    RES    BHP    THP   VFP    VFP
-- NAME  SHUT   MODE   RATE   RATE   RATE  RATE   RATE   PRES   PRES  TABLE  ALFQ
WCONPROD
OP01     OPEN   ORAT   3000   1*     1*    1*     1*     750.0  500.  9      1* /
/
DATES
01 FEB 2000 /
/
--
--       WELL PRODUCTION AND INJECTION TARGETS
--
--  WELL WELL   TARGET
--  NAME TARG   VALUE
WELTARG
OP01     ORAT   2000                                    /
/
```

From January 1, 2000 to February 1, 2000 well OP01 is open and is on oil rate control and has a target oil rate of 3,000 stb/d, and uses VFPPROD vertical lift table number 9 with a minimum tubing head pressure constraint of 500 psia. After February 1, 2000 the well's oil rate is reduced to 2,000 stb/d and all the other parameters remain unchanged.

12.3.284 WELTRAJ – DEFINE WELL TRAJECTORY DATA

Description

The WELTRAJ keyword defines a trajectory well together with the well trajectory data (simplified directional survey data), and is used in conjunction with the COMPTRAJ keyword in the SCHEDULE section to define the well connections to the simulation grid blocks. The keyword can only be used for trajectory wells that employ the COMPTRAJ keyword to define the connections to the grid, that is, one cannot use COMPDAT keyword in the SCHEDULE section for declaring the connections to the grid for these type of wells.

Although WELTRAJ and COMPTRAJ keywords are sufficient to define the wellbore path and connections to the grid, it is still necessary to defined the general well specification parameters using the WELSPECS keyword in the SCHEDULE section. In this case, the wellhead location parameters, WELSPECS(I, J), should be defaulted with 1*.

Note

This is an OPM Flow specific keyword and will therefore cause an error in the commercial simulator.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WELNAME	A character string of up to eight characters in length that defines the well name for which the well trajectory data are being defined. Note that the well name (WELNAME) must have been declared previously using the WELSPECS keyword in the SCHEDULE section, otherwise an error may occur. Secondly, the wellhead location parameters on the WELSPECS keyword, WELSPECS(I, J), should be defaulted with 1* for trajectory wells, as the well location will be calculated by the simulator.			None
2	IBRANCH	A positive integer greater than or equal to one and less than or equal to MXBRAN on WSEGDIMS keyword in the RUNSPEC section that defines the branch number of a segment. All segments on the main stem must have IBRANCH set to one and lateral branches should have values between two and MXSEGS on the WSEGDIMS keyword in the RUNSPEC section. Only the default value of one is currently supported, that is only the main branch of a multi-segment well is supported, or a single trajectory for a conventional well.			1
3	XCORD	A real positive value representing the X coordinate in three dimensional space of the well trajectory path. The Geodesy Reference System should be same as the static model used to generate the dynamic model.			
		feet	m	cm	None
4	YCORD	A real positive value representing the Y coordinate in three dimensional space of the well trajectory path. Again, the Geodesy Reference System should be same as the static model used to generate the dynamic model.			
		feet	m	cm	None

No.	Name	Description			Default
		Field	Metric	Laboratory	
5	TVD	A real negative or positive value that defines the True Vertical Depth along the wellbore path at XCORD and YCORD. Normally, TVD is referenced to the subsea elevation, that is TVDSS.			
		feet	m	cm	None
6	MD	A real positive value that defines the equivalent measured depth along the wellbore path at TVD. Normally, MD is referenced to a reference point on the drilling rig that drilled the well, for example, Relative to Kelly Bushing, MDRKB.			
		feet	m	cm	None

Notes:

1) For this keyword the Laboratory set of units are not supported.
2) The keyword is followed by any number records with each record terminated by a "/" and the keyword should be terminated by a "/".
3) Note that the total number of individual wells declared by the WELSPECS cannot exceed the valued entered via the MXWELS variable on the WELLDIMS keyword in the RUNSPEC section.

Table 12.96: WELTRAJ Keyword Description

Using the WELTRAJ and COMPTRAJ keywords to define wells and how they are connected to grid, offers several advantages compared to the conventional approach based on the (I, J, K) co-ordinates of the grid. The approach allows for the wells to be independent of the grid, which is particularly useful when running ensemble cases, as the well connections are no longer required to be re-calculated for each ensemble case. In addition, quality control of the model is improved by using consistent perforation data in both the static and dynamic models.

See the example on the following page.

Example

The following example defines two trajectory wells oil wells, OP01 and OP02, using the WELSPECS and WELTRAJ keywords, together with their perforations using the COMPTRAJ keyword.

```
--
--       WELL SPECIFICATION DATA
--
-- WELL  GROUP     LOCATION  BHP    PHASE  DRAIN  INFLOW  OPEN  CROSS  PVT
-- NAME  NAME        I    J  DEPTH  FLUID  AREA   EQUANS  SHUT  FLOW   TABLE
WELSPECS
OP01     PLATFORM   1*   1*   1*     OIL    1*     STD    SHUT   NO     1*     /
OP02     PLATFORM   1*   1*   1*     OIL    1*     STD    SHUT   NO     1*     /
/
--
--       WELL TRAJECTORY DATA
--
-- WELL   BRAN  XCORD          YCORD          TVDSS          MD
-- NAME   NO                                 DEPTH          DEPTH
-- -----  ----  ------------  ------------  ------------  ------------
WELTRAJ
OP01      1*    2.805445e+06  3.602948e+06  -100.000000   0.0          /
OP01      1*    2.805445e+06  3.602948e+06  877.0000000   977.0        /
OP01      1*    2.805445e+06  3.602948e+06  957.9950240   1058.0       /
OP01      1*    2.805444e+06  3.602946e+06  1051.976081   1152.0       /
.....                                                                  /
OP02      1*    2.810828e+06  3.604507e+06  9371.792711   11418.0      /
OP02      1*    2.810885e+06  3.604525e+06  9443.657000   11511.0      /
OP02      1*    2.810952e+06  3.604546e+06  9531.966162   11624.0      /
OP02      1*    2.810973e+06  3.604553e+06  9560.411742   11660.0      /
/
--
--
--       WELL TRAJECTORY CONNECTION DATA
--
-- WELL  BRAN  -- PERFORATION --  COMPL OPEN  SAT CONN  WELL   KH    SKIN  D
-- NAME  NO.    TOP    BOT   REF  NO.   SHUT  TAB FACT  DIA    FACT  FACT  FACT
COMPTRAJ
OP01     1*     8230   8244  MD    1    SHUT  1*  1*   0.708  1*    0.0   1*  /
OP01     1*     8352   8380  MD    1    SHUT  1*  1*   0.708  1*    0.0   1*  /
OP01     1*     9070   9100  MD    1    SHUT  1*  1*   0.708  1*    0.0   1*  /
OP01     1*     9220   9250  MD    2    SHUT  1*  1*   0.708  1*    0.0   1*  /
OP01     1*     9266   9280  MD    2    SHUT  1*  1*   0.708  1*    0.0   1*  /
OP01     1*     9693   9703  MD    3    SHUT  1*  1*   0.708  1*    0.0   1*  /
OP01     1*     9940   9974  MD    3    SHUT  1*  1*   0.708  1*    0.0   1*  /
OP02     1*     9979   9985  TVD   1*   SHUT  1*  1*   0.708  1*    0.0   1*  /
OP02     1*    10173  10183  TVD   1*   SHUT  1*  1*   0.708  1*    0.0   1*  /
OP02     1*    10190  10204  TVD   1*   SHUT  1*  1*   0.708  1*    0.0   1*  /
OP02     1*    10327  10333  TVD   1*   SHUT  1*  1*   0.708  1*    0.0   1*  /
OP02     1*    10339  10345  TVD   1*   SHUT  1*  1*   0.708  1*    0.0   1*  /
OP02     1*    11528  11538  TVD   1*   SHUT  1*  1*   0.708  1*    0.0   1*  /
```

Here, well OP01 has eight perforation intervals, with the intervals one to three grouped into one completion, perforation intervals four to five grouped into completion number two, and finally the bottom three perforations are grouped into completion number three. In contrast, OP02 has six perforated intervals with their completion interval defaulted to one.

12.3.285 WFOAM - DEFINE WELL FOAM INJECTION CONCENTRATIONS

Description

The WFOAM keyword defines an injection wells foam concentration. The foam option must be activated by the FOAM keyword in the RUNSPEC section in order to use this keyword. Note if a well's foam concentration is not set with this keyword then default value of zero is assigned to a well.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WELNAME	A character string of up to eight characters in length that defines the well name for which the well injection foam concentration is being defined.			None
2	FOAMCON	A real positive value that defines the well's injection foam concentration. Units are dependent on the transport phase specified via the FOAMOPT1 variable on the FOAMOPTS keyword in the PROPS section. FOAMOPT1 should be set to either GAS or WATER. Currently OPM Flow only supports injecting foam via the GAS phase.			
		Gas: lb/Mscf Water: lb/stb	Gas: kg/sm^3 Water: kg/sm^3	Gas: gm/scc Water: gm/scc	None

Notes:

1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".
2) Note that the total number of individual wells declared by the WELSPECS cannot exceed the valued entered via the MXWELS variable on the WELLDIMS keyword in the RUNSPEC section.

Table 12.97: WFOAM Keyword Description

See also the FOAM keyword in the RUNSPEC section, the FOAMADS, FOAMOPTS and FOAMROCK keywords in the PROPS section.

Example

```
--
--       WELL INJECTION FOAM CONCENTRATION
--
-- WELL  FOAM
-- NAME  FOAMCON
WFOAM
GI01     0.020                                                    /
GI02     0.020                                                    /
GI03     0.020                                                    /
/
```

Here three gas wells are given an injection foam concentration of 0.020 lb/Mscf, assuming field units.

12.3.286 WFRICSEG – CONVERT FRICTION WELL TO MULTI-SEGMENT WELL

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

WFRICSEG converts a previously defined friction well, as per the WFRICTN keyword in the SCHEDULE section, to a multi-segment well. The keyword thus acts as a replacement for the WELSEGS and COMPSEGS keywords for multi-segment wells. See also the WFRICSGL keyword in the SCHEDULE section that performs similar functionality for wells in Local Grid Refinements.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.287 WFRICSGL – CONVERT FRICTION WELL TO MULTI-SEGMENT WELL (LGR)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

WFRICSGL converts a previously defined Local Grid Refinement ("LGR") friction well, as per the WFRICTNL keyword in the SCHEDULE section, to a multi-segment LGR well. The keyword thus acts as a replacement for the WELSEGS and COMPSEGL keywords for LGR multi-segment wells. See also the WFRICSEG keyword in the SCHEDULE section that performs similar functionality for wells in the global grid.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.288 WFRICTN – DEFINE WELL AS A FRICTION WELL

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The WFRICTN keyword is used to declare a previously defined well as a friction well and to set the characteristics for this type of well including: tubing size, pipe roughness, and the connections to the grid. Wellbore friction is important in horizontal and multi-lateral wells where the pressure loss along the pipe can effect a well's performance. Note that unlike other SCHEDULE section well keywords, multiple wells cannot be entered with one WFRICTN keyword, that is, the keyword must be repeated for each well.

See also the WFRICTNL keyword in the SCHEDULE section that performs similar functionality for wells in Local Grid Refinements.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.289 WFRICTNL – DEFINE WELL AS A FRICTION WELL (LGR)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The WFRICTNL keyword is used to declare a previously defined Local Grid Refinement ("LGR") well as a LGR friction well and to set the characteristics for this type of well including: tubing size, pipe roughness, and the connections to the grid. Wellbore friction is important in horizontal and multi-lateral wells where the pressure loss along the pipe can effect a well's performance. Note that unlike other SCHEDULE section well keywords, multiple wells cannot be entered with one WFRICTNL keyword, that is, the keyword must be repeated for each well.

See also the WFRICTN keyword in the SCHEDULE section that performs similar functionality for wells in the global grid.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.290 WGASPROD – DEFINE SALE GAS WELL PRODUCTION TARGETS

Description

The WGASPROD keyword declares wells to be Sales Gas producers and sets the incremental gas rate for a well and the maximum number of increments that this rate can be increased. Wells must have been previously been defined via the WELSPECS and WCONPROD keywords in the SCHEDULE section and are subject to any targets or constraints on WCONPROD keyword.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.291 WGORPEN – DEFINE WELL GOR PENALTY PARAMETERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The WGORPEN keyword defines a well's Gas-Oil Ratio ("GOR") penalty parameters used to calculate a well's oil production target for the current month, as a function of the well's previous month's average GOR. The WGORPEN calculated oil rate overwrites any oil targets set by the WCONPROD and WELTARG keywords in the SCHEDULE section. In North American, it is common practice for the regulator to enforce GOR penalties, in order to control gas production in depletion drive oil reservoirs, with the stated intention to maximize oil recovery by limiting the energy loss from the reservoir by excessive gas production.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.292 WGRUPCON – DEFINE WELL GUIDE RATES FOR GROUP CONTROL

Description

The WGRUPCON keyword defines a well's production or injection guide rate for when a well is under group control. The guide rate is used to determine a well's production target under group control in order to satisfy a group's targets and constraints, including any higher level related groups as well as the FIELD group.

Wells must have been previously defined and allocated to a group by the WELSPECS keyword in the SCHEDULE section. Wells defined to be under group control will have their production rates controlled by the group to which they belong, in addition to any well constraints defined for the wells via the WCONPROD and WCONINJE keywords in the SCHEDULE section.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WELNAME	A character string of up to eight characters in length that defines the well name for which the well production targets and constraints data are being defined. Note that the well name (WELNAME) must have been declared previously using the WELSPECS keyword in the SCHEDULE section, otherwise an error may occur.			None
2	STATUS	A defined character string that declares the status of the well to be under group control or not under group control. STATUS should be set to one of the following character strings: 1) YES: the well is under group control and its production behavior will be influenced by its assigned group, including connecting higher level groups as well as the FIELD group. 2) NO: the well is NOT under group control and its production behavior will only be influenced by its own targets and constraints. Note the default value of YES puts all wells under group control unless specified otherwise by the STATUS variable, or the TARGET variable on the WCONPROD and WCONINJE keywords in the SCHEDULE section.			YES
3	GUIDERAT	A dimensionless real number that determines the well's share of it's group production (or injection) target rate. If GUIDERAT is a positive number then the guide rate for the well is fixed until modified by this keyword at a subsequent time. If TARGET variable on this keyword is not equal to the group's controlling phase, then the GUIDERAT is converted into the groups' controlling phase and is updated every time step. If GUIDERAT is less than or equal to zero then the well's guide rate is based on the well's potential (unrestricted flow) and the potential is calculated every time step.			
		dimensionless	dimensionless	dimensionless	-1.0

No.	Name	Description			Default
		Field	Metric	Laboratory	
4	TARGET	A defined character string that sets the well's guide rate phase that the GUIDERAT value should be applied to. TARGET should be set to one of the following character strings: 1) OIL: the well's guide rate applies to the surface oil production rate. 2) WAT: the well's guide rate applies to the surface water production rate. 3) GAS: the well's guide rate applies to the surface gas production rate. 4) LIQ: the well's guide rate applies to the surface liquid (oil plus water) production rate. 5) RES: the well's guide rate applies to the in situ reservoir volume rate. 6) RAT: the well's guide rate applies to the surface rate of the injection phase. This should only be used if the well has been declared an injection well via the WCONINJE keyword in the SCHEDULE section. 7) COMB: the well's guide rate applies to the linearly combined calculated rate. This option is not supported by OPM Flow. TARGET may be defaulted if GUIDERAT has been defaulted, either by 1* or a value less than or equal to zero.			None
5	SCALE	A real value that is used to multiple the GUIDERAT or the calculated well potentials to determine the final GUIDERAT for the well.			
		dimensionless	dimensionless	dimensionless	1.0
Notes: 1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".					

Table 12.98: WGRUPCON Keyword Description

See also the GCONPROD the GCONINJE keywords to define a group's production and injection targets and constraints, and the WCONPROD and WCONINJE keyword to define a well's production and injection characteristics. All the aforementioned keywords are described in the SCHEDULE section.

Example

The following example defines the guides rates all oil and gas producers and the gas injectors as follows:

```
--
--       DEFINE WELL GUIDES FOR GROUP CONTROL
--
-- WELL  GRUP  GUIDE  GUIDE  SCALE
-- NAME  CNTL  RATE   PHASE  FACT
WGRUPCON
'GI*'    YES   0      RAT    1.0                    /
'GP*'    YES   0      GAS    1.0                    /
'OP*'    NO    2      OIL    1.0                    /
/
```

Both the gas producers ('GP*') and injectors ('GI'*) are under group control with their guide rates based on their potentials. The gas injectors are controlled based on their potential surface gas injection rates and the gas producers on their potential surface gas production rates. In comparison, the oil wells (OP*) are controlled by their own targets and constraints.

12.3.293 WHEDREFD – DEFINE WELL HYDRAULIC HEAD REFERENCE DEPTH

Description

The WHEDREFD keyword sets the hydraulic head reference depth for reporting the hydraulic head pressure for the well, for wells that have previously been defined by the WELSPECS keyword in the SCHEDULE section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WELNAME	A character string of up to eight characters in length that defines the well name for which the well hydraulic head reference depth data is being defined.			None
2	HYDREF	A real value that defines the hydraulic head reference depth for reporting the hydraulic head pressure for the well. HYDREF cannot be defaulted on the keyword; however if a well has not been set by this keyword HYDREF is set equal to the value on the HYDRAHEAD keyword.			
		feet	m	cm	None
Notes: 1) The keyword is followed by any number records with each record terminated by a "/" and the keyword should be terminated by a "/".					

Table 12.99: WHEDREFD Keyword Description

See also the HYDRAHEAD in the PROPS section.

Example

The following example defines three wells hydraulic head reference depths for reporting, using the WHEDREFD keyword

```
--
--        WELL HYDRAULIC HEAD REFERENCE DEPTH
--
-- WELL   HYDREF
-- NAME   DEPTH
WHEDREFD
OP01      150.0                                                    /
OP02      175.0                                                    /
OP03      150.0                                                    /
/
```

Here, well OP01 and OP03 have their hydraulic head reference depths set to 150.0 ft and well OP02's hydraulic head reference depth is set to 175.0 ft.

12.3.294 WHISTCTL - DEFINE WELL HISTORICAL TARGET PHASE

Description

The WHISTCTL keyword changes the target control for wells declared as history match wells via the WCONHIST keyword in the SCHEDULE section. The target phase is set on the WCONHIST keyword and WHISTCTL overrides this value for all subsequent entries on the WCONHIST keyword.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	TARGET	A defined character string that sets the observed target production phase for the well, all the other phases are calculated unconstrained and used for reporting only. The simulator will attempt to meet the TARGET based on the phase rate stated in items (4) to (6) and (10) on the WCONHIST keyword. TARGET should be set to one of the following character strings: 1) ORAT: the target is set to the surface oil production rate as defined by item (4) on the WCONHIST keyword. 2) WRAT: the target is set to the surface water production rate as defined by item (5) on the WCONHIST keyword. 3) GRAT: the target is set to the surface gas production rate as defined by item (6) on the WCONHIST keyword. 4) LRAT: the target is set to the surface liquid (oil plus water) production rate and is calculated by the simulator using (4) and (5) on the WCONHIST keyword. 5) RESV: the target is set to the in situ reservoir volume rate and is calculated by the simulator using items (4), (5) and (6) on the WCONHIST keyword. 6) BHP: the target rate is set to the bottom-hole pressure as defined by item (10) on the WCONHIST keyword. 7) NONE: revert back to the TARGET control mode on the WCONHIST keyword. The TARGET control mode defined on this keyword resets the TARGET control mode on the WCONHIST keyword in the SCHEDULE section, from the time the WHISTCNTL is invoked, thus avoiding changing the control model on all subsequent WCONHIST keywords.			None
2	END	A defined character string that defines if the simulation should terminate if the well has switch to BHP control by the simulator, and should be set to one of the following character strings: 1) NO: no action is taken and the run continues. 2) YES: terminate the run at the next report time step. Wells set to BHP control via the WCONHIST or WHISCTL keywords are ignored. Only END equal to NO is currently supported in OPM Flow.			NO

Notes:

1) The keyword is terminated by a "/".

Table 12.100: WHISTCTL Keyword Description

History matching wells are handled differently then ordinary wells that use the WCONPROD keyword for controlling their production targets and constraints. However, the wells still need to be defined like ordinary production wells using the WELSPECS keyword in the SCHEDULE section. History matching wells are converted to ordinary wells by restating a well's control mode using either the WCONPROD or WELTARG keywords in the SCHEDULE section.

See also the WCONHIST and WCONINJH keywords that are used to define the historical production and injection data, All the aforementioned keywords are described in the SCHEDULE section.

Example

The example below shows the observed gas rates for the OP01 oil producer for the first quarter of 2000.

```
-- ------------------------------------------------------------------------
-- 01 JAN 2000 START OF SCHEDULE SECTION
-- ------------------------------------------------------------------------
--
--       DEFINE WELL HISTORICAL TARGET PHASE
--
--       CNTL    BHP
--       MODE    STOP
WHISTCTL
         RESV    NO                                                    /
--
--       WELL HISTORICAL PRODUCTION CONTROLS
--
-- WELL  OPEN/  CNTL   OIL    WAT    GAS    VFP    VFP   THP   BHP
-- NAME  SHUT   MODE   RATE   RATE   RATE   TABLE  ALFQ  PRES  PRES
WCONHIST
OP01     OPEN   ORAT  15.5E3  100.0  1550   10      1*   900.0 1*     /
/
DATES
01 FEB 2000 /
/
--
--       WELL HISTORICAL PRODUCTION CONTROLS
--
-- WELL  OPEN/  CNTL   OIL    WAT    GAS    VFP    VFP   THP   BHP
-- NAME  SHUT   MODE   RATE   RATE   RATE   TABLE  ALFQ  PRES  PRES
WCONHIST
OP01     OPEN   ORAT  15.2E3  150.0  1520   1*      1*   875.0 3250.0 /
/
DATES
01 MAR 2000 /
/
--
--       WELL HISTORICAL PRODUCTION CONTROLS
--
-- WELL  OPEN/  CNTL   OIL    WAT    GAS    VFP    VFP   THP   BHP
-- NAME  SHUT   MODE   RATE   RATE   RATE   TABLE  ALFQ  PRES  PRES
WCONHIST
OP01     OPEN   ORAT  15.0E3  200.0  1500   1*      1*   850.0 1*     /
/
```

From January 1, 2000 the WCONHIST keyword defines well OP01, which is open and is on oil rate control, to produce 15,500 stb/d oil, with the observed rates of 100 stb/d of water and 15.5 MMscf/d of gas. However the WHISCTL keyword resets the target control to reservoir voidage from January 1, 2000 and onward. This is useful in initial history matching runs to get a "reasonable" pressure match, by ensuring that the total reservoir withdrawals are correct, although the individual phase withdrawals will not match. Once a reasonable pressure match is achieved for the reservoir then one can reset TARGET to the sales phase, OIL or GAS, and continue with the matching of all the phases.

12.3.295 WHTEMP – DEFINE WELL TUBING HEAD TEMPERATURE PARAMETERS

Description

This keyword, WHTEMP, sets the parameters for the Tubing Head Temperature calculation, which can either be a constant value, or from a table lookup using a VFPPROD table, via the VFPPROD keyword in the SCHEDULE section, containing tubing head temperature data.

This keyword can only be used if the thermal option has been activated by the THERMAL keyword in the RUNSPEC section. Note this is different to the commercial simulator that uses the TEMP keyword in the RUNSPEC section to activate the black-oil thermal model.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WELNAME	A character string of up to eight characters in length that defines the well name for which the well production targets and constraints data are being defined. Note that the well name (WELNAME) must have been declared previously using the WELSPECS keyword in the SCHEDULE section, otherwise an error may occur.			None
2	VFPTAB	A positive integer greater than or equal to zero that references the production vertical lift performance table (VFPPROD), containing the tubing head temperature data for the well. Note, a well must have both a VFPPROD pressure and a VFPPROD temperature table, if the tubing head temperatures are to be calculated. Alternatively, if a constant tubing head temperature for a production well is to be defined via the TEMP parameter, then VFPTAB should be defaulted with 1* instead.			None
		dimensionless	dimensionless	dimensionless	
3	TEMP	A real positive value greater than zero that defines a constant tubing head temperature for a production well. In this case the VFPTAB parameter should be defaulted with 1* if a constant tubing head temperature for a production well.			
		°F	°C	°C	None

Notes:

1) Note either VFPTAB or TEMP must be supplies, that is one can specify both.
2) The keyword is followed by any number records with each record terminated by a "/" and the keyword should be terminated by a "/".

Table 12.101: WHTEMP Keyword Description

See also the VFPPROD keyword in the SCHEDULE section.

Example

The following example defines three wells tubing head temperature parameters using the WHTEMP keyword

```
--
--       DEFINE WELL TUBING HEAD TEMPERATURE PARAMETERS
--
-- WELL  VFP    TUB
-- NAME  TABLE  TEMP
WHTEMP
OP01     5                                                              /
OP02     1*     150                                                     /
OP03     5                                                              /
/
```

Here, well OP01 and OP03 used VFPPROD table number five to calculate the tubing head temperature, and well OP02's uses a constant 150o tubing head temperature.

12.3.296 WINJCLN – CLEAN A FRACTION OF A DEPOSITED FILTER CAKE

Description

The WINJCLN keyword signals that a filter cake should be completely or partially cleaned – this effectively multiplies the accumulated filter cake skin.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WELNAME	A character string of up to eight characters in length that defines the well name for which the filter cake properties are being defined. Note that the well name (WELNAME) must have been declared previously using the WELSPECS keyword in the SCHEDULE section, otherwise an error may occur.			None
2	FCLNFRAC	A real positive value between 0 and 1 that defines the fraction of filter cake permeability (skin factor) to be removed. The accumulated filter cake skin factor for matching connections will be multiplied by (1 – FCLNFRAC), so the default value of 1 will completely clean the filter cake.			
		dimensionless	dimensionless	dimensionless	1
3	I	An integer that defines the matching connection location in the I-direction. If set to < 1 then all connections in the I-direction that also satisfy J and K criteria are selected.			-1
4	J	An integer that defines the matching connection location in the J-direction. If set to < 1 then all connections in the J-direction that also satisfy I and K criteria are selected.			-1
5	K	An integer that defines the matching connection location in the K-direction. If set to < 1 then all connections in the K-direction that also satisfy I and J criteria are selected.			-1

Notes:

1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".
2) Repeated occurrences of this keyword will have a multiplicative effect.

Table 12.102: WINJCLN Keyword Description

See also the WINJDAN keyword to define the filter cake properties and the WINJFCNC keyword to define a well's injected filtrate concentration. All the aforementioned keywords are described in the SCHEDULE section.

Example

The following example signals the filter cake clean up for water injection wells using WINJCLN:

```
--
```

```
--       WELL FILTER CAKE CLEAN UP
--
-- WELL  CLEAN  --LOCATION--
-- NAME  FRAC    II  JJ  KK
WINJDAM
INJ-A1   0.4      0   0   3  /
INJ-A1   0.4      0   0   4  /
INJ-B*   0.9    /
/
```

In well INJ-A1 forty percent of the filter cake is cleaned up in well connections in layers 3 and 4 (filter cake skin is multiplied by 0.6). In wells matching INJ-B* ninety percent of the filter cake is cleaned up in all well connections (filter cake skin is multiplied by 0.1).

12.3.297 WINJDAM – DEFINE WELL FILTER CAKE PROPERTIES

Description

The WINJDAM keyword defines filter cake properties for wells that have previously been defined by the WELSPECS keyword in the SCHEDULE section. This keyword must be accompanied by the WINJFCNC keyword to define the injected filtrate concentration.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WELNAME	A character string of up to eight characters in length that defines the well name for which the filter cake properties are being defined. Note that the well name (WELNAME) must have been declared previously using the WELSPECS keyword in the SCHEDULE section, otherwise an error may occur.			None
2	GEOMETRY	A defined character string that the filter cake geometry of the well. GEOMETRY must be set to one of the following character strings: 1) LINEAR: the filter cake model will assume a linear geometry. 2) RADIAL: the filter cake model will assume a radial geometry.			None
3	FCPERM	A real positive value that defines the filter cake permeability.			
		mD	mD	mD	None
4	FCPORO	A real positive value that defines the filter cake porosity.			
		dimensionless	dimensionless	dimensionless	0.3
5	FCRADIUS	Well radius to use in skin factor calculations. If FCRADIUS is defaulted then half the diameter from the COMPDAT keyword item 9 will be used if it is defined otherwise 0.5 feet will be used.			
		feet	m	cm	0.5 feet
6	FCAOF	A real positive value that defines the flow area for each connection. If FCAOF is defaulted then the value will be evaluated as $2\pi r_w h$, where r_w is equal to FCRADIUS, and h is evaluated using the permeability thickness kh value from the COMPDAT keyword item 10 and the permeability FCPERM item 3 above.			
		ft^2	m^2	cm^2	$2\pi r_w h$
7	I	An integer that defines the matching connection location in the I-direction. If set to < 1 then all connections in the I-direction that also satisfy J and K criteria are selected.			-1
8	J	An integer that defines the matching connection location in the J-direction. If set to < 1 then all connections in the J-direction that also satisfy I and K criteria are selected.			-1

No.	Name	Description			Default
		Field	Metric	Laboratory	
9	K	An integer that defines the matching connection location in the K-direction. If set to < 1 then all connections in the K-direction that also satisfy I and J criteria are selected.			-1
Notes: 1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/". 2) New records overwrite previously specified data for matching connections.					

Table 12.103: WINJDAM Keyword Description

See also the WINJFCNC keyword to define a well's injected filtrate concentration and the WINJCLN keyword to signal that a filter cake should be completely or partially cleaned. All the aforementioned keywords are described in the SCHEDULE section.

Example

The following example defines the filter cake properties for a water injection well using WINJDAM:

```
--
--       WELL FILTER CAKE PROPERTIES
--
-- WELL  LINEAR/  PERM   PORO   WELL   FLOW   --LOCATION--
-- NAME  RADIAL                 RAD    AREA    II  JJ  KK
WINJDAM
INJ      LINEAR   10     0.32   /
INJ      RADIAL   1      0.25   1*     1*       0   0   3 /
/
```

Well INJ initially has a linear filter cake model for all completions with permeability of 10 mD, porosity of 0.32, and default well radius and flow area based on data from the COMPDAT keyword. This is then overwritten for completions in layer 3 with a radial filter cake model with permeability of 1 mD and porosity of 0.25 (again with default well radius and flow area).

12.3.298 WINJFCNC – DEFINE INJECTION WELL FILTRATE CONCENTRATION

Description

The WINJFCNC keyword defines the injected filtrate concentration for wells that have previously been defined by the WELSPECS keyword in the SCHEDULE section.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WELNAME	A character string of up to eight characters in length that defines the well name for which the filter cake properties are being defined. Note that the well name (WELNAME) must have been declared previously using the WELSPECS keyword in the SCHEDULE section, otherwise an error may occur.			None
2	FCONCPPM	A real positive value that defines the volumetric concentration of filtrate in the injected water. Note this can be defined by a user-defined argument (UDA).			
		ppm	ppm	ppm	0

Notes:

1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".
2) New records overwrite previously specified data.

Table 12.104: WINJFCNC Keyword Description

See also the WINJDAM keyword to define the filter cake properties and the WINJCLN keyword to signal that a filter cake should be completely or partially cleaned. All the aforementioned keywords are described in the SCHEDULE section.

Example

The following example defines the filtrate injection concentration for a water injection well using WINJFCNC:

```
--
--       WELL FILTRATE INJECTION CONCENTRATION
--
-- WELL  FILTRATE
-- NAME  CONC
WINJFCNC
INJ-A1   10   /
INJ-B*   30   /
/
```

Well INJ-A1 has a filtrate concentration of 10 ppm in the injection water, and wells matching INJ-B* have a concentration of 30 ppm.

12.3.299 WINJGAS – DEFINE GAS INJECTION PROPERTIES FOR A WELL

Description

The WINJGAS keyword defines the properties of the injection gas stream, for a given well. Once a gas well stream has been defined via the WELLSTRE keyword in the RUNSPEC section, it can be used with either the WINJGAS or GINJGAS keywords, to set the injected gas composition. Similarly, if an oil well stream has been defined by WELLSTRE, then the well stream can be used with the WINJOIL keyword in the SCHEDULE section, to specify the injected oil composition. Note that, it is unnecessary to use WINJGAS for wells subordinate to a group having gas injection control, with the gas properties set by GINJGAS keyword in the RUNSPEC section. In this case the injection stream is defined by the GINJGAS keyword. However, if a gas injection well under group control users the WINJGAS keyword, then this fluid, and not the group's fluid will be injected instead, at a rate controlled by the group.

The keyword should only be used if the CO2STORE and GASWAT keywords in the RUNSPEC section have also be activated for the gas-water two component model.

Note

This is an OPM Flow keyword used with OPM Flow's CO2STORE and GASWAT keywords in the RUNSPEC section, and should not be confused with the more general version of the WINJGAS keyword used in the commercial compositional simulator.

Secondly, although OPM Flow parses the keyword, the simulator currently ignores the data for this keyword.

No.	Name	Description	Default
1	WELNAME	A character string of up to eight characters in length that defines the injection well name, for which the gas injection properties are being specified. Note that the well name (WELNAME) must have been declared previously using the WELSPECS keyword in the SCHEDULE section, otherwise an error may occur.	None
2	STREAM	A defined character string that determines the properties of the injection gas, and as should be set to one of the following values: 1) GAS: The field separator gas composition is used as the injected gas. 2) GRUP: The well's group or upstream group injection fluid is used as the injected gas. 3) GV: This option enables the vapor production of a group, as declared via the SOURCE parameter on this record, to be used as the injected gas composition. 4) MIX: In this case, the gas injection composition is taken from either the WINJMIX or WINJORD keywords in the SCHEDULE section, with the name of injected fluid given by the SOURCE parameter on this record. 5) STREAM: Here the gas injection composition is given by the WELLSTRE keyword in the SCHEDULE section, with the name of injected fluid given by the SOURCE parameter on this record. 6) WV: This option enables the vapor production from a given well to be used as the injected gas, with the name of the well given by the SOURCE parameter on this record.	GRUP

No.	Name	Description	Default
		Only the STREAM option is supported by OPM Flow.	
3	SOURCE	A character string of up to eight characters in length, that defines the source of the gas injection stream, based on the value of STREAM. If STREAM equals GV, then source should be set to a group name. For STREAM equal to MIX or STREAM, then SOURCE should be set to the name of the gas stream, as defined by the WINJMIX, WINJORD, or WELLSTRE keywords.	None
4	MAKEUP	The name of the well stream used for the make-up gas, if make-up gas is required for WELNAME to match the injection target for the well. This option is not supported by OPM Flow.	None
5	STAGE	STAGE defines the separator stage from which the injection gas should be taken from. In this case, the vapor phase from any stage may be used, and the default value of zero users the total vapor phase from the separator. This option is not supported by OPM Flow.	0
Notes: 1) The keyword is followed by any number records with each record terminated by a "/" and the keyword should be terminated by a "/".			

Table 12.105: WINJGAS Keyword Description

Example

The following example defines how to specify a two component formulation, together with defining the names of the composition components, to be used with the CO2STORE and GASWAT options.

```
-- ==================================================================================
--
-- PROPS SECTION
--
-- ==================================================================================
PROPS              --
--       CONFIRM NUMBER OF COMPOSITIONAL COMPONENTS (OPM FLOW KEYWORD)
--
NCOMPS
         2                                                                        /
--
--       DEFINE COMPOSITIONAL COMPONENTS NAMES (OPM FLOW KEYWORD)
--
CNAMES
         'CO2'
         'H2O'                                                                    /
```

The second part of the example, defines the well stream for the above two component CO_2 water system.

```
-- ================================================================================
--
-- SCHEDULE SECTION
--
-- ================================================================================
SCHEDULE
--
--       WELL STREAM INJECTION COMPOSITION (OPM FLOW Keyword)
--
-- WELL     -- WELL STREAM COMPOSITIONAL COMPONENT       --
-- STREAM   --           MOLE FRACTIONS                  --
WELLSTRE
'C02STREAM'    1.000   0.000                              /
/
--
--       WELL GAS INJECTION PROPERTIES
--
-- WELL  STREAM   SOURCE      MAKEUP   SEP
-- NAME  OPTION   DEPTH       GAS      STAGE
WINJGAS
GI01     STREAM   C02STREAM   1*       1* /
/
```

Here the well stream consists of 100% CO2 and zero water, with well GI01 using the gas injection properties as defined by the WELLSTRE keyword and allocated via the WINJGAS keyword.

Finally, the gas injection rate is set via the WCONINJE keyword as shown below.

```
--
--       WELL INJECTION CONTROLS
--
-- WELL  FLUID  OPEN/  CNTL  SURF   RESV   BHP   THP   VFP
-- NAME  TYPE   SHUT   MODE  RATE   RATE   PRES  PRES  TABLE
WCONINJE
GI01     GAS    OPEN   RATE  10E4   1*     300    1*    1*                    /
/
```

Thus, gas injector GI01 will injection 10 x 10^4 m^3 of CO_2 per day, assuming metric units.

12.3.300 WINJMULT – DEFINE WELL PRESSURE DEPENDENT INJECTIVITY MULTIPLIERS

Description

The WINJMULT keyword defines pressure dependent injectivity multipliers for injection wells and can be used to approximate the increase or decrease in a well's injectivity due to hydraulic fracturing in water injection wells. Only injection wells are processed by this keyword, even if production wells have been entered by the keyword.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WELNAME	A character string of up to eight characters in length that defines the well name for which the well connection data are being defined. Note that the well name (WELNAME) must have been declared previously using the WELSPECS keyword in the SCHEDULE section, otherwise an error may occur.			None
2	FRACPRES	FRACPRES is the fracture opening pressure (Pfractue) used in equation 12.34.			None
		psia	barsa	atma	
3	ALPHA	ALPHA is the multiplier gradient, α, in equation 12.34.			
		1/psia 0.0	1/barsa 0.0	1/atma 0.0	Defined
4	OPTION	A defined character string that determines how the data on this keyword is applied, and should be set to one of the following character strings: 1) CIRR: The injectivity multiplier as applied to the selected connections (I, J, K) is irreversible, and the pressure at the connection's sand face (Pwbhp) is used in equation 12.34, instead of the well's flowing bottom-hole pressure. This option means that even if the pressures in the wellbore, and therefore the sand face pressures, later declines, injectivity remains unchanged for all the connections. 2) CREV: The injectivity multiplier as applied to the selected connections (I, J, K) is reversible, and the pressure at the connection's sand face is used in equation 12.34, The reversibility of this option means that if the pressures in the wellbore, and therefore the sand face pressures, later declines, injectivity will also decline for all the connections. 3) WREV: The injectivity multiplier as applied to all connections in the well is reversible, and the wells' flowing bottom-hole pressure (P_{wbhp}) is used in equation 12.34, instead of the pressure at the connection's sand face. The connections stipulated by (I, J, K) are ignored. The reversibility of this option means that if the pressures in the wellbore, later declines, injectivity will also decline for all the connections.			WREV

No.	Name	Description			Default
		Field	Metric	Laboratory	
5	I	An integer value less than or equal to NX that defines the connection location in the I-direction. If set to zero, a negative value, or defaulted with 1* then all connections in the I-direction will be multiplied by ALPHA, depending on the selected OPTION value.			1*
6	J	An integer value less than or equal to NY that defines the connection location in the J-direction. If set to zero, a negative value, or defaulted with 1* then all connections in the J-direction will be multiplied by ALPHA, depending on the selected OPTION value.			1*
7	K	An integer value less than or equal to NZ that defines the connection location in the K-direction. If set to zero, a negative value, or defaulted with 1* then all connections in the K-direction will be multiplied by ALPHA, depending on the selected OPTION value.			1*
Notes: 1) The keyword is followed by any number of records. 2) Each record is terminated by a "/" and the keyword should be terminated by a "/".					

Table 12.106: WINJMULT Keyword Description

The methodology for applying the well pressure dependent injectivity multipliers is outlined in equation 12.34.

$$\begin{aligned} Multiplier = \quad & 1.0 + \alpha\left(P_{WBHP} - P_{fracture}\right) \quad \text{for } P_{WBHP} > P_{fracture} \\ Multiplier = \quad & 1.0 \quad \text{for } P_{WBHP} < P_{fracture} \end{aligned} \tag{12.34}$$

where:

$Multiplier$ = the resulting multiplier to be applied to the selected connections.
α = the ALPHA multiplier gradient in Table 12.106.
P_{WBHP} = either the well's current flowing bottom-hole pressure, or the selected individual connection's sand face pressure.
$P_{fracture}$ = the effective fracture opening pressure, FRACPRES in Table 12.106.

The equation is applied every time there is a calculation to determine a well's flow rate, this results in the calculated mobility rates being scaled up by the Multiplier value in equation 12.34. Note also that since the scaling is performed on the connection fluid mobility values, then the reported connection transmissibilities in the print file etc. remain unchanged.

Note

If all the connection parameters (I, J, K) are defaulted, or OPTION is set equal to WREV, then the multiplier is applied to all connections in the well.

If any of the connection parameters (I, J, K) have positive values and OPTION is set equal to CIRR or CREV, then the multiplier is applied to the selected connections as determined by the (I, J, K) parameters.

Example

The example below show the WINJMULT keyword for three water injection wells.

```
--
--       DEFINE WELL CONNECTION MULTIPLIERS
--
-- WELL  FRAC  MULT    FRAC  --LOCATION--
-- NAME  PRES  VALUE   OPTN    I   J   K
WINJMULT
WI01     4200  0.0250  1*      1*  1*  1*        /
WI02     4250  0.0025  CIRR    1*  1* 145       /
WI02     4250  0.0025  CIRR    1*  1* 146       /
WI02     4250  0.0025  CIRR    1*  1* 147       /
WI03     4400  0.0055  CREV    1*  1* 160       /
WI03     4400  0.0055  CREV    1*  1* 165       /
/
```

The first well, WI01, uses the default value for OPTION, that is WREV, which means that the injectivity multiplier will be applied to all the connections in the well and the well's bottom-hole pressure is used in the calculation. In this case the process is reversible. The second well, WI02, applies the injectivity multiplier to all connections in layers 145 to 147 using the sand face pressures in the calculation, and the process is irreversible. Finally for well WI03, the multiplier is applied to all connections in layers 160 and 165 using the sand face pressures in the calculation, and in this case the process is reversible.

12.3.301 WINJTEMP – DEFINE INJECTION FLUID THERMAL PROPERTIES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

WINJTEMP defines the injection fluid thermal properties for when the thermal option has been activated by the THERMAL keywords in the RUNSPEC. Only water and gas injection is supported.

This keyword can only be used if OPM Flow's thermal option has been activated by the THERMAL keyword in the RUNSPEC section. Note this is different to the commercial simulator that uses the TEMP keyword in the RUNSPEC section to activate the black-oil thermal model.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WELNAME	A character string of up to eight characters in length that defines the well name for which the well injection fluid thermal properties are being defined. Note that the well name (WELNAME) must have been declared previously using the WELSPECS keyword in the SCHEDULE section, otherwise an error may occur.			None
2	STEAMQAL	STEAMQAL is a real positive value greater than or equal to zero and less than or equal to one that defines the steam quality of the injected fluid for the defined well. This parameter should be defaulted using 1* as STEAMQUAL is not used by OPM FLOW, as only water and gas injection is supported. This data is used by the commercial simulator's THERMAL option and is not supported by OPM Flow's THERMAL option.			
		dimensionless	dimensionless	dimensionless	1*
3	TEMP	TEMP is a real positive value that defines the temperature of the injected fluid for the defined well.			
		°F	°C	°C	None
4	PRES	PRES is a real positive value that defines the pressure of the injected fluid for the defined well.			
		psia	barsa	atma	None
5	ENTHALPY	ENTHALPY is a real positive value that defines the specific enthalpy of the injected fluid for the defined well. This is data is used by the commercial simulator's THERMAL option and is not supported by OPM Flow's THERMAL option.			
		Btu/lbs-M	kJ/kg-M	J/gm-M	None

Notes:

1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".

Table 12.107: WINJTEMP Keyword Description

Example

The following example shows the WINJTEMP keyword for when OPM Flow's temperature option has been activated by the THERMAL keyword in the RUNSPEC section.

```
--
--        INJECTION FLUID THERMAL PROPERTIES
--
-- WELL   STEAM   INJ     INJ     SPEC
-- NAME   QUAL    TEMP    PRES    ENTH
WINJTEMP
WI01      1*      68.0    220.0     1*                          /
WI02      1*      70.0    230.0     1*                          /
/
```

Here the water injection fluid's temperature and pressure, in field units, for two water injections well are defined. Notice that both the steam quality and the specific enthalpy of the injected fluid for the defined wells are defaulted (or skipped), as OPM Flow's THERMAL option does not support this data.

12.3.302 WLIFT – DEFINE WELL RE-TUBING, THP AND LIFT SWITCHING WORKOVER OPERATIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The WLIFT defines the automatic workovers parameters for changing out wellbore tubing, changing the THP limit (for example switching from the high stage pressure separator to the low stage pressure separator), or changing the artificial lift parameters, for wells.

OPM Flow does not have this feature and hence this keyword is ignored by OPM Flow and the WLIFT keyword has no effect on the simulation.

12.3.303 WLIFTOPT – DEFINE WELL GAS LIFT OPTIMIZATION PARAMETERS

Description

The WLIFTOPT defines which wells should use the Gas Lift Optimization facility in order to maximize oil production, as well as defining the associated gas lift optimization parameters for a given well. The keyword can also be used to switch off gas lift optimization for a well. Gas lift optimization is invoked via the LIFTOPT keyword in the SCHEDULE section. Note that the LIFTOPT keyword should precede the WLIFTOPT keyword in the SCHEDULE section in order to activate the gas lift optimization facility.

Wells are allocated to groups when the wells are specified by the WELSPECS keyword in the SCHEDULE section. Wells defined to be under group control will have their production rates and gas lift gas constraints (GLIFTOPT keyword in the SCHEDULE section) controlled by the group to which they belong, in addition to any well constraints defined for the wells, including the gas optimization parameters on the WLIFTOPT keyword.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WELNAME	A character string of up to eight characters in length that defines the well name for which the well gas lift optimization parameters are being defined. Note that the well name (WELNAME) must have been declared previously using the WELSPECS keyword in the SCHEDULE section, otherwise an error may occur.			None
2	OPTLIFT	A defined character string that sets if a well's gas lift gas rate should be calculated by the gas lift optimization facility or not, and should be set to: 1) NO: In this case the gas lift gas is a constant determined from the MXLIFT variable on this keyword, the ALQ-WELL variable on the WCONPROD keyword, or the TARGET and VALUE variables on the WELTARG keyword. 2) YES: Activates the gas lift optimization for the given well.			None
3	MXLIFT	A real value that defines the total amount of gas lift gas available for this well, multiplied by the well's efficiency factor. 1) If OPTLIFT is defined as NO then MXLIFT is considered a fix gas lift gas rate. However, if MXLIFT is defaulted (1*) then MXLIFT is unchanged from the previous entered value. 2) If OPTLIFT equals YES and MXLIFT is defaulted (1*), then MAXLIFT is taken from the largest value of the ALQ variable on the well's associated VFPPROD table. Note that the value entered here should be in the range entered in the VFPPROD table allocated to the well, otherwise errors may occur when optimizing the gas lift gas injection rate for the well.			
		Mscf/d	sm^3/day	scc/hour	1*

No.	Name	Description			Default
		Field	Metric	Laboratory	
4	OPTWGT	OPTWGT (βw) is real positive value that defines a weighting factor for allocating the available gas lift gas to a well. An increment of gas lift gas supply is allocated to a well based on the well's current incremental gradient multiplied by OPTWGT using the following formulae: $$Gradient = \left(\frac{(\beta_w \times \Delta Q_{Oil})}{(\Delta Q_{GasLift} + \beta_g \times \Delta Q_{Gas})} \right)$$ Where: βw = is OPTWGT, the weighting factor for the preferential allocation of lift gas, βg = is the gas production rate weighting factor, ΔQOil = is the increment/decrement in oil production rate, ΔQ_{gas} = is the increment/decrement in gas production rate, and $\Delta Q_{GasLift}$ = is the increment/decrement in gas lift gas rate. Note by default βg, the gas production rate weighting factor, is set to zero, and therefore the gradient equation simplifies to: $$Gradient = \left(\frac{\beta_w \times \Delta Q_{Oil}}{\Delta Q_{GasLift}} \right)$$ OPTWGT is ignored if OPTLIFT is equal NO.			
		dimensionless	dimensionless	dimensionless	1
5	MINGAS	A real value that defines the minimum amount of gas lift gas available to the well, multiplied by the well's efficiency factor. The allocation of the gas lift gas is determined by: 1) If MINGAS is a positive value then this value is allocated to the well unless the well is unable flow with this quantity of gas lift gas. Alternatively, if the well is able to meet it's target rate without applying MINGAS, then the MINGAS rate is not applied to the well. 2) If MINGAS is a negative value, then the well is supplied with sufficient gas lift gas to allow the well to flow, subject to the maximum allowed gas lift quantity, as per MXLIFT variable on this keyword. The negative value itself is not used in any calculations. 3) If there is insufficient available gas lift gas, the wells are assigned values of MINGAS based on the decreasing order of their weighting factors as calculated per OPTWGT variable. 4) Wells belonging to groups that can meet their production targets without gas lift, will have their MINGAS values not applied, that is no gas lift is applied. The exception is that if OPTWGT has been set to a value greater than or equal to one, then the well will use the MINGAS value for it's gas lift gas, even if the group's target can be satisfied without gas lift. However, if both the well's group and the well can meet their production targets, then MINGAS will not be applied. This parameter is ignored if OPTLIFT is defined as NO.			
		Mscf/d	sm^3/day	scc/hour	0.0

No.	Name	Description			Default
		Field	Metric	Laboratory	
6	OPTGAS	OPTGAS (βg) is real positive value that defines the incremental gas weighting factor for allocating the available gas lift gas to a well. An increment of gas lift gas supply is allocated to a well based on the well's current incremental gradient as described by the definition of the OPTWGT (βg) variable above, that is by the following formulae: $$Gradient = \left(\frac{(\beta_w \times \Delta Q_{Oil})}{(\Delta Q_{GasLift} + \beta_g \times \Delta Q_{Gas})} \right)$$ See OPTWGT for a definition of the variables in the equation. This parameter is ignored if OPTLIFT is defined as NO.			
		dimensionless1	dimensionless	dimensionless	0.0
7	OPTLIMIT	A defined character string that defines if additional gas lift gas should be applied to the well, if the well's group gas target has been satisfied but the group's oil rate limit has not been achieved. 1) NO: Additional gas lift gas is not available for the given well. 2) YES: Additional gas lift gas is available for the given well. In cases where a well receiving additional gas lift may cause the well's group to exceed the group's gas target, normally the well will not be assigned the additional gas lift gas. However, if OPTLIMIT is set to YES, then this constraint is removed. This results in the gas lift optimization procedure continuing to maximize the oil rate, subject to available constraints. However, upon completion of the optimization process, applying the group controls may negate the gain from the gas lift optimization process. This parameter is ignored if OPTLIFT is defined as NO.			NO

Notes:

1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".

Table 12.108: WLIFTOPT Keyword Description

See also the LIFTOPT keyword to activate gas lift optimization, the GLIFTOPT keyword to define the group gas lift optimization controls, GRUPTREE keyword to define the hierarchy of the groups below the FIELD level, the WCONPROD and WCONINJE keywords to define a well's production and injection rate targets and constraints. All the aforementioned keywords are described in the SCHEDULE section.

Example

The following example fist switches on gas lift optimization via the LIFTOPT keyword and then defines the artificial lift constraints for PLAT-A, using the GLIFTOPT keyword and sets the well gas lift parameters using the WLIFTOPT keyword.

```
--       ACTIVATE GAS LIFT OPTIMIZATION AND PARAMETERS
--
-- INCR   INCR   TSTEP    NEWTON
-- GAS    OIL    INTVAL   OPTN
LIFTOPT
12.5E3    5E-3   0.0      YES                                                  /
/
--
--       GROUP GAS LIFT OPTIMIZATION CONSTRAINTS
--
-- GRUP  MAX       MAX
-- NAME  GAS ALQ   TOTAL GAS
GLIFTOPT
PLAT-A    200E3    1*                                                         /
/
--
--       WELL GAS LIFT OPTIMIZATION PARAMETERS
--
-- WELL  OPTN   MAX    WEIGHT  MIN    GAS      OPTN
-- NAME  LIFT   LIFT   FACTOR  LIFT   FACTOR   LIMIT
WLIFTOPT
OP01     YES    150E3  1.01    -1.0                                           /
OP02     YES    150E3  1.01    -1.0                                           /
OPO3     YES    150E3  1.01    -1.0                                           /
OP04     YES    150E3  1.01    -1.0                                           /
OP05     YES    150E3  1.01    -1.0                                           /
/
```

Here the LIFTOPT keyword defines the maximum incremental gas lift gas quantity to be 12.5 x 103 m3, the minimum incremental oil gain per m3 of gas lift gas is set to 5.0 x 10-3 m3, the time step interval is set to zero to perform the gas optimization every time step, and finally the gas lift optimization will be performed NUPCOL Newton iterations for the time step.

The GLIFTOPT keyword sets the maximum amount of gas lift gas for PLAT-A to 200,000 m3 and there is no maximum limit for the total maximum amount of gas that the group can process. In addition, WLIFTOPT sets all five wells to have gas lift gas optimization implemented with a maximum gas lift gas value of 150,000 m3 per well, with equal weighting factors and all wells are supplied with sufficient gas lift gas to allow the wells to flow, (minimum lift set to a negative value) subject to the maximum allowed gas lift quantity for the well (150,000 m3).

12.3.304 WLIMTOL – DEFINE WELL CONSTRAINT TOLERANCE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

WLIMTOL keyword defines the tolerance to be used for various constraints applied to connections, completions (if connections have been lumped via the COMPLUMP keyword in the SCHEDULE section), wells, and groups, including the field group. See also the GCONTOL keyword in the SCHEDULE section that sets the tolerance parameters for groups.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.305 WLIST – DEFINE WELL LISTS (STATIC)

Description

WLIST declares a group of wells to belong to a named static well list. Wells in a named well list are treated as a group of wells for which the standard well keywords can be applied. For example, instead of repeating a well keyword for each well, the keyword only needs to have the named well list instead, for the action to be applied to all wells in the named well list. In general any well keyword that allows well name roots as a well name, for example, PROD*, can use a named well list.

No.	Name	Description	Default
1	WLIST	A character string of up to eight characters in length, enclosed in quotes, that defines the well list name for the WELNAMES declared by this record. Note the first character must be asterisk ("*") and the second character must be a letter, for example, *PROD.	None
2	ACTION	A defined character string that determines how the WELNAMES should be handled with respect to the named well list (WLIST). ACTION should be set to one of the following:: 1) ADD: Add the WELNAMES to an existing WLIST. 2) DEL; Delete WELNAMES from an existing WLIST. 3) MOV: WELNAMES from another existing named well list and ADD them to WLIST. 4) NEW: Define a new named well list and add the WELNAMES to WLIST.	
3-52	WELNAMES	A character string of up to eight characters in length that defines the well name that belongs to the named well list (WLIST). A total of 50 well names can be added to WLIST at a time. If additional wells are needed to added then use the ADD option of ACTION to add additional wells. Well names roots may all be used in WELNAMES as long as they are enclosed in quotes and end with an asterisk ("*"). In this case all wells that match the specification will be added to the list. For example, wells named OP01, OP02 and OP03, can be added as group by using "OP*" as the well name. Note that the well names must have been declared previously using the WELSPECS keyword in the SCHEDULE section, otherwise an error may occur	

Notes:

1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".

Table 12.109: WLIST Keyword Description

Note that named well list production data in the SUMMARY file is well dependent, that is, if the wells belonging to a named well list is changed through time, the SUMMARY data will be based on the wells in the named well list group at the end of the run. Thus, if there are three wells in a named well list called *PROD1 at the beginning of a run; OP01, OP02 and OP03, and during the run OP03 is moved to a well named list

called *PROD2, then the SUMMARY data for *PROD1will only contain the production data for wells OP01 and OP02 and *PROD2 will only contain the SUMMARY data for OP03 from the start to the end of the run.

Example

The following example defines two named well lists using the WLIST keyword.

```
--
--          WELL LIST SPECIFICATION
--
-- LIST     OPER     WELL NAME LIST
-- NAME
WLIST
'*BLK-1'    NEW      WEL-01M WEL-02M WEL-03M WEL-04M WEL-05M WEL-06M WEL-07M   /
'*BLK-1'    ADD      WEL-08M WEL-09M WEL-10M WEL-11M WEL-12M WEL-13M WEL-14M   /
'*BLK-1'    ADD      WEL-15M WEL-16M WEL-17M WEL-18M WEL-19M WEL-20M WEL-23M   /
'*BLK-1'    ADD      WEL-24M WEL-25M WEL-26M WEL-28M                           /

'*BLK-2'    NEW      WEL-03U WEL-05U WEL-06U WEL-10U WEL-11U WEL-13U WEL-14U   /
'*BLK-2'    ADD      WEL-15U WEL-16U WEL-17U WEL-18U WEL-19U WEL-25U WEL-27U   /
/
DATES
          1  JAN   2020  /
/
--
--       DEFINE WELL AND WELL CONNECTIONS FLOWING STATUS
--
--  WELL WELL    --LOCATION--  COMPLETION
--  NAME STAT     I   J    K  FIRST LAST
WELOPEN
'*BLK-1' OPEN                                        /
'*BLK-1' OPEN     0   0    0     0     0             /
/
DATES
          1  JAN   2021  /
          1  JLY   2021  /
          1  OCT   2021  /
/
--
--       DEFINE WELL AND WELL CONNECTIONS FLOWING STATUS
--
--  WELL WELL    --LOCATION--  COMPLETION
--  NAME STAT     I   J    K  FIRST LAST
WELOPEN
'*BLK-2' OPEN                                        /
'*BLK-2' OPEN     0   0    0     0     0             /
/
```

In this example the wells in named well list "*BLK-1" are opened on January 1, 2020 and wells in named well list '*BLK-2" are opened October 1, 2021.

12.3.306 WLISTARG – MODIFY WELL LIST TARGET AND CONSTRAINT VALUES (STATIC)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The WLISTARG keyword modifies the target and constraint values of both rates and pressures for wells previously defined in a well list by the WLIST or WLISTNAM keywords. WLISTARG is similar to the WELTARG keyword in it that allows for modifying targets and constraints without having to define all the variables on the well control keywords: WCONPROD, WCONHIST, WCONINJE, or WCONINJH keywords. Variables not changed by the WLISTARG keyword remain the same as those previously entered via the well control keywords or previously entered WLISTARG keywords. Note that the well must still be initially be fully defined using the WCONPROD or WCONINJE keywords. All the aforementioned keywords are described in the SCHEDULE section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WLIST	A character string of up to eight characters in length, enclosed in quotes, that defines the well list name declared by the WLIST keyword. Note the first character must be asterisk (“*”) and the second character must be a letter, for example, *PROD.			None

No.	Name	Description			Default
		Field	Metric	Laboratory	
2	TARGET	A defined character string that sets the item to be changed for the well the value of the item is set by item (3). 1) ORAT: reset the surface oil production rate value as defined by item (3). 2) WRAT: reset the surface water production rate value as defined by item (3). 3) GRAT: reset the surface gas production rate value as defined by item (3). 4) LRAT: reset the surface liquid (oil plus water) production rate value as defined by (3). 5) RESV: reset the in situ reservoir volume rate value as defined by (3). 6) BHP: reset the bottom-hole pressure value as defined by item (3). 7) THP: reset the tubing head pressure value for the well as defined by item (3). 8) VFP: reset the vertical lift performance table number as defined by (3). 9) LIFT: reset the artificial lift quantity for use with vertical lift performance tables. 10) GUID: reset the guide rate value for wells operating under group control. Note TARGET only defines the variable to be changed, it does not change how a well is controlled. For example, if a well is operating on ORAT control, as defined by the previously entered WCONPROD keyword, entering TARGET equal to LRAT with a value, changes the liquid constraint but the well still remains on ORAT control. Use the WELCNTL keyword in the SCHEDULE section to change the control mode of a well.			None
3	VALUE	A real positive vector of values that defines the value of the variable declared by TARGET for all the wells contained in WLIST. For example if there are four wells in WLIST then there must four real numbers for VALUE. The vector should be terminated by a "/" as indicated in the notes below.			
	Liquid	stb/d	sm^3/day	scc/hour	
	Gas	Mscf/d	sm3/day	scc/hour	
	Res Vol	rb/d	rm^3/day	rcc/hour	
	Pressure	psia	barsa	atma	
	VFP	dimensionless	dimensionless	dimensionless	
	LIFT	same as VFPPROD or VFPINJ	same as VFPPROD or VFPINJ	same as VFPPROD or VFPINJ	None

Notes:

1) The keyword is followed by any number records with each record terminated by a "/" and the keyword should be terminated by a "/".

Table 12.110: WLISTARG Keyword Description

If a well is currently a history matching well, then WLISELTARG should only be used to change a wells bottom-hole pressure limit, vertical flow performance table number or the artificial lift quantity.

See also the WELCNTL keyword, in the SCHEDULE section that can be used to reset the control mode, as well as a well's target and constraints of both rates and pressures.

Example

The following example defines two named well lists using the WLIST keyword.

```
--
--       WELL LIST SPECIFICATION
--
-- LIST  OPER     WELL NAME LIST
-- NAME
WLIST
'*BLK-1' NEW      WEL-01M WEL-02M WEL-03M                                  /
'*BLK-2' NEW      WEL-03U WEL-05U WEL-06U WEL-10U                          /
/
--
--       WELL PRODUCTION AND INJECTION TARGETS
--
--  WELL WELL   TARGET
--  NAME TARG   VALUE
WLISTARG
'*BLK-1' ORAT   2000.0  2000.00  2000.0                    /
'*BLK-2' ORAT   3000.0  3500.00  4000.0  2000.0            /
/
```

The wells in the '*BLK-1' well list are all given an oil rate of 2,000 stb/d and wells in the '*BLK-2' well list are given rates of 3,000, 3,500, 4,000 and 2,000 stb/d.

12.3.307 WLISTNAM – DEFINE WELL LISTS (WLISTARG)

Description

WLISTNAM declares a group of wells to belong to a named WLISTARG well list for use with the WLISTARG keyword. Only the WLISTARG keyword can be used with this type of well list, and therefore it is better to use the WLIST keyword instead, that defines a static well list but offers more flexibility than a WLISTNAM well list.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description	Default
1	WLIST	A character string of up to eight characters in length, enclosed in quotes, that defines the well list name for the WELLNAMES declared by this record. Note the first character must be asterisk ("*") and the second character must be a letter, for example, *PROD.	None
2-51	WELNAMES	A character string of up to eight characters in length that defines the well name that belongs to the named well list (WLIST). A total of 50 well names can be added to WLISTNAM at a time. If the first well name in the list is the default value ("*1"), then the list is first cleared of all wells, before adding the subsequent wells in WELLNAMES. Well names roots may all be used in WELLNAMES as long as they are enclosed in quotes and end with an asterisk ("*"). In this case all wells that match the specification will be added to the list. For example, wells named OP01, OP02 and OP03, can be added as group by using "OP*" as the well name. Note that the well names must have been declared previously using the WELSPECS keyword in the SCHEDULE section, otherwise an error may occur.	1*

Notes:

1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".

Table 12.111: WLISTNAM Keyword Description

Note that named well list production data in the SUMMARY file is well dependent, that is, if the wells belonging to a named well list is changed through time, the SUMMARY data will be based on the wells in the named well list group at the end of the run. Thus, if there are three wells in a named well list called *PROD1 at the beginning of a run; OP01, OP02 and OP03, and during the run OP03 is moved to a well named list called *PROD2, then the SUMMARY data for *PROD1 will only contain the production data for wells OP01 and OP02 and *PROD2 will only contain the SUMMARY data for OP03 from the start to the end of the run.

Example

The following example defines two named well lists using the WLISTNAM keyword.

```
--
--          WELL LIST SPECIFICATION
--
-- LIST     WELL NAME LIST
-- NAME
WLISTNAM
'*BLK-1'    WEL-01M WEL-02M WEL-03M WEL-04M WEL-05M WEL-06M WEL-07M            /
'*BLK-1'    WEL-08M WEL-09M WEL-10M WEL-11M WEL-12M WEL-13M WEL-14M            /
'*BLK-1'    WEL-15M WEL-16M WEL-17M WEL-18M WEL-19M WEL-20M WEL-23M            /
'*BLK-1'    WEL-24M WEL-25M WEL-26M WEL-28M                                    /

'*BLK-2'    1*      WEL-03U WEL-05U WEL-06U WEL-10U WEL-11U WEL-13U WEL-14U    /
'*BLK-2'            WEL-15U WEL-16U WEL-17U WEL-18U WEL-19U WEL-25U WEL-27U    /
/
```

Here well list "*BLK-1' contains 28 wells, that is wells WEL-01M to WEL-28M. For the '*BLK-2' well list all wells are first deleted due to the "1*" default value and then wells WEL-03U to WEL-27U are added to the list.

12.3.308 WMICP – DEFINE WATER INJECTION WELL'S MICROBIAL, OXYGEN, AND UREA CONCENTRATIONS

Description

The WMICP keyword defines a water injection well's microbial, growth, and cementation injection stream solutions, where the rate-limiting components are suspended microbes, oxygen, and urea concentrations respectively. These concentrations are used when the MICP keyword in the RUNSPEC section has been used to activate OPM Flow's Microbially Induced Calcite Precipitation model. See Landa-Marbán et al[335] and [336] for a description of the model.

Note the keyword should only be used for wells declared as water injection wells via the WCONINJE keyword in the RUNSPRC section.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WELNAME	A character string of up to eight characters in length that defines the well name for which the well connection data is being defined. Note that the well name (WELNAME) must have been declared previously using the WELSPECS keyword in the SCHEDULE section, otherwise an error may occur.			None
2	MICRCON	MICRCON is a real positive value that defines the microbial concentration of the well's injection stream.			
		lb/stb	kg/sm^3	gm/scc	None
3	OXYGCON	A real positive value that defines the oxygen concentration of the well's injection stream.			
		lb/stb	kg/sm3	gm/scc	None
4	UREACON	UREACON is a real positive value that defines the urea concentration of the well's injection stream.			None
		lb/stb	kg/sm^3	gm/scc	

Notes:

1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".

Table 12.112: WMICP Keyword Description

Water injection wells that are not declared via this keyword have their concentrations defaulted to zero.

[335] *Landa-Marbán, D., Tveit, S., Kumar, K., Gasda, S.E., 2021. Practical approaches to study microbially induced calcite precipitation at the eld scale. Int. J. Greenh. Gas Control 106, 103256. https://doi.org/10.1016/j.ijggc.2021.103256.*

[336] *Landa-Marbán, D., Kumar, K., Tveit, S., Gasda, S.E., 2021. Numerical studies of CO2 leakage remediation by micp-based plugging technology. In: Røkke, N.A. and Knuutila, H.K. (Eds) Short Papers from the 11th International Trondheim CCS conference, ISBN: 978-82-536-1714-5, 284-290.*

Example

```
--
--       DEFINE WATER INJECTION WELL MICROBIAL, OXYGEN, AND UREA CONCENTRATIONS
--
-- WELL  MICROBIAL  OXYGEN     UREA
-- NAME  MICRCON    OXYGCON    UREACON
--       --------   --------   --------
WMICP
WI01     0.01                              /
WI02     1*         0.04                   /
WI03     1*         1*         60.0        /
WI04     1*         0.04       60.0        /
/
```

Here the microbial concentration for well WI01 is set to 0.01, the oxygen concentration for well WI02 is set to 0.04, the urea concentration for well WI03 is set to 60, and the oxygen and urea concentrations for well WI04 are set to 0.04 and 60 respectively.

12.3.309 WNETCTRL – DEFINE WELL CONTROL FOR NETWORK CONTROL OPTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The WNETCNTL keyword sets a well's control mode that should remain fixed after each network balancing calculation, for when the either the Standard Network or the Extended Network options have been activated, and the well is part of a network. The keyword allows for a well's Tubing Head Pressure ("THP"), oil, gas, liquid, or water rate to be selected as fixed after each network balance calculation. Normally this should be the THP, and if the keyword is absent from the input deck then THP will be used as the default value. The Standard Network option is invoked if the GRUPTREE, GRUPNET, GNETINJE, GNETPUMP, etc. series of keywords have been used in the SCHEDULE section. Whereas, the Extended Network option is activated by the NETWORK keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.310 WNETDP – DEFINE WELL THP TO NETWORK PRESSURE DROP

Description

The WNETDP keyword allows for a constant pressure drop between a well's Tubing Head Pressure ("THP") and the well's connecting network node, for when the either the Standard Network or the Extended Network options have been activated, and the well is part of a network. For production wells in a production network, WNETDP is added to the well's connecting network node pressure to arrive at the well's THP value. Whereas for injection wells in an injection network, WNETDP is subtracted from the well's connecting network node pressure to arrive at the well's THP value. The Standard Network option is invoked if the GRUPTREE, GRUPNET, GNETINJE, GNETPUMP, etc. series of keywords have been used in the SCHEDULE section. The Extended Network option is activated by the NETWORK keyword in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.311 WORKLIM – DEFINE WELL WORKOVER TIME

Description

WORKLIM sets the numbers of days taken to complete a workover.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.312 WORKTHP – DEFINE WELL WORKOVER OPTIONS FOR THP KILLED WELLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The WORKTHP keyword defines workover options for when a well dies, that is unable to produce at the current operating conditions, when under Tubing Head Pressure ("THP") control. For example, if a well is producing to the high pressure separator and therefore has a high THP constraint, then the WORKTHP keyword can be used to switch the well to the lower pressure separator via re-setting the THP constraint.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.313 WPAVE – WELL BLOCK AVERAGE PRESSURE CALCULATION PARAMETERS FOR ALL WELLS

Description

The WPAVE keyword defines the method and parameters for calculating a well's block average pressures for all wells in the model. The resulting average pressure can be written out to the SUMMARY and RSM files in order to compare with field observed data via the WBP, WBP4, WBP5 and WBP9 vectors in the SUMMARY section.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WPAVE1	A real dimensionless value that defines the weighting factor between the inner block and the surrounding blocks used in the calculation of the connection factor weighted average pressure. If WPAVE1 is greater than or equal to zero and less than or equal to one, then the average pressure for each well connection is calculated based on this weighting factor. A value of zero indicates only the surrounding blocks should be used in the calculation; and a value of one indicates only the inner blocks should be used. If WPAVE1 is less than zero, then the average pressure for each well connection is weighted based on the pore volumes of the inner and surrounding blocks.			0.5
2	WPAVE2	A real dimensionless value greater than or equal to zero and less than or equal to one, that defines the weighting factor between the connection factor weighted average pressures and the pore volume weighted average pressures. If WPAVE2 is equal to one, then the average pressures are calculated based only on the connection factor weighted average pressures. If WPAVE2 is equal to zero, then average pressures are calculated based only on the pore volumes weighted average pressures.			1.0
3	WPAVE3	A defined character string that determines how the hydrostatic head calculation is performed in correcting the pressures to the BHP reference depth on the WELSPECS or WPAVEDEP keywords in the SCHEDULE section. WPAVE3 should be set to one of the following character strings: 1) WELL: the hydrostatic head is calculated using the density of the fluid in the wellbore at the well connections. 2) RES: he hydrostatic head is calculated using the density of the fluid in the reservoir with well connections and averaged over the connections. 3) NONE: no hydrostatic correction is applied to the pressures.			WELL

No.	Name	Description			Default
		Field	Metric	Laboratory	
4	WPAVE4	A defined character string that determines which connections should be used in the calculations, WPAVE4 should be set to one of the following character strings: 1) OPEN: only open connections and associated grid blocks should be used in the calculations. This option may result in pressure discontinuities if connections are opened and closed during the run. 2) ALL: all currently defined open and closed connections and associated grid blocks are used in the calculations. The pressure discontinuities issue mentioned above can be avoided with this option and defining all the well connections for a well at the beginning of the run. Only the OPEN option is currently supported by the simulator.			OPEN
Notes: 1) The keyword should be terminated by a "/".					

Table 12.113: WPAVE Keyword Description

The keyword is not applicable and should not be used with radial and spider grid geometries.

See also the WELSPECS keyword that defines a well and a well's bottom-hole pressure reference depth, the WPAVEDEP keyword that also defines a well's bottom-hole pressure reference depth, and the COMPDAT keyword to define a well's connections. All the aforementioned keywords are described in the SCHEDULE section.

Examples

The following example defines the default well block average pressure calculation parameters

```
--
--        DEFINE WELL BLOCK AVERAGE PRESSURE CALCULATION PARAMETERS
--
--        INNER    PORV    WELL   OPEN
--        OUTER    CONN    RES    ALL
WPAVE
          0.5      1.0     WELL   ALL                                    /
```

And the next example shows the parameters used in the Norne model.

```
--
--        DEFINE WELL BLOCK AVERAGE PRESSURE CALCULATION PARAMETERS
--
--        INNER    PORV    WELL   OPEN
--        OUTER    CONN    RES    ALL
WPAVE
          1*       0.0     WELL   ALL                                    /
```

Here only pore volume weighting is used instead of connection weighting.

12.3.314 WPAVEDEP – DEFINE WELL REFERENCE DEPTH FOR PRESSURE CALCULATIONS

Description

The WPAVEDEP keyword defines the reference depth to be used to calculate and report grid block average bottom-hole pressures for a well. This keyword can be used to override the values entered or defaulted on the WELPSPECS keyword in the SCHEDULE section. The simulator corrects the grid block calculated pressures to a well's reference depth using the hydrostatic well of the producing fluids.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WELNAME	A character string of up to eight characters in length that defines the well name for which the well and well connection status data is being defined. Note that the well name (WELNAME) must have been declared previously using the WELSPECS keyword in the SCHEDULE section, otherwise an error may occur.			None
2	BHPREF	A real value that defines the reference depth for reporting the bottom-hole pressure for the well. Ideally this value should be set to the midpoint of the perforations as defined by the COMPDAT keyword in the SCHEDULE section. If defaulted by 1* or set to a value less than or equal to zero, then the mid-point of shallowest connection defined by the COMPDAT keyword will be used.			Mid-point of shallowest connection defined by the COMPDAT keyword
		feet	m	cm	

Notes:

1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".

Table 12.114: WPAVDEP Keyword Description

See also the WELSPECS keyword that defines a well, the COMPDAT keyword to define a well's connections, and the WPAVE for defining how the average bottom-hole pressure should be calculated. All the aforementioned keywords are described in the SCHEDULE section.

Note

The keyword is normally used to reset a well's bottom-hole pressure depth to match the pressure gauge depth for when observed pressure is available, for example when conducting a history match for a well test, or when attempting to match static bottom-hole surveys conducted on a well.

Example

The following example illustrates how to set the bottom-hole reference depth for wells completed in different reservoirs that have different datum depths. Here it is assumed that all wells in a reservoir A have RES-A as part of their well name, and similarly for reservoirs B and C.

```
--
-- WELL SPECIFICATION DATA
--
-- WELL      GROUP       LOCATION  BHP    PHASE  DRAIN  INFLOW  SHUT  CROSS  PRESS
-- NAME      NAME         I    J   DEPTH  FLUID  AREA   EQUA.   IN    FLOW   TABLE
WELSPECS
RES-AOP1  PLATFORM     14   13   1*      OIL   1*     STD     OPEN   NO    1*  /
RES-AOP2  PLATFORM     17   16   1*      OIL   1*     STD     OPEN   NO    1*  /
RES-AOP3  PLATFORM     21   19   1*      OIL   1*     STD     OPEN   NO    1*  /
RES-BOP4  PLATFORM     28   96   1*      OIL   1*     STD     OPEN   NO    1*  /
RES-BOP5  PLATFORM     34   89   1*      OIL   1*     STD     OPEN   NO    1*  /
RES-COP6  PLATFORM    128   52   1*      OIL   1*     STD     OPEN   NO    1*  /
RES-COP7  PLATFORM    134   56   1*      OIL   1*     STD     OPEN   NO    1*  /
RES-COP8  PLATFORM    138   50   1*      OIL   1*     STD     OPEN   NO    1*  /
RES-COP9  PLATFORM    120   52   1*      OIL   1*     STD     OPEN   NO    1*  /
/
--
--       DEFINE WELL REFERENCE DEPTH FOR PRESSURE CALCULATIONS
--
-- WELL  REF
-- NAME  DEPTH
-- ----  ------
WPAVEDEP
'RES-A*' 3100.0                                                              /
'RES-B*' 3300.0                                                              /
'RES-C*' 5909.0                                                              /
/
```

In the example the all wells dedicated to RES-A will have their bottom-hole reference depth set to 3,000 ft. TVDSS, RES-B wells to 3,300 ft. TVDSS and well RES-C wells to 5909 ft. TVDSS.

12.3.315 WPIMULT – DEFINE WELL CONNECTION MULTIPLIERS

Description

The WPIMULT keyword defines a well connection multiplier factor that scales the existing well connection values. The resulting effect is to scale the well's productivity at the reporting time step the keyword is entered.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WELNAME	A character string of up to eight characters in length that defines the well name for which the well and well connection status data is being defined. Note that the well name (WELNAME) must have been declared previously using the WELSPECS keyword in the SCHEDULE section, otherwise an error may occur.			None
2	PIMULT	A real positive value that will be used to scale the well connection factors defined by I, J, K, K1 and K2 below.			1.0
3	I	An integer less than or equal to NX that defines the connection locations in the I-direction.			1*
4	J	An integer less than or equal to NY that defines the connection locations in the J-direction.			1*
5	K	An integer less than or equal to NZ that defines the connection locations in the K-direction.			1*
6	K1	An integer less than or equal to NZ that defines the UPPER completion location in the K-direction. Connections are lumped into completions via the COMPLUMP keyword, and K1 refers to the first completion number, as defined by the COMPLUMP keyword, and all the connections contained within the K1 completion.			1*
7	K2	An integer less than or equal to NZ that defines the LOWER completion location in the K-direction. Connections are lumped into completions via the COMPLUMP keyword, and K2 refers to the last completion number, as defined by the COMPLUMP keyword, and all the connections contained within the K2 completion.			1*

Notes:

1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".

Table 12.115: WPIMULT Keyword Description

If variables I, J K, K1 and K2 are all defaulted with a negative value or 1*, then PIMULT is applied to all the well connections in the well. If variables I, J K, K1 and K2 are set to zero (meaning any or all values), or a positive value then PIMULT is applied to the defined connections. The defined connections are those with the I, J, K variables in the specified location and a completion number in the range specified by K1 and K2. Note, that PIMULT is applied at the time the WPIMULT keyword is entered and is cumulative if applied to the same well connections, provided there are intervening report time steps between consecutive WPIMULT keywords. Consequently, if there are no intervening report time steps between consecutive WPIMULT keywords utilizing the same well connections, then only the last set is applied.

See also the WELPI keyword to set a well's productivity or injectivity index at the time the keyword is activated, and also the PIMULTAB keyword that defines productivity index multiplier versus water cut tables that are used to scaled a well's connection factors based on a wells connection current producing water cut. Both keyword are documented in the SCHEDULE section.

Example

The following example defines three vertical oil wells using the WELSPECS keyword and their associated connection data.

```
--
-- WELL SPECIFICATION DATA
--
-- WELL      GROUP       LOCATION  BHP    PHASE  DRAIN  INFLOW  SHUT  CROSS  PRESS
-- NAME      NAME         I    J   DEPTH  FLUID  AREA   EQUA.   IN    FLOW   TABLE
WELSPECS
OP01       PLATFORM      14   13   1*      OIL   1*     STD     OPEN   NO    1*  /
OP02       PLATFORM      28   96   1*      OIL   1*     STD     OPEN   NO    1*  /
OP03       PLATFORM     128   56   1*      OIL   1*     STD     OPEN   NO    1*  /
/
--
--        WELL PRODUCTION WELL CONTROLS
--
-- WELL  OPEN/  CNTL   OIL    WAT    GAS    LIQ    RES    BHP    THP   VFP    VFP
-- NAME  SHUT   MODE   RATE   RATE   RATE   RATE   RATE   PRES   PRES  TABLE  ALFQ
WCONPROD
'*'       SHUT   GRUP   1*     1*     1*     1*     1*     200.0                 /
/
--
--        WELL CONNECTION DATA
--
-- WELL   --- LOCATION ---   OPEN   SAT   CONN   WELL   KH    SKIN   D     DIR
-- NAME    II   JJ  K1  K2   SHUT   TAB   FACT   DIA    FACT  FACT   FACT  PEN
COMPDAT
OP01       1*  1*   1  10   OPEN   1*    1*    0.708   1*    0.0    1*    'Z' /
OP01       1*  1*  15  30   OPEN   1*    1*    0.708   1*    0.0    1*    'Z' /
OP01       1*  1*  35  90   OPEN   1*    1*    0.708   1*    0.0    1*    'Z' /
OP02       1*  1*   1  10   OPEN   1*    1*    0.708   1*    0.0    1*    'Z' /
OP03       1*  1*  35  90   OPEN   1*    1*    0.708   1*    0.0    1*    'Z' /
/
--        ASSIGN WELL CONNECTIONS TO COMPLETIONS
--
-- WELL   --- LOCATION ---   COMPL
-- NAME    II   JJ  K1  K2   NO.
COMPLUMP
OP03       1*  1*  35  45    1                          / COMPLETION NO. 01
OP03       1*  1*  50  90    2                          / COMPLETION NO. 02
/
```

```
--
--       DEFINE WELL CONNECTION MULTIPLIERS
--
-- WELL  PI     --LOCATION--  COMPLETION
-- NAME  MULT     I   J    K  FIRST LAST
WPIMULT
OP01     1.250    1*  1*   1*   1*    1*                  /
OP02     0.750    1*  1*   10   1*    1*                  /
OP03     1.100    1*  1*   1*   1     2                   /
/
```

In this example the WPIMULT scales the well productivity of well OP01 by 1.25, and scales all the well connection factors in layer 10 only by 0.75 for well OP02. For well OP03, WPIMULT scales all the connections in completions one and two by 1.100.

12.3.316 WPIMULTL – DEFINE WELL CONNECTION MULTIPLIERS (LGR)

Description

The WPIMULTL keyword defines a well connection multiplier factor that scales the existing well connection values, for a well in a Local Grid Refinement ("LGR"). The resulting effect is scale the well's productivity at the reporting time step the keyword is entered.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WELNAME	A character string of up to eight characters in length that defines the well name for which the well and well connection status data is being defined. Note that the well name (WELNAME) must have been declared previously using the WELSPECS keyword in the SCHEDULE section, otherwise an error may occur.			None
2	PIMULT	A real positive value that will be used to scale the well connection factors defined by I, J, K, K1 and K2 below.			1.0
3	LGRNAME	A character string of up to eight characters in length that defines the LGR name for which the well LGR connection multiplier factor (PIMULT) is being defined. Note that the well name (LGRNAME) must have been declared previously using the WELSPECL keyword in the SCHEDULE section, otherwise an error may occur. If defaulted with 1* the LGR on the WELSPECL keyword will be utilized.			Defined
3	I	An integer less than or equal to NX that defines the connection location in the I-direction.			1*
4	J	An integer less than or equal to NY that defines the connection location in the J-direction.			1*
5	K	An integer less than or equal to NZ that defines the connection location in the K-direction.			1*
6	K1	An integer less than or equal to NZ that defines the UPPER completion location in the K-direction. Connections are lumped into completions via the COMPLUMP keyword, and K1 refers to the first completion number, as defined by the COMPLUMP keyword, and all the connections contained within the K1 completion			1*
7	K2	An integer less than or equal to NZ that defines the LOWER completion location in the K-direction. Connections are lumped into completions via the COMPLUMP keyword, and K2 refers to the last completion number, as defined by the COMPLUMP keyword, and all the connections contained within the K2 completion			1*

No.	Name	Description			Default
		Field	Metric	Laboratory	
Notes: 1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".					

Table 12.116:WPIMULT Keyword Description

If variables I, J K, K1 and K2 are all defaulted with a negative value or 1*, then PIMULT is applied to all the well connections in the well. If variables I, J K, K1 and K2 are set to zero (meaning any or all values), or a positive value then PIMULT is applied to the defined connections. The defined connections are those with the I, J, K variables in the specified location and a completion number in the range specified by K1 and K2.

Note, that PIMULT is applied at the time the WPIMULT keyword is entered and is cumulative if applied to the same well connections, provided there are intervening report time steps between consecutive WPIMULT keywords. Consequently, if there are no intervening report time steps between consecutive WPIMULT keywords utilizing the same well connections, then only the last set is applied.

See also the PIMULTAB keyword that defines productivity index multiplier versus water cut tables that are used to scaled a well's connection factors based on a wells connection current producing water cut. The keyword is documented in the SCHEDULE section.

Example

The following example defines two vertical oil wells using the WELSPECS keyword and their associated connection data.

```
--
--       WELL LGR SPECIFICATION DATA
--
--       WELL GROUP  LGR    -LOCATION- BHP    PHASE DRAIN INFLOW SHUT CROSS PVT
--       NAME NAME   NAME     I    J   DEPTH  FLUID AREA  EQUA.  IN   FLOW  TABLE
WELSPECL
         OP01  PLAT OP01LGR  14   13   1*     OIL   1*    STD    SHUT  NO   1*  /
         OP02  PLAT OP02LGR  28   96   1*     OIL   1*    STD    SHUT  NO   1*  /
/
--
--       WELL LGR CONNECTION DATA
--
--       WELL  LGR   ---LOCATION---   OPEN   SAT CONN  WELL KH   SKIN  D    DIR
--       NAME  NAME   II  JJ  K1  K2  SHUT   TAB FACT  DIA  FACT FACT  FACT PEN
COMPDATL
         OP01 OP01LGR 1*  1*  20  56  OPEN   1*  1*  0.708   1*   1*   1*   Z /
         OP01 OP01LGR 1*  1*  75 100  SHUT   1*  1*  0.708   1*   1*   1*   Z /
        OP02 OP02LGR 1*  1*  75 100  OPEN   1*  1*  0.708   1*   1*   1*   Z /
         OP03 OP02LGR 1*  1*  75 100  OPEN   1*  1*  0.708   1*   1*   1*   Z /
/
--
--       ASSIGN WELL CONNECTIONS TO COMPLETIONS
--
--       WELL    LGR      --- LOCATION ---  COMPL
--       NAME    NAME      II  JJ  K1  K2   NO.
COMPLMPL
            OP03    OP02LGR   1*  1*  75  85    1          / COMPLETION NO. 01
            OP03    OP21LGR   1*  1*  86 100    2          / COMPLETION NO. 02
/
```

```
--
--       DEFINE WELL CONNECTION MULTIPLIERS
--
-- WELL  PI     LGR     --LOCATION--  COMPLETION
-- NAME  MULT   NAME      I   J    K  FIRST LAST
WPIMULTL
OP01     1.250  OP01LGR   1*  1*   1*    1*    1*          /
OP02     0.750  OP01LGR   1*  1*   10    1*    1*          /
OP03     1.100  OP02LGR   1*  1*   1*    1     2           /
/
```

In this example the WPIMULT scales the well productivity of well OP01 by 1.25, scales all the well connection factors in layer 10 only by 0.75 for well OP02, and for OP03, scales all the connections in completions one and two by 1.100.

12.3.317 WPITAB - ASSIGN WELL PRODUCTIVITY INDEX VERSUS WATER CUT TABLES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The WPITAB keyword assigns the well productivity index multiplier versus water cut tables, that are used to scaled a well's connection factors based on the connection's current producing water cut, to a well. The tables are defined via the PIMULTAB keyword in the SCHEDULE section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WELNAME	A character string of up to eight characters in length that defines the well name for which the well productivity index multiplier versus water cut table, PIMULTAB, is being assigned. Note that the well name (WELNAME) must have been declared previously using the WELSPECS keyword in the SCHEDULE section, otherwise an error may occur.			None
2	PIMULTAB	A positive integer value that defines the corresponding PIMULTAB table to be allocated to the well. A value less than or equal to zero means that no PIMULTAB table is allocated to the well			0
Notes: 1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".					

Table 12.117: WPITAB Keyword Description

See also the PIMULTAB keyword that defines productivity index multiplier versus water cut tables and also the WPIMULT keyword that scales a well's productivity index by a constant value, both of which are in the SCHEDULE section.

Example

Given NTPIMT equals two on the PIMTDIMS keyword in the RUNSPEC section, then:

```
--
--       ASSIGN WELL PRODUCTIVITY INDEX VS WATER CUT TABLE
--
-- WELL  PI
-- NAME  TABLE
WPITAB
OP01     1                                                    /
OP02     1                                                    /
OP03     2                                                    /
/
```

Assigns PIMULTAB table one to wells OP01 and OP02 and table two to OP03.

12.3.318 WPLUG – DEFINE WELL PLUG BACK LENGTH

Description

Various keywords in the SCHEDULE section (WECON, GECON etc.) allow for a well to be automatically plugged back if the well violates a constraint, that is to close existing perforations (well connections). For example if the water cut exceeds 90%, then plug back the well. The WPLUG keyword defines for automatic plug backs the length of the perforations (length of connections) to be closed each time an automatic plug back is performed, together with various options on how the workover should be performed, top down, bottom up, etc.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.319 WPMITAB - ASSIGN WELL POLYMER MOLECULAR MODEL INJECTION TABLES

Description

The WPMITAB keyword assigns the well polymer molecular injection tables to water injection wells in OPM Flow's Polymer Molecular Weight Transport option, that uses the polymer molecular weight in calculating the polymer viscosity, as well as accounting for formation damage due to the water and polymer injection, by adjusting the wellbore skin pressure. This keyword should only be used if the POLYMER and POLYMW keywords in the RUNSPEC section are also activated. The keyword assigns the PLYMWINJ tables that are defined via the PLYMWINJ keyword in the PROPS section.

Note

This is an OPM Flow specific keyword that employs an alternative polymer flood model based on a Polymer Molecular Weight Transport equation, that is not available in the commercial simulator.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WELNAME	A character string of up to eight characters in length, that defines the water injection well name, for which the well polymer molecular injection table, PLYMWINJ, is to be assigned. Note that the well name (WELNAME) must have been declared previously using the WELSPECS keyword in the SCHEDULE section, otherwise an error may occur.			None
2	PLYMWINJ	A positive integer value that defines the corresponding PLYMWINJ table to be allocated to the water injection well. A value less than or equal to zero means that no PLYMWIN table is allocated to the well			0

Notes:

1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".

Table 12.118: WPMITAB Keyword Description

See also the PLYMWINJ keyword in the PROPS section, that describe the relationship of the injected polymer molecular weight as a function of polymer throughput and polymer velocity, for the simulator's Polymer Molecular Weight Transport option. As wells as the SKPRWAT, SKPRPOLY, and PLYVMH keywords, also in the PROPS section, that are the additional keywords required for the Polymer Molecular Weight Transport option.

The WSKPTAB keyword in the SCHEDULE section may be used to assign the SKPRWAT and SKPRPOL tables to water injections wells, that enable the calculation of the wellbore skin pressure based on the fluids being injected.

Note that the standard polymer property data keywords: PLYROCK, PLYADS, PLYMAX, etc., are still required to fully describe the polymer fluid.

Example

Given NTPMWINJ equals two on the PINTDIMS keyword in the RUNSPEC section, then:

```
--
--       ASSIGN WELL POLYMER MOLECULAR MODEL INJECTION TABLES
--
-- WELL  PLYMWINJ
-- NAME  TABLE
WPMITAB
WI01     1                                                    /
WI02     1                                                    /
WI03     2                                                    /
/
```

Assigns PLYMWINJ table one to wells WI01 and WI02 and table two to WI03.

12.3.320 WPOLYMER - DEFINE WATER INJECTION WELL POLYMER AND SALT CONCENTRATIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The WPOLYMER keyword defines a water injection well's polymer and salt injection stream concentrations that are to be used for when the polymer and salt options have been activated by the POLYMER and BRINE keywords in the RUNSPEC section.

Note that if the Brine option has not be activated by the BRINE keyword in the RUNSPEC section, then the salt concentrations in the third column are ignored. Secondly, if the brine phase is declared but the polymer phase has not been made active, then the WSALT keyword in the SCHEDULE section can be used to set the salt concentration.

Currently the Brine option is not implemented in OPM Flow and therefore both the SALTCON and GRPSALT variables on this keyword are ignored.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WELNAME	A character string of up to eight characters in length that defines the well name for which the well connection data is being defined. Note that the well name (WELNAME) must have been declared previously using the WELSPECS keyword in the SCHEDULE section, otherwise an error may occur.			None
2	POLCON	A real positive value that defines the polymer concentration of the well's injection stream.			
		lb/stb	kg/sm^3	gm/scc	None
3	SALTCON	A real positive value that defines the salt concentration of the well's injection stream. This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.			
		lb/stb	kg/sm3	gm/scc	None
4	GRPPOL	A character string of up to eight characters in length that defines the group name for which the group's produced polymer concentration should be used instead of the well's POLCON value stated on this keyword.			None
5	GRPSALT	A character string of up to eight characters in length that defines the group name for which the group's produced salt concentration should be used instead of the well's SALTCON value stated on this keyword. This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.			None

Notes:

1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".

Table 12.119: WPOLYMER Keyword Description

Water injection wells that are not declared via this keyword have their concentrations defaulted to zero.

See also the GCONPROD and GCONINJE keywords to define a group's production and injection targets and constraints, and the WCONINJE keyword to define an injection well's targets and constraints. All the aforementioned keywords are described in the SCHEDULE section.

Example

The following example defines the polymer and salt injection stream concentrations for three water injection wells for when the polymer option has been activated by the POLYMER keyword in the RUNSPEC section.

```
--
--       DEFINE WATER INJECTION WELL POLYMER AND SALT CONCENTRATIONS
--
-- WELL  POLYMER    SALT       POLYMER    SALT
-- NAME  POLCON     SALTCON    GROUP      GROUP
--       -------    --------   --------   --------
WPOLYMER
WI01     0.2500                                    /
WI02     1*         1*         GRPINJ1              /
WI03     0.2500     1*         GRPINJ1              /
/
```

The polymer concentration for well WI01 is set to 0.25 and the stated polymer concentration for well WI02 will be ignored, as both WI02 and WI03 will re-inject the produced polymer from the GRPINJ1 group.

12.3.321 WPOLYRED – DEFINE WELL POLYMER-WATER VISCOSITY REDUCTION FACTOR

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The WPOLYRED keyword defines the polymer-water reduction factor for injection wells, for when the polymer phase has been activated by the POLYMER keyword in the RUNSPEC section. WPOLYRED should be set to a value greater than or equal to zero and less than or equal to one that determines the injection mixture's viscosity. A value of zero indicates for pure water injection and a value of one will use the simulator's valuated mixture viscosity. A value between zero and one will use an interpolated mixture viscosity.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.322 WREGROUP – AUTOMATIC RE-ASSIGNMENT OF WELLS TO GROUPS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

WREGROUP defines the criteria to automatically re-assign wells to various other groups. This can be used, for example, to move wells on THP control flowing through a high pressure separator group to a low pressure separator group in order for the wells to be under different group controls for low pressure wells.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.323 WRFT – ACTIVATE WELL RFT REPORTING TO THE RFT FILE

Description

This keyword activates reporting of a well's pressure and saturation profile versus depth for the connected grid blocks, to the RFT file for the requested wells at the time the keyword is activated. Data written out by OPM Flow is used to match the field measured data collected from a Repeat Formation Tester ("RFT") tool.

The RFT tool is an open hole device which is an updated version of the Formation Interval Tester ("FIT"), both of which are run on wire line. Both tools take multiple pressure readings (at various depths) thus enabling a pressure depth profile to be obtained from the formation, and, in addition, they can also take fluid samples from the formation. The latest tool available from Schlumberger is the Modular Formation Dynamics Tester ("MDT"), which, as its name suggests, is a modular tool that can be assembled in different configurations depending on what are the objectives for running the tool. Note other vendors have similar wire line logging tools with alternative names for the tools. Throughout this section the term RFT applies to all tools that measure a pressure profile versus depth (RFT/FIT/MDT etc.).

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WELNAME	A columnar vector of character strings of up to eight characters in length for each item, that defines the well name for which the RFT data should be written to the RFT file. Note that the WELNAME must have been declared previously using the WELSPECS keyword in the SCHEDULE section, otherwise an error may occur. If the WELNAME is left blank then the data is written out for all wells at the time they are first opened to flow. If the WELNAME is given, then the RFT data for the well at the time step the keyword is invoked is written out.			None
Notes: 1) The keyword is followed by any number of records. 2) Each record is terminated by a "/" and the keyword should be terminated by a "/".					

Table 12.120: WRFT Keyword Description

See also the WRFTPLT keyword in the SCHEDULE section that has more flexible reporting options.

Examples

The first example activates RFT reporting for all wells at the time a well is first opened to flow:

```
--
--       ACTIVATE WELL RFT REPORTING TO THE RFT FILE
--
-- WELL
-- NAME
WRFT
/
```

Ideally, this version of the keyword should be place at the beginning of the SCHEDULE section to obtain the data for the wells in the run before they are opened up through time.

The next example shows how to use the keyword to request the output for several wells at different reporting time steps.

```
-- ------------------------------------------------------------------------
-- 01 JAN 2000 START OF SCHEDULE SECTION
-- ------------------------------------------------------------------------
DATES
15 JAN 2000 /
/
--
--       WELL HISTORICAL PRODUCTION CONTROLS
--
-- WELL  OPEN/  CNTL   OIL    WAT    GAS    VFP    VFP   THP   BHP
-- NAME  SHUT   MODE   RATE   RATE   RATE   TABLE  ALFQ  PRES  PRES
WCONHIST
OP01     OPEN   ORAT  15.5E3  0.0    1550   10      1*   900.0 1*     /
OP02     SHUT                                                       /
/
--
--       ACTIVATE WELL RFT REPORTING TO THE RFT FILE
--
-- WELL
-- NAME
WRFT
OP01                                                                 /
OP02                                                                 /
/
DATES
01 FEB 2000 /
/
--
--       WELL HISTORICAL PRODUCTION CONTROLS
--
-- WELL  OPEN/  CNTL   OIL    WAT    GAS    VFP    VFP   THP   BHP
-- NAME  SHUT   MODE   RATE   RATE   RATE   TABLE  ALFQ  PRES  PRES
WCONHIST
OP01     OPEN   ORAT  15.5E3  0.0    1550   10      1*   900.0 1*     /
OP02     SHUT                                                       /
/
--
--       ACTIVATE WELL RFT REPORTING TO THE RFT FILE
--
-- WELL
-- NAME
WRFT
OP01                                                                 /
OP02                                                                 /
/
DATES
01 MAR 2000 /
/
--
--       WELL HISTORICAL PRODUCTION CONTROLS
--
-- WELL  OPEN/  CNTL   OIL    WAT    GAS    VFP    VFP   THP   BHP
-- NAME  SHUT   MODE   RATE   RATE   RATE   TABLE  ALFQ  PRES  PRES
WCONHIST
OP01     OPEN   ORAT  15.5E3  0.0    1550   10      1*   900.0 1*     /
OP02     OPEN   ORAT  10.5E3  0.0    1000   10      1*   900.0 1*     /
/
```

In this example, both well's have their RFT written out on February 1 and March 1 2000.

12.3.324 WRFTPLT – ACTIVATE WELL RFT AND PLT REPORTING TO THE RFT FILE

Description

This keyword activates reporting of a well's depth pressure and fluid rates profile to the RFT file for the requested wells at the time the keyword is activated. Data written out by the simulator is used to match the field measured data collected from both the Repeat Formation Tester ("RFT") tool and various Production Logging Tools ("PLT").

See the WRFT keyword in the SCHEDULE section for a brief description of the RFT data set. This keyword also actives the writing out of each well connection's fluid rates, connection factors and KH data, etc., as the PLT data. The PLT data is used to compare with measured data from wire line production logging tools.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WELNAME	A columnar vector of character strings of up to eight characters in length for each item, that defines the well name for which the RFT data should be written to the RFT file. Note that the WELNAME must have been declared previously using the WELSPECS keyword in the SCHEDULE section, otherwise an error may occur. If the WELNAME is left blank then the data is written out for all wells at the time they are first opened to flow. If the WELNAME is given, then the RFT data for the well at the time step the keyword is invoked is written out.			None
2	RFT	A defined character string that sets the RFT data set output options and should be set to one of the following character strings. 1) NO: do not write RFT data for the well. 2) YES: write out the RFT data at the current reporting time step. 3) REPT: write out the RFT data at the current reporting time step and all subsequent reporting time steps. 4) TIMESTEP: write out the RFT data at the current reporting time step and all subsequent time steps. 5) FOPN: write out the RFT data at the current reporting time step for the well if it is opened, otherwise write the RFT data out the first time the named well is opened.			NO
3	PLT	A defined character string that sets the PLT data set output options and should be set to one of the following character strings. 1) NO: do not write PLT data for the well. 2) YES: write out the PLT data at the current reporting time step. 3) REPT: write out the PLT data at the current reporting time step and all subsequent reporting time steps. 4) TIMESTEP: write out the PLT data at the current reporting time step and all subsequent time steps.			NO

No.	Name	Description			Default
		Field	Metric	Laboratory	
4	MULTISEG	A defined character string that sets the output options for multi-segment wells, that is the flow rates and pressures through each well segment, and should be set to one of the following character strings. 1) NO: do not write multi-segment well data for the well. 2) YES: write out the multi-segment well data at the current reporting time step. 3) REPT: write out the multi-segment well data at the current reporting time step and all subsequent reporting time steps. 4) TIMESTEP: write out the multi-segment well data at the current reporting time step and all subsequent time steps. Note the commercial simulator also uses MULTISEG to control the output of "rivers" for when the RIVERS Model has been enabled via the RIVRDIMS keyword in the RUNSPEC section. OPM Flow does not support the RIVERS Model.			NO
Notes: 1) The keyword is followed by any number of records terminated by a "/" and the keyword should be terminated by a "/".					

Table 12.121: WRFTPLT Keyword Description

See also the WRFT keyword in the SCHEDULE section that has less flexible reporting options.

Examples

The first example activates RFT output at the current reporting time step for all the wells that are opened to flow, otherwise the RFT data is written out the first time a well is opened.

```
--
--        WELL RFT, PLT AND SEGMENT DATA
--
-- WELL   RFT    PLT    SEGMENT
-- NAME   DATA   DATA   DATA
WRFTPLT
'*'       FOPN                                                    /
/
```

The next example writes out the RFT and PLT data for two wells at the current reporting time step.

```
--
--        WELL RFT, PLT AND SEGMENT DATA
--
-- WELL   RFT    PLT    SEGMENT
-- NAME   DATA   DATA   DATA
WRFTPLT
OP01      YES    YES                                             /
OP02      YES    YES                                             /
/
```

The final example is shown below:

```
--
--       WELL RFT, PLT AND SEGMENT DATA
--
-- WELL  RFT   PLT   SEGMENT
-- NAME  DATA  DATA  DATA
WRFTPLT
OP01     REPT  NO                                                    /
OP02     NO    YES                                                   /
/
```

In this case the RFT data for well OP01 is written out at the current reporting time step and all subsequent reporting time steps. For well OP02, no RFT is written out but the PLT data is written out for the current report time step only.

12.3.325 WSALT - DEFINE WATER INJECTION WELL SALT CONCENTRATIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The WSALT keyword defines a water injection well's salt injection stream concentration that is to be used for when the salt option has been activated by the BRINE keywords in the RUNSPEC section. Note that if the Polymer option has also been activated by the POLYMER keyword in the RUNSPEC section, then the WPOLYMER keyword in the SCHEDULE section should be used to enter both the polymer and salt concentrations.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WELNAME	A character string of up to eight characters in length that defines the well name for which the injection salt concentrations are being defined. Note that the well name (WELNAME) must have been declared previously using the WELSPECS keyword in the SCHEDULE section, otherwise an error may occur.			None
2	SALTCON	A real positive vector of values that defines the salt concentration of the well's injection stream and consists of: 1) If the Standard Brine model has been invoked by the BRINE keyword, then SALTCON consist of one value representing the injection salt concentration; 2) If OPM Flow's Water Vaporization and Salt Precipitation models have been activated by the VAPWAT and PRECSALT keywords in the RUNSPEC section, then SALTCON consist of one value representing the injection salt concentration; or, 3) If the Multi-Component Brine option has been activated by the BRINE and ECLMC keywords in the RUNSPEC section, then SALTCON consists of a vector of values representing the salt concentration of each brine within the injected brine mixture. Only options (1) and (2) are currently supported. Note if SALTCON is defaulted (1*) then the well's salt concentration will be use the well's group salt concentration.			
		lb/stb	kg/sm3	gm/scc	1*

Notes:

1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".

Table 12.122: WSALT Keyword Description

Water injection wells that are not declared via this keyword have their concentrations defaulted to zero.

See also the GCONPROD and GCONINJE keywords to define a group's production and injection targets and constraints, and the WCONINJE keyword to define an injection well's targets and constraints. All the aforementioned keywords are described in the SCHEDULE section.

Examples

The following example defines the salt injection stream concentration for three water injection wells for when the brine phase has been activated by the BRINE keyword in the RUNSPEC section.

```
--
--       DEFINE WATER INJECTION WELL SALT CONCENTRATIONS (STANDARD)
--
-- WELL  SALT-1     SALT-2     SALT-3     SALT-4
-- NAME  SALTCON    SALTCON    SALTCON    SALTCON
--       -------    -------    -------    -------
WSALT
WI01     0.2500                                    /
WI02     1*                                        /
WI03     0.2500                                    /
/
```

The salt concentration for both well WI01 and WI03 is set to 0.25, and for well WI02 the salt concentration will be taken from the well's group salt concentration.

The next example is based on using the Multi-Component Brine option, that is the BRINE and ECLMC keywords have been used in the RUNSPEC section, and assuming three salts.

```
--
--       DEFINE WATER INJECTION WELL SALT CONCENTRATIONS (MULTIPLE)
--
-- WELL  SALT-1     SALT-2     SALT-3     SALT-4
-- NAME  SALTCON    SALTCON    SALTCON    SALTCON
--       -------    -------    -------    -------
WSALT
WI01     0.1500     0.0500     0.0500                /
WI02     0.1500     0.0500     0.0500                /
WI03     0.2000     0.0500     0.0600                /
/
```

Here the salt concentrations for both well WI01 and WI02 are set to 0.1500, 0.0500, 0.0500 for the three salts and for well WI03 the salt concentrations are 0.2000, 0.0500 and 0.0600.

Note that OPM Flow does not currently support the Multi-Component brine model.

12.3.326 WSCCLEAN – WELL DEPOSITED SCALE ADJUSTMENT

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The WSCCLEAN keyword adjusts the amount of scale currently accumulated around a well's well connections for wells located in the global grid. For example, if a workover has been performed on a well to remove (or reduce) the deposited scale over the perforations, then this keyword can be used to implement the effects of the workover. Scale deposits reduce the productivity of well and this relationship is defined in the SCDPTAB and SCDATAB keywords in SCHEDULE section. The tables are allocated to a well via the WSCTAB keyword, which is also in the SCHEDULE section. Note that the Scale Deposition option must have been activated by declaring the dimensions of the scaling deposition tables using the SCDPDIMS keyword in the RUNSPEC section.

See also the WSSCLENL keyword in the SCHEDULE section that performs similar functionality for wells located in a Local Grid Refinement.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.327 WSCCLENL – WELL DEPOSITED SCALE ADJUSTMENT (LGR)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The WSCCLENL keyword adjusts the amount of scale currently accumulated around a well's well connections for wells located in a Local Grid Refinement (LGR). For example, if a workover has been performed on a well to remove (or reduce) the deposited scale over the perforations, then this keyword can be used to implement the effects of the workover. Scale deposits reduce the productivity of well and this relationship is defined in the SCDPTAB and SCDATAB keywords in SCHEDULE section. The tables are allocated to a well via the WSCTAB keyword, which is also in the SCHEDULE section. Note that the Scale Deposition option must have been activated by declaring the dimensions of the scaling deposition tables using the SCDPDIMS keyword in the RUNSPEC section.

See also the WSSCLEAN keyword in the SCHEDULE section that performs similar functionality for wells located in the global grid.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.328 WSCTAB – ASSIGN WELL SCALE DEPOSITION AND SCALE DAMAGE TABLES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

WSCTAB assigns scale deposition and scale damage tables to a well, for when the Scale Deposition option has been activated by declaring the dimensions of the scaling deposition tables using the SCDPDIMS keyword in the RUNSPEC section. Scale deposits reduce the productivity of well and this relationship is defined in the SCDPTAB and SCDATAB keywords in the SCHEDULE section, and are allocated to a well by the WSCTAB keyword.

See also the WPIMULT keyword in the SCHEDULE section that adjusts a well's productivity index by a constant value.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.329 WSEGAICD – DEFINE MULTI-SEGMENT WELL AUTONOMOUS ICD CONNECTIONS

Description

The WSEGAICD keyword defines a multi-segment well segment to be an autonomous Inflow Control Device. ("ICD") as part of a completion for a multi-segment well. Note that the well must have been previously define by the WELSPECS and WELSEGS keywords in the SCHEDULE section and that the data for the keyword should be repeated for each multi-segment completion that contains an autonomous ICD.

An ICD is a well completion component usually installed along the producing section of a well to minimize the unwanted water and gas breakthrough in an oil well, or early water production in a gas well, due to an uneven flow profile over the completed interval. Permeability variations over the producing interval cause the high permeability zones to produce higher quantities of fluids than the lower permeability zones and this uneven producing fluid profile may result in bypassed hydrocarbons. Secondly, for horizontal wells, the pressure loss from the "toe" to the "heel" of the well again results in an uneven fluid profile over the producing interval. In order to rectify this ICDs can be installed so that the well fluids have to flow through an ICD before entering the tubing; thus, creating an additional "designed" pressure loss.

An autonomous ICD inhibits the production of high-mobility fluids such as water and gas since the pressure drop in each unit is dependent on fluid properties and mobility, the device automatically increases the pressure differential across zones with high water or gas saturations, thus choking back production from these zones.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WELNAME	A character string of up to eight characters in length that defines the well name for which a multi-segment well is being defined. Note that the well name (WELNAME) must have been declared previously using both the WELSPECS and WELSEGS keywords in the SCHEDULE section, otherwise an error may occur.			None
2	ISEG1	A positive integer greater than or equal to two and less than or equal to MXSEGS on the WSEGDIMS keyword in the RUNSPEC section that defines the start of a segment.			None
3	ISEG2	A positive integer greater than or equal to two and not less then ISEG1 on this record and less than or equal to MXSEGS on the WSEGDIMS keyword in the RUNSPEC section, that defines the end of a segment			None
4	ICDSTREN	A real positive value greater than zero that defines an empirical constant for the strength of the given ICD as determined from measurements using the calibrated fluid.			
		psia/((lb /ft3)/(rft3 day)2)	barsa/((kg/m3)(rm3/ day)2)	atma/((gm/cc)(rcc/ hr)2)	None

No.	Name	Description			Default
		Field	Metric	Laboratory	
5	ICDLEN	A real value that defines the length of the ICD used in conjunction with NSCAFAC to calculate a scaling factor to be applied to the reservoir flow to adjust the flow through each ICD, that is: 1) If NSCAFAC equals zero: then the scale factor is equal to the length of the ICD (ICDLEN) divided by the length of the tubing section, that is the parent of the ICDs, then this allows for the case when the ICD segment may represent a number of ICDs in parallel. 2) If NSCAFAC equals one: then the scale factor is equal to the absolute value of ICDLEN. 3) If NSCAFAC equals two: then the scale factor is equal to the length of ICDLEN, divided by the total length of the completions which supply the ICD. NSCALFAC explicitly sets which of the above three options is used. If NSCALFAC is defaulted, then option 1) is used whenever ICDLEN is positive and option 2) when ICDLEN is negative.			
		feet 39.37	m 12.00	cm 1,2000	Defined
6	CALDEN	CALDEN is a real positive value greater than zero that defines the density of the calibrating fluid at surface conditions.			
		lb/ft^3 62.416	kg/m^3 1000.25	gm/cc 1.00025	Defined
7	CALVISC	CALVISC is a real positive value greater than zero that defines the viscosity of the calibrating fluid at surface conditions.			
		cP	cP	cP	1.45
8	EMLCRT	EMLCRT is a real positive value greater than zero that defines the "local water" in liquid fraction used to determine whether the "water-in-oil" or "oil-in-water" viscosity emulation equation should be applied.			
		dimensionless	dimensionless	dimensionless	0.5
9	EMLTRANS	EMLTRANS is a real positive value greater than zero that defines the width of the transition zone around EMLCRT and is used to ensure that the calculated viscosity forms a continuous function of water in liquid fraction. Within this region, the emulsion viscosity is a linear interpolation between the "water-in-oil" and "oil-in-water" viscosity values either side of the region.			
		dimensionless	dimensionless	dimensionless	0.05
10	EMLMAX	EMLMAX is a real positive value greater than zero that defines the maximum emulsion viscosity to continuous phase viscosity (oil or water) ratio.			
		dimensionless	dimensionless	dimensionless	5.0

No.	Name	Description			Default
		Field	Metric	Laboratory	
11	NSCAFAC	NSCALFAC is a positive integer value that is greater than or equal to zero, that sets the method to be used when applying the scaling factor and should be set to one of the following: 1) If NSCAFAC equals zero: then the scale factor is equal to the length of the ICD (ICDLEN) divided by the length of the tubing section, that is the parent of the ICDs, then this allows for the case when the ICD segment may represent a number of ICDs in parallel. 2) If NSCAFAC equals one: then the scale factor is equal to the absolute value of ICDLEN. 3) If NSCAFAC equals two: then the scale factor is equal to the length of ICDLEN, divided by the total length of the completions which supply the ICD. NSCALFAC explicitly sets which of the above three options is used. If NSCALFAC is negative, then option 1) is used whenever ICDLEN is positive and option 2) when ICDLEN is negative.			
		dimensionless	dimensionless	dimensionless	-1
12	CALRATE	A real positive value that defines the maximum surface flow rate for which the ICD was calibrated. Values calculated greater than CALRATE will use linear extrapolation.			
		scf/d	sm^3/day	scc/hour	None
13	RATEXP	A real positive number greater than or equal to zero that defines flow rate exponent in equation (12.35).			
		dimensionless	dimensionless	dimensionless	None
14	VISCEXP	A real positive number greater than or equal to zero that defines viscosity exponent in equation (12.35).			
		dimensionless	dimensionless	dimensionless	None
15	STATUS	A defined character string of length four that defines the ICD's operational status, STATUS should be set to one of the following character strings: 1) OPEN: the ICD connection is are open to flow. 2) SHUT: the ICD connections is closed to flow (shut-in).			OPEN
16	A1	Density mixture OIL flowing exponent in equation (12.36)			1.0
17	A2	Density mixture WATER flowing exponent in equation (12.36)			1.0
18	A3	Density mixture GAS flowing exponent in equation (12.36)			1.0
19	B1	Viscosity mixture OIL flowing exponent in equation (12.37)			1.0
20	B2	Viscosity mixture WATER flowing exponent in equation (12.37)			1.0
21	B3	Viscosity mixture GAS flowing exponent in equation (12.37)			1.0

No.	Name	Description			Default
		Field	Metric	Laboratory	
Notes: 1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".					

Table 12.123: WSEGAICD Keyword Description

The total number of wells should be defined via the WELLSDIMS keyword and the number of multi-segment wells should be declared on the WSEGDIMS keyword, both keywords are in the RUNSPEC section. In addition, the WELSPECS keyword should be used to define wells, the COMPDAT keyword to define the well completions for both ordinary wells and multi-segment wells, and the COMPSEGS keyword to define a multi-segment segment completions. Finally, the WSEGSICD keyword can then be use to define ICD connections for the well. All the aforementioned keywords are described in the SCHEDULE section.

The equations used to calculate the pressure drop across the ICD are given below and illustrate how the pressure reduction is dependent on the density and viscosity of the fluid flowing through the device.

$$\Delta P = \left(\frac{\rho_{mixture}}{\rho_{calibrated}}\right)\cdot\left(\frac{\mu_{calibrated}}{\mu_{mixture}}\right)^{\mu_{\exp}} \cdot \rho_{mixture} \cdot \beta \cdot \left(\frac{q}{q_{calibrated}}\right)^{(Q_{\exp}-2)} \cdot q^2 \quad (12.35)$$

Where:

ΔP = the pressure drop across the device.
ρmixture = the density of the mixture, as per:

$$\rho_{mixture} = \left(\alpha_{oil}^{a_1}\cdot\rho_{oil}\right) + \left(\alpha_{wat}^{a_2}\cdot\rho_{wat}\right) + \left(\alpha_{gas}^{a_3}\cdot\rho_{gas}\right) \quad (12.36)$$

$\rho_{calibrated}$ = CALDEN, the density of the calibrating fluid at surface conditions.
μcalibrated = CALVISC, the viscosity of the calibrating fluid at surface conditions.
μmixture = the viscosity of the mixture, as per:

$$\mu_{mixture} = \left(\alpha_{oil}^{b_1}\cdot\mu_{oil}\right) + \left(\alpha_{wat}^{b_2}\cdot\mu_{wat}\right) + \left(\alpha_{gas}^{b_3}\cdot\mu_{gas}\right) \quad (12.37)$$

β = ICDSTREN, the strength of the ICD as measured using the calibrated fluid.
q = the local flow rate through the ICD at local conditions adjusted for scaling based on ICDLEN and NSCAFAC parameters in Table 12.123.
qcalibrated = CALRATE, the maximum surface flow rate for which the ICD was calibrated.
μEXP = VISCEXP, the viscosity exponent in Table 12.123.
QEXP = RATEXP, the flow rate exponent in Table 12.123.

In equation Error: Reference source not found αi represents the volume fraction of oil, water and gas at local conditions and ai the density exponents of the three phases (A1, A2 and A3 in Table 12.123). Similarity for equation (12.37), αi represents the volume fraction of oil, water and gas at local conditions and Bi the viscosity exponents of the three phases (B1, B2 and B3 in Table 12.123).

See also the WSEGSICD keyword in the SCHEDULE section for spiral ICDs, that work in a similar fashion to how autonomous ICDs work.

Example

The following example defines one producing well segment oil well (OP01) using the WELSPECS, WELSEGS COMPDAT and COMPSEGS keywords, followed by the WSEGAICD keyword to define the autonomous inflow control devices for the well.

```
--
--       WELL SPECIFICATION DATA
--
-- WELL  GROUP    LOCATION  BHP   PHASE  DRAIN  INFLOW  OPEN  CROSS PVT   DEN  FIP
-- NAME  NAME       I    J  DEPTH FLUID  AREA   EQUANS  SHUT  FLOW  TABLE CAL  NUM
WELSPECS
OP01     PLATFORM 14    8  1*    OIL    0.0    STD     SHUT  YES   0     SEG  0 /
/
--
--       WELL CONNECTION DATA
--
-- WELL   --- LOCATION ---   OPEN   SAT  CONN          WELL  KH    SKIN   D    DIR
-- NAME    II  JJ  K1  K2    SHUT   TAB  FACT          DIA   FACT  FACT   FACT PEN
COMPDAT
OP01      14   8   1   1  OPEN   1*   1.972960E+2 0.216  1*    0.00   1*   'Z' /
OP01      14   7   1   1  OPEN   1*   1.619450E+2 0.216  1*    0.00   1*   'Z' /
OP01      14   7   2   2  OPEN   1*   4.126449E+2 0.216  1*    0.00   1*   'Z' /
OP01      14   7   3   3  OPEN   1*   2.033290E+1 0.216  1*    0.00   1*   'Z' /
OP01      15   7   3   3  OPEN   1*   9.095613E+1 0.216  1*    0.00   1*   'Z' /
OP01      15   6   3   3  OPEN   1*   2.090607E+2 0.216  1*    0.00   1*   'Y' /
OP01      15   6   4   4  OPEN   1*   3.010669E+1 0.216  1*    0.00   1*   'Y' /
OP01      16   6   4   4  OPEN   1*   7.123814E+1 0.216  1*    0.00   1*   'Y' /
OP01      16   5   4   4  OPEN   1*   4.414386E+2 0.216  1*    0.00   1*   'Y' /
OP01      16   4   4   4  OPEN   1*   4.345126E+2 0.216  1*    0.00   1*   'Y' /
OP01      16   3   4   4  OPEN   1*   2.894573E+2 0.216  1*    0.00   1*   'Y' /
OP01      17   3   4   4  OPEN   1*   1.329523E+2 0.216  1*    0.00   1*   'Y' /
OP01      17   2   4   4  OPEN   1*   6.981252E+1 0.216  1*    0.00   1*   'Y' /
OP01      17   2   5   5  OPEN   1*   1.392382E+2 0.216  1*    0.00   1*   'Y' /
/
--
--       WELL SEGMENT SPECIFICATION DATA
--
-- WELL  NODAL       LEN     WELL    DEPH    PRESS   FLOW
-- NAME  DEPTH       TUBING  VOLM    OPTN    CALC    MODEL
WELSEGS
   OP01  2041.56259  0.0000  1*      INC     'HF-'         /
--
--       SEG   SEG   BRAN    SEG     TUBING  NODAL   TUBE   TUBE      XSEC  VOL
--       ISTR  IEND  NO      NO      LENGTH  DEPTH   ID     ROUGH     AREA  SEG
          2     2    1        1     9.56005  0.61168 0.152  0.0000100         /
          3     3    1        2    17.40714  1.11376 0.152  0.0000100         /
          4     4    1        3    41.24996  2.38255 0.152  0.0000100         /
          5     5    1        4    38.35922  2.06899 0.152  0.0000100         /
          6     6    1        5    27.13029  1.02248 0.152  0.0000100         /
          7     7    1        6    73.45099  2.40389 0.152  0.0000100         /
          8     8    1        7    54.95304  1.64832 0.152  0.0000100         /
          9     9    1        8    12.37381  0.23781 0.152  0.0000100         /
         10    10    1        9    62.61459  0.73403 0.152  0.0000100         /
         11    11    1       10    106.9805  1.37749 0.152  0.0000100         /
         12    12    1       11    88.42739  0.90931 0.152  0.0000100         /
         13    13    1       12    51.59899  0.27327 0.152  0.0000100         /
         14    14    1       13    24.75582  0.30814 0.152  0.0000100         /
         15    15    1       14    29.77334  0.49371 0.152  0.0000100         /
```

```
--
-- PERFORATION VALVE SEGMENTS
--
--       SEG   SEG   BRAN    SEG     TUBING  NODAL   TUBE    TUBE     XSEC  VOL
--       ISTR  IEND  NO      NO      LENGTH  DEPTH   ID      ROUGH    AREA  SEG
         16    16    2        2    0.10000   0       0.152  0.0000100         /
         17    17    3        3    0.10000   0       0.152  0.0000100         /
         18    18    4        4    0.10000   0       0.152  0.0000100         /
         19    19    5        5    0.10000   0       0.152  0.0000100         /
         20    20    6        6    0.10000   0       0.152  0.0000100         /
         21    21    7        7    0.10000   0       0.152  0.0000100         /
         22    22    8        8    0.10000   0       0.152  0.0000100         /
         23    23    9        9    0.10000   0       0.152  0.0000100         /
         24    24   10       10    0.10000   0       0.152  0.0000100         /
         25    25   11       11    0.10000   0       0.152  0.0000100         /
         26    26   12       12    0.10000   0       0.152  0.0000100         /
         27    27   13       13    0.10000   0       0.152  0.0000100         /
         28    28   14       14    0.10000   0       0.152  0.0000100         /
         29    29   15       15    0.10000   0       0.152  0.0000100         /
/
--
--       MULTISEGMENT WELL COMPLETION SEGMENT SPECIFICATION DATA
--
-- WELL
-- NAME
COMPSEGS
OP01                                                                          /
--
--      --LOCATION--  BRAN  TUBING     NODAL     DIR  LOC     MID     COMP     ISEG
--       II  JJ  K1   NO    LENGTH     DEPTH     PEN  I,J,K   PERFS   LENGTH   NO.
         14   8   1   2     0.000000   0.10000                                  /
         14   7   1   3     0.000000   0.10000                                  /
         14   7   2   4     0.000000   0.10000                                  /
         14   7   3   5     0.000000   0.10000                                  /
         15   7   3   6     0.000000   0.10000                                  /
         15   6   3   7     0.000000   0.10000                                  /
         15   6   4   8     0.000000   0.10000                                  /
         16   6   4   9     0.000000   0.10000                                  /
         16   5   4   10    46.76168   46.86168                                 /
         16   4   4   11    0.000000    0.10000                                 /
         16   3   4   12    0.000000    0.10000                                 /
         17   3   4   13    0.000000    0.10000                                 /
         17   2   4   14    0.000000    0.10000                                 /
         17   2   5   15    42.50573   42.60573                                 /
/
--
--       MULTI-SEGMENT WELL ICD SEGMENT SPECIFICATION DATA
--
-- WELL SEG  SEG  ICD     ICD       CAL  CAL  EML  EML  EML SCAL CAL RATE VISC OPEN
-- NAME ISTR IEND STREN   LEN       DEN  VISC CRIT TRAN MAX FAC  RAT EXP  EXP CLOSE
WSEGAICD
OP01    16   16  7.5e-5 -0.028975 1020 0.48 0.7  1*   1*  1*   1*  2.2  0.5  7* /
OP01    17   17  7.5e-5 -0.023783 1020 0.48 0.7  1*   1*  1*   1*  2.2  0.5  7* /
OP01    18   18  7.5e-5 -0.101240 1020 0.48 0.7  1*   1*  1*   1*  2.2  0.5  7* /
OP01    19   19  7.5e-5 -0.015022 1020 0.48 0.7  1*   1*  1*   1*  2.2  0.5  7* /
OP01    20   20  7.5e-5 -0.067205 1020 0.48 0.7  1*   1*  1*   1*  2.2  0.5  7* /
OP01    21   21  7.5e-5 -0.155410 1020 0.48 0.7  1*   1*  1*   1*  2.2  0.5  7* /
OP01    22   22  7.5e-5 -0.011141 1020 0.48 0.7  1*   1*  1*   1*  2.2  0.5  7* /
OP01    23   23  7.5e-5 -0.026362 1020 0.48 0.7  1*   1*  1*   1*  2.2  0.5  7* /
OP01    24   24  7.5e-5 -0.163410 1020 0.48 0.7  1*   1*  1*   1*  2.2  0.5  7* /
OP01    25   25  7.5e-5 -0.160830 1020 0.48 0.7  1*   1*  1*   1*  2.2  0.5  7* /
OP01    26   26  7.5e-5 -0.107180 1020 0.48 0.7  1*   1*  1*   1*  2.2  0.5  7* /
--
```

```
-- WELL SEG  SEG  ICD    ICD      CAL  CAL  EML  EML  EML SCAL CAL RATE VISC OPEN
-- NAME ISTR IEND STREN  LEN      DEN  VISC CRIT TRAN MAX FAC  RAT EXP  EXP CLOSE
OP01    27   27  7.5e-5 -0.049206 1020 0.48 0.7  1*   1*  1*   1*  2.2  0.5  7* /
OP01    28   28  7.5e-5 -0.025824 1020 0.48 0.7  1*   1*  1*   1*  2.2  0.5  7* /
OP01    29   29  7.5e-5 -0.064414 1020 0.48 0.7  1*   1*  1*   1*  2.2  0.5  7* /
/
```

The WSEGAICD keyword defines the autonomous ICD over segments 16 to 29 using the default value for NSCALFAC of -1, which sets the scale factor equal to the absolute value of ICDLEN, as all the ICDLEN values are negative. Note that the WSEGAICD should be repeated for each multi-segment well that has this type of ICD. So for example for two wells one would have:

```
--
--       MULTI-SEGMENT WELL ICD SEGMENT SPECIFICATION DATA
--
-- WELL SEG  SEG  ICD    ICD      CAL  CAL  EML  EML  EML SCAL CAL RATE VISC OPEN
-- NAME ISTR IEND STREN  LEN      DEN  VISC CRIT TRAN MAX FAC  RAT EXP  EXP CLOSE
WSEGAICD
OP01    16   16  7.5e-5 -0.028975 1020 0.48 0.7  1*   1*  1*   1*  2.2  0.5  7* /
…….....……...……..
OP01    28   28  7.5e-5 -0.025824 1020 0.48 0.7  1*   1*  1*   1*  2.2  0.5  7* /
/

-- WELL SEG  SEG  ICD    ICD      CAL  CAL  EML  EML  EML SCAL CAL RATE VISC OPEN
-- NAME ISTR IEND STREN  LEN      DEN  VISC CRIT TRAN MAX FAC  RAT EXP  EXP CLOSE
WSEGAICD
OP02    17   17  1.75E-4 .0049945 1020 0.48 1*   1*   1*  1*   1*  2.1  1.2  7* /
…….....……...……..
OP02    31   31  1.75E-4 .0038583 1020 0.48 1*   1*   1*  1*   1*  2.1  1.2  7* /
/
```

Figure 12.16 show the layout of the two example wells.

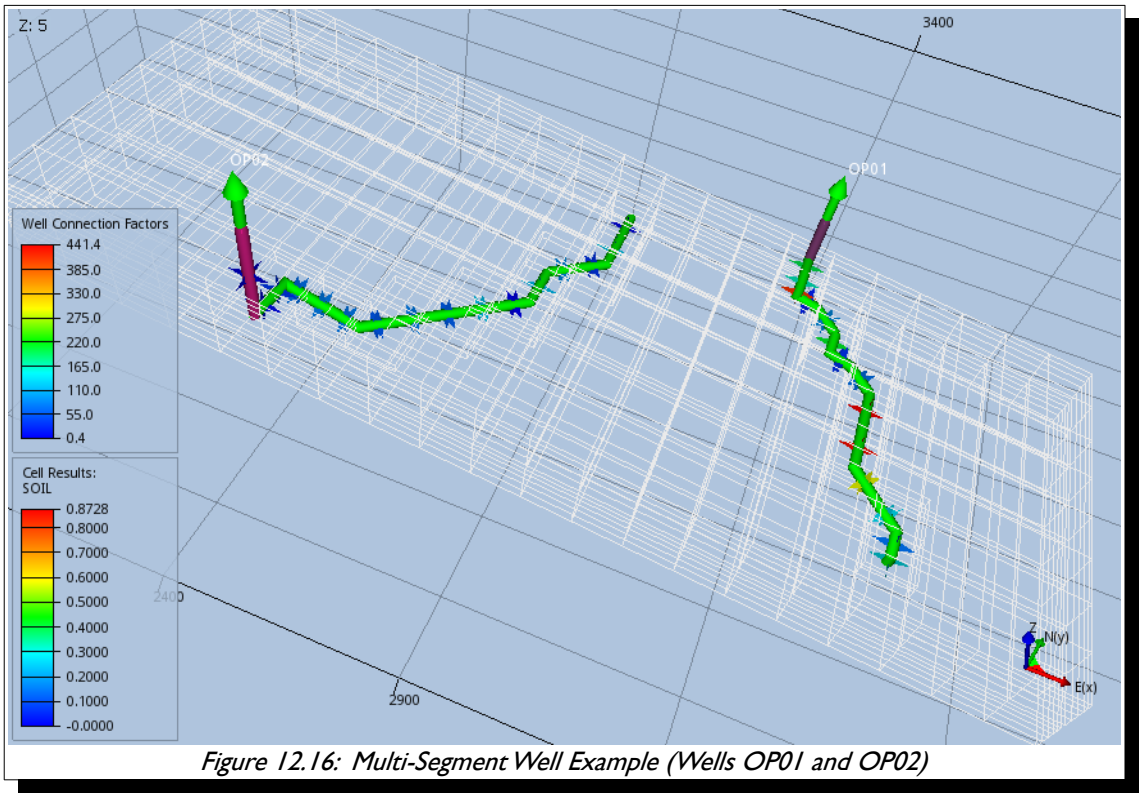


Figure 12.16: Multi-Segment Well Example (Wells OP01 and OP02)

12.3.330 WSEGDFIN – DEFINE MULTI-SEGMENT WELL DRIFT FLUX SLIP MODEL INPUT DATA

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The WSEGDFIN keyword defines a multi-segment well's drift flux slip model parameters. A slip model[337] and [338] enables the different phases in the wellbore to flow at different velocities, for example gas will flow up the tubing at a higher velocity than oil and water. The option is activated by the FLOWOPT parameter on the WELSEGS keyword in the SCHEDULE section; however, the slip model flow calculation is not available in OPM Flow.

See also the WSEGFMD keyword in the SCHEDULE section that sets which type of slip model should be used.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

[337] *Shi, H., Holmes, J.A., Durlofsky, L. J., Aziz, K., Diaz, L. R., Alkaya, B., and Oddie, G. "Drift-Flux Modeling of Two-Phase Flow in Wellbores," paper SPE 84228, Society of Petroleum Engineers Journal (2005) 10, No. 1, 24-33; also presented as "Drift-Flux Modeling of Multiphase Flow in Wellbores," at the SPE Annual Technical Conference and Exhibition, Denver, Colorado, USA(October 5-8, 2003).*

[338] *Shi, H., Holmes, J.A., Diaz, L. R., Durlofsky, L. J., and Aziz, K. "Drift-Flux Parameters for Three-Phase Steady-State Flow in Wellbores," paper SPE 89836, Society of Petroleum Engineers Journal(2005) 10, No. 2, 130-137; also presented at the SPE Annual Technical Conference and Exhibition, Houston, Texas, USA (September 26-29, 2004).*

12.3.331 WSEGDFMD – DEFINE MULTI-SEGMENT WELL DRIFT FLUX SLIP MODEL

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The WSEGDFMD keyword defines a multi-segment well's drift flux slip model definition that sets the type of slip model to be used. A slip model[339] and [340] enables the different phases in the wellbore to flow at different velocities, for example gas will flow up the tubing at a higher velocity than oil and water. The option is activated by the FLOWOPT parameter on the WELSEGS keyword in the SCHEDULE section; however, the slip model flow calculation is not available in OPM Flow.

See also the WSEGDFIN keyword that sets the slip model's input parameters and the WSEGDFPA keyword that allows the model default values employed by WSEGDFMD to be modified..

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

[339] *Shi, H., Holmes, J.A., Durlofsky, L. J., Aziz, K., Diaz, L. R., Alkaya, B., and Oddie, G. "Drift-Flux Modeling of Two-Phase Flow in Wellbores," paper SPE 84228, Society of Petroleum Engineers Journal (2005) 10, No. 1, 24-33; also presented as "Drift-Flux Modeling of Multiphase Flow in Wellbores," at the SPE Annual Technical Conference and Exhibition, Denver, Colorado, USA(October 5-8, 2003).*

[340] *Shi, H., Holmes, J.A., Diaz, L. R., Durlofsky, L. J., and Aziz, K. "Drift-Flux Parameters for Three-Phase Steady-State Flow in Wellbores," paper SPE 89836, Society of Petroleum Engineers Journal(2005) 10, No. 2, 130-137; also presented at the SPE Annual Technical Conference and Exhibition, Houston, Texas, USA (September 26-29, 2004).*

12.3.332 WSEGDFPA – DEFINE MULTI-SEGMENT WELL DRIFT FLUX SLIP MODEL PARAMETERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, WSEGDFPA, enables modification of a multi-segment well's drift flux slip model default parameters used by the WSEGDFMA keyword in the SCHEDULE section to define the model. A slip model[341] and [342] enables the different phases in the wellbore to flow at different velocities, for example gas will flow up the tubing at a higher velocity than oil and water. The option is activated by the FLOWOPT parameter on the WELSEGS keyword in the SCHEDULE section; however, the slip model flow calculation is not available in OPM Flow.

See also the WSEGDFIN keyword that sets the slip model's input parameters and the WSEGDFMD keyword that sets which drift flux slip model should be used. Note if the WSEGDFPA keyword is used than it must be placed after the WSEGDFMD keyword, but before the WELSEGS keyword that defines a multi-segment well. All the aforementioned keywords are in the SCHEDULE section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

[341] *Shi, H., Holmes, J.A., Durlofsky, L. J., Aziz, K., Diaz, L. R., Alkaya, B., and Oddie, G. "Drift-Flux Modeling of Two-Phase Flow in Wellbores," paper SPE 84228, Society of Petroleum Engineers Journal (2005) 10, No. 1, 24-33; also presented as "Drift-Flux Modeling of Multiphase Flow in Wellbores," at the SPE Annual Technical Conference and Exhibition, Denver, Colorado, USA(October 5-8, 2003).*

[342] *Shi, H., Holmes, J.A., Diaz, L. R., Durlofsky, L. J., and Aziz, K. "Drift-Flux Parameters for Three-Phase Steady-State Flow in Wellbores," paper SPE 89836, Society of Petroleum Engineers Journal(2005) 10, No. 2, 130-137; also presented at the SPE Annual Technical Conference and Exhibition, Houston, Texas, USA (September 26-29, 2004).*

12.3.333 WSEGEXSS – DEFINE MULTI-SEGMENT WELL IMPORT-EXPORT SEGMENT VOLUMES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, WSEGEXSS, enables the import or export of fluids from a segment in a multi-segment well. This can be used to, for example, model gas lift injection for oil wells under artificial lift, or to approximate the behavior of a down-hole separator. The import-export fluid volumes can either be expressed as rates or defined as a function of a segment's pressure value.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.334 WSEGFLIM - DEFINE MULTI-SEGMENT WELL ARTIFICIAL CHOKE CONNECTIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

WSEGFLIMS enables an artificial choke that chokes a given phase flow rate for a segment in a multi-segment well. This can be used, for example, to constraint unwanted production phase through a section of tubing, or to model a down-hole choke. The keyword provides coefficients that are applied to the frictional pressure drop across a multi-segment well's segment in order to inhibit production from that particular zone or segment. As such, the keyword does not actually model a down-hole choke; hence, the term artificial.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.335 WSEGFMOD – DEFINE MULTI-SEGMENT WELL MODEL

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The WSEGFMOD declares the multi-phase flow model to be used to calculate the pressure drop within an individual segment for multi-segment wells. The FLOWOPT parameter on the WELSEGS keyword in the SCHEDULE section sets the default multi-segment well model. FLOWOPT is a character string that can be set to HO that activates the homogeneous model, that is all phases flow at the same velocity, or DF that invokes the Drift Flux Slip model (note OPM Flow only supports the default value of HO for the homogeneous model). Here WSEGFMOD can be used to set the flow model for a segment to either the homogeneous model or the Drift Flux Slip model, and addition a: VLP table allocated via the WSEGTABL keyword, or a specific model as defined by the WSEGVALV, WSEGFLIM and WSEGLABY keywords. All the aforementioned keywords are in the SCHEDULE section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.336 WSEGINIT - DEFINE MULTI-SEGMENT WELL INITIAL CONDITIONS

Description

Normally the simulator calculates the initial conditions for multi-segment wells, that is the pressure and fluid distributions in each segment. However, there are occasions when manually setting the pressures and phase distributions for each segment to investigate certain flow conditions may be useful. In this case the WSEGINIT keyword may be used to specify the initial conditions manually. Note that segments not initialized by this keyword will be automatically initialized by the simulator

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.337 WSEGITER – DEFINE MULTI-SEGMENT WELLS ITERATION PARAMETERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The WSEGSITER keyword defines the multi-segment well solution iteration sequence and solution controls.

OPM Flow uses a different numerical scheme which makes this keyword redundant; hence, OPM Flow ignores this keyword. It is documented here for completeness. See section *2.2 Running OPM Flow 2023-10 From The Command Line* on how to set the numerical control parameters for OPM Flow.

12.3.338 WSEGLABY - DEFINE MULTI-SEGMENT WELL LABYRINTH ICD CONNECTIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The WSEGSICD keyword defines a multi-segment well segment to be a labyrinth Inflow Control Device. ("ICD") as part of a completion for a multi-segment well. Note that the well must have been previously define by the WELSPECS and WELSEGS keywords in the SCHEDULE section and that the data for the keyword should be repeated for each multi-segment completion that contains a labyrinth ICD.

An ICD is a well completion component usually installed along the producing section of a well to minimize the unwanted water and gas breakthrough in an oil well, or early water production in a gas well, due to an uneven flow profile over the completed interval. Permeability variations over the producing interval cause the high permeability zones to produce higher quantities of fluids than the lower permeability zones and this uneven producing fluid profile may result in bypassed hydrocarbons. Secondly, for horizontal wells, the pressure loss from the "toe" to the "heel" of the well again results in an uneven fluid profile over the producing interval. In order to rectify this ICDs can be installed so that the well fluids have to flow through an ICD before entering the tubing; thus, creating an additional "designed" pressure loss.

A labyrinth ICD is a type of frictional ICD that adds an additional pressure loss by directing the fluid along a series of channels before entering the tubing. The channel flow path is designed in such a manner as to create the desired pressure loss for a given ICD. By placing various ICD's over the production interval one can design a completion that results in a more uniform producing fluid profile throughout the length of the producing interval. Although this type of ICD is not implemented in OPM Flow, it works in a similar fashion to how a spiral ICD works. Spiral ICDs are implemented in OPM Flow and the data is entered via the WSEGSICD keyword in the SCHEDULE section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.339 WSEGLINK - DEFINE MULTI-SEGMENT WELL LOOPED FLOW PATHS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, WSEGLINK, specifies multi-segment well looped flow paths as part of a completion for a multi-segment well. A looped segment results in the nodes of the two individual segments that are looped (or connected) having the same solution pressures and oil, water and gas flowing rates.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.340 WSEGMULT – DEFINE MULTI-SEGMENT WELL FRICTIONAL PRESSURE LOSS MULTIPLIERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

WEGMULT supplies a set of constants used to modify (or scale) a multi-segment well's segment frictional pressure drop between connecting segments The constants enable either a constant pressure to be applied, or for the pressure drop to vary as a function of the Gas-Oil Ratio ("GOR") or the Water-Oil Ratio ("WOR"). The simulator calculated pressure drop is multiplied by the following resulting value:

$$Frictional\ Loss\ Multipler\ =\ min\left(x_1 + x_2(WOR)^{x_3} + x_4\left(\frac{GOR}{GOR_{min}}\right)^{x^5},\ 1.0\right) \tag{12.38}$$

Where the constants x_1 to x_5 are defined by the values on this keyword.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.341 WSEGPROP – MODIFY MULTI-SEGMENT WELLS AND THEIR SEGMENT STRUCTURE

Description

The WSEGPROP keyword allows for the editing of exiting multi-segment wells created by WELSEGS keyword in the SCHEDULE section without having to re-define all the information that is on the WELSEGS keyword. Note that the well must have been previously define by both the WELSPECS and WELSEGS keywords in the SCHEDULE section to use the WSEGPROP keyword.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WELNAME	A character string of up to eight characters in length that defines the well name for which a multi-segment well is being defined. Note that the well name (WELNAME) must have been declared previously using the WELSPECS keyword in the SCHEDULE section, otherwise an error may occur.			None
2	ISEG1	A positive integer greater than or equal to two and less than or equal to MXSEGS on WSEGDIMS keyword in the RUNSPEC section that defines the start of a segment			None
3	ISEG2	A positive integer greater than or equal to two and less than or equal to ISEG1 on this record and MXSEGS on the WSEGDIMS keyword in the RUNSPEC section that defines the end of a segment.			None
4	ID	A real positive value that defines the tubing internal diameter of the segment for the well.			Previous Entered Value
		feet	m	cm	
5	EPSILON	A real positive value that defines the tubing absolute roughness of the segment for the well.			Previous Entered Value
		feet	m	cm	
6	XAREA	XAREA is real positive value equal to or greater than zero that defines the cross sectional area for fluid flow. Currently this option is not supported by OPM Flow.			Previous Entered Value
		ft^2	m^2	cm^2	
7	VOLSEG	VOLSEG is a real positive value that defines the effective segment volume for the this segment. Currently this option is not supported by OPM Flow.			Previous Entered Value
		ft3	m3	cm3	

No.	Name	Description			Default
		Field	Metric	Laboratory	
8	XAREAS	XAREAS is real positive value equal to or greater than zero that defines the cross sectional area of the pipe wall for this segment, that is used in thermal conductivity calculations for when the temperature calculation is activated by the TEMP keyword in the RUNSPEC section. Currently this option is not supported by OPM Flow.			Previous Entered Value
		ft^2	m^2	cm^2	
9	VHEATCAP	VHEATCAP is real positive value equal to or greater than zero that defines the volumetric heat capacity of the pipe wall used in thermal conductivity calculations for when the temperature calculation is activated by the TEMP keyword in the RUNSPEC section. Currently this option is not supported by OPM Flow.			Previous Entered Value
		Btu/ft/day/°R	kJ/m/day/K	J/cm/hr/K	
10	THCON	THCON is real positive value equal to or greater than zero that defines the thermal conductivity of the pipe wall used in thermal conductivity calculations for when the temperature calculation is activated by the TEMP keyword in the RUNSPEC section. Currently this option is not supported by OPM Flow.			Previous Entered Value
		Btu/ft/day/°R	kJ/m/day/K	J/cm/hr/K	

Notes:

1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".

Table 12.124: WSEGPROP Keyword Description

The total number of wells should be defined via the WELLSDIMS keyword and the number of multi-segment wells should be declared on the WSEGDIMS keyword, both keywords are in the RUNSPEC section.

See also the WELSPECS keyword to define wells, the COMPDAT keyword to define the well completions for both ordinary wells and multi-segment wells, and the COMPSEGS keyword to define a multi-segment well segment completions. All the aforementioned keywords are described in the SCHEDULE section.

Example

The following example modifies two segments in well OP01 and one segment in well OP02.

```
--
--        WELL SEGMENT SPECIFICATION DATA
--
-- WELL   SEG    SEG    TUBE    TUBE      XSEC   VOL
-- NAME   ISTR   IEND   ID      ROUGH     AREA   SEG
WSEGPROP
OP01       12     14    0.3     0.00010                    /
OP01       13     15    0.275   0.00010                    /
OP02       14     14    0.275   0.00010                    /
/
```

Note that the two multi-segment wells and their respective segments must have been previously defined by the WELSEGS keyword.

12.3.342 WSEGPULL – DEFINE A MULTI-SEGMENT WELL DOWN-HOLE SEPARATOR PUMP

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, WSEGPULL, specifies a multi-segment well segment to be a pull-through pump for a down-hole water separator, defined by the WSEGSEP keyword in the SCHEDULE section, and defines the various parameters for this type of pump. Down-hole separators are used to separate water or free gas from the in situ fluid entering the wellbore in order to increase hydrocarbon recovery.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.343 WSEGSEP – DEFINE A MULTI-SEGMENT WELL DOWN-HOLE SEPARATOR

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

WSEGSEP specifies a multi-segment well segment to be a down-hole separator, that enables the separation of fluids down-hole. Down-hole separators are used to separate water or free gas from the in situ fluid entering the wellbore in order to increase hydrocarbon recovery. See also the WSEGPULL keyword in the SCHEDULE section that specifies a pull-through pump for a down-hole water separator.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.344 WSEGSICD – DEFINE MULTI-SEGMENT WELL SPIRAL ICD CONNECTIONS

Description

The WSEGSICD keyword defines a multi-segment well segment to be a spiral Inflow Control Device. ("ICD") as part of a completion for a multi-segment well. Note that the well must have been previously define by the WELSPECS and WELSEGS keywords in the SCHEDULE section and that the data for the keyword should be repeated for each multi-segment completion that contains a spiral ICD.

An ICD is a well completion component usually installed along the producing section of a well to minimize the unwanted water and gas breakthrough in an oil well, or early water production in a gas well, due to an uneven flow profile over the completed interval. Permeability variations over the producing interval cause the high permeability zones to produce higher quantities of fluids than the lower permeability zones and this uneven producing fluid profile may result in bypassed hydrocarbons. Secondly, for horizontal wells, the pressure loss from the "toe" to the "heel" of the well again results in an uneven fluid profile over the producing interval. In order to rectify this ICDs can be installed so that the well fluids have to flow through an ICD before entering the tubing; thus, creating an additional "designed" pressure loss.

A spiral ICD is a type of frictional ICD that adds an additional pressure loss by directing the fluid along a "channel" (spiral/helix) before entering the tubing. The channel flow path is designed in such a manner as to create the desired pressure loss for a given ICD. By placing various ICD's over the production interval one can design a completion that results in a more uniform producing fluid profile throughout the length of the producing interval.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WELNAME	A character string of up to eight characters in length that defines the well name for which a multi-segment well is being defined. Note that the well name (WELNAME) must have been declared previously using both the WELSPECS and WELSEGS keywords in the SCHEDULE section, otherwise an error may occur.			None
2	ISEG1	A positive integer greater than or equal to two and less than or equal to MXSEGS on the WSEGDIMS keyword in the RUNSPEC section that defines the start of a segment			None
3	ISEG2	A positive integer greater than or equal to two and not less then ISEG1 on this record and less than or equal to MXSEGS on the WSEGDIMS keyword in the RUNSPEC section, that defines the end of a segment			None
4	ICDSTREN	A real positive value greater than zero that defines an empirical constant for the strength of the given ICD as determined from measurements using the calibrated fluid.			
		psia(rft3/day)2	barsa/(rm3/day)2	atma/(rcc/hr)2	None

No.	Name	Description			Default
		Field	Metric	Laboratory	
5	ICDLEN	A real value that defines the length of the ICD used in conjunction with NSCAFAC to calculate a scaling factor to be applied to the reservoir flow to adjust the flow through each ICD, that is: 1) If NSCAFAC equals zero: then the scale factor is equal to the length of the ICD (ICDLEN) divided by the length of the tubing section, that is the parent of the ICDs, then this allows for the case when the ICD segment may represent a number of ICDs in parallel. 2) If NSCAFAC equals one: then the scale factor is equal to the absolute value of ICDLEN. 3) If NSCAFAC equals two: then the scale factor is equal to the length of ICDLEN, divided by the total length of the completions which supply the ICD. NSCALFAC explicitly sets which of the above three options is used. If NSCALFAC is defaulted, then option 1) is used whenever ICDLEN is positive and option 2) when ICDLEN is negative.			
		feet 39.37	m 12.00	cm 1,2000	Defined
6	CALDEN	CALDEN is a real positive value greater than zero that defines the density of the calibrating fluid at surface conditions.			
		lb/ft^3 62.416	kg/m^3 1000.25	gm/cc 1.00025	Defined
7	CALVISC	CALVISC is a real positive value greater than zero that defines the viscosity of the calibrating fluid at surface conditions.			
		cP	cP	cP	1.45
8	EMLCRT	EMLCRT is a real positive value greater than zero that defines the "local water" in liquid fraction used to determine whether the "water-in-oil" or "oil-in-water" viscosity emulation equation should be applied.			
		dimensionless	dimensionless	dimensionless	0.5
9	EMLTRANS	EMLTRANS is a real positive value greater than zero that defines the width of the transition zone around EMLCRT and is used to ensure that the calculated viscosity forms a continuous function of water in liquid fraction. Within this region, the emulsion viscosity is a linear interpolation between the "water-in-oil" and "oil-in-water" viscosity values either side of the region.			
		dimensionless	dimensionless	dimensionless	0.05
10	EMLMAX	EMLMAX is a real positive value greater than zero that defines the maximum emulsion viscosity to continuous phase viscosity (oil or water) ratio.			
		dimensionless	dimensionless	dimensionless	5.0

No.	Name	Description			Default
		Field	Metric	Laboratory	
11	NSCAFAC	NSCALFAC is a positive integer value that is greater than or equal to zero, that sets the method to be used when applying the scaling factor and should be set to one of the following: 1) If NSCAFAC equals zero: then the scale factor is equal to the length of the ICD (ICDLEN) divided by the length of the tubing section, that is the parent of the ICDs, then this allows for the case when the ICD segment may represent a number of ICDs in parallel. 2) If NSCAFAC equals one: then the scale factor is equal to the absolute value of ICDLEN. 3) If NSCAFAC equals two: then the scale factor is equal to the length of ICDLEN, divided by the total length of the completions which supply the ICD. NSCALFAC explicitly sets which of the above three options is used. If NSCALFAC is defaulted, then option 1) is used whenever ICDLEN is positive and option 2) when ICDLEN is negative.			
		dimensionless	dimensionless	dimensionless	-1
12	CALRATE	A real positive value that defines the maximum surface flow rate for which the ICD was calibrated.			
		scf/d	sm^3/day	scc/hour	None
13	STATUS	A defined character string of length four that defines the ICD's operational status, STATUS should be set to one of the following character strings: 1) OPEN: the ICD connection is are open to flow. 2) SHUT: the ICD connections is closed to flow (shut-in).			OPEN

Notes:

1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".

Table 12.125: WSEGSICD Keyword Description

The total number of wells should be defined via the WELLSDIMS keyword and the number of multi-segment wells should be declared on the WSEGDIMS keyword, both keywords are in the RUNSPEC section. In addition, the WELSPECS keyword should be used to define wells, the COMPDAT keyword to define the well completions for both ordinary wells and multi-segment wells, and the COMPSEGS keyword to define a multi-segment segment completions. Finally, the WSEGSICD keyword can then be use to define ICD connections for the well. All the aforementioned keywords are described in the SCHEDULE section.

The equations used to calculate the pressure drop across the ICD are given below and illustrate how the pressure reduction is dependent on the density and viscosity of the fluid flowing through the device.

$$\Delta P = \left(\frac{\rho_{calibrated}}{\rho_{mixture}} \cdot \frac{\mu_{mixture}}{\mu_{calibrated}}\right)^{\frac{1}{4}} \cdot \frac{\rho_{mixture}}{\rho_{calibrated}} \cdot \beta \cdot q^2 \quad (12.39)$$

Where:

ΔP = the pressure drop across the device.
ρmixture = the density of the mixture, as per:

$$\rho_{mixture} = \left(\alpha_{oil} \cdot \rho_{oil}\right) + \left(\alpha_{wat} \cdot \rho_{wat}\right) + \left(\alpha_{gas} \cdot \rho_{gas}\right) \quad (12.40)$$

$\rho_{calibrated}$ = CALDEN, the density of the calibrating fluid at surface conditions.
μcalibrated = CALVISC, the viscosity of the calibrating fluid at surface conditions.
μmixture = the viscosity of the mixture, as per:

$$\mu_{mixture} = \left(\alpha_{oil} + \alpha_{wat}\right) \cdot \mu_{emulsion} + \left(\alpha_{gas} \cdot \mu_{gas}\right) \quad (12.41)$$

μemulsion = the viscosity of the oil-water emulsion at local conditions.\
β = ICDSTREN, the strength of the ICD as measured using the calibrated fluid.
q = the local flow rate through the ICD at local conditions adjusted for scaling based on ICDLEN and NSCAFAC parameters in Table 12.125.

In equation (12.40), αi represents the volume fraction of oil, water and gas at local conditions and ρi the density of the three phases. Similarity for equation (12.41), αi represents the volume fraction of oil, water and gas at local conditions and μi the viscosity of the three phases.

See also the WSEGAICD keyword in the SCHEDULE section for autonomous ICDs, that work in a similar fashion to how sprial ICDs work.

Example

The following example defines one producing well segment oil well (OP01) using the WELSPECS, WELSEGS COMPDAT and COMPSEGS keywords, followed by the WSEGSICD keyword to define the spiral inflow control devices for the well.

```
--
--        WELL SPECIFICATION DATA
--
-- WELL   GROUP      LOCATION  BHP    PHASE  DRAIN  INFLOW  OPEN  CROSS  PVT
-- NAME   NAME         I    J  DEPTH  FLUID  AREA   EQUANS  SHUT  FLOW   TABLE
WELSPECS
OP01      PLATFORM    10   10   1*     OIL                                  /
/
--
--        WELL CONNECTION DATA
--
-- WELL   --- LOCATION ---   OPEN   SAT   CONN   WELL   KH    SKIN   D     DIR
-- NAME    II  JJ  K1  K2    SHUT   TAB   FACT   DIA    FACT  FACT   FACT  PEN
COMPDAT
OP01       10  10   1   1    OPEN   1*    200.   0.5                        /
OP01       10  10   2   2    OPEN   1*    200.   0.5                        /
OP01       10  10   3   6    OPEN   1*    200.   0.4                        /
OP01       10  10   4   4    OPEN   1*    200.   0.4                        /
OP01       10  10   5   5    OPEN   1*    200.   0.4                        /
```

```
OP01      10  10   6   6   OPEN   1*    200.   0.4                          /

OP01       9  10   2   2   OPEN   1*    200.   0.4                          /
OP01       8  10   2   2   OPEN   1*    200.   0.4                          /
OP01       7  10   2   2   OPEN   1*    200.   0.4                          /
OP01       6  10   2   2   OPEN   1*    200.   0.4                          /
OP01       5  10   2   2   OPEN   1*    200.   0.4                          /

OP01      10   9   3   3   OPEN   1*    200.   0.4                          /
OP01      10   8   3   3   OPEN   1*    200.   0.4                          /
OP01      10   7   3   3   OPEN   1*    200.   0.4                          /
OP01      10   6   3   3   OPEN   1*    200.   0.4                          /
OP01      10   5   3   3   OPEN   1*    200.   0.4                          /

OP01       9  10   5   5   OPEN   1*    200.   0.4                          /
OP01       8  10   5   5   OPEN   1*    200.   0.4                          /
OP01       7  10   5   5   OPEN   1*    200.   0.4                          /
OP01       6  10   5   5   OPEN   1*    200.   0.4                          /
OP01       5  10   5   5   OPEN   1*    200.   0.4                          /

OP01      10   9   6   6   OPEN   1*    200.   0.4                          /
OP01      10   8   6   6   OPEN   1*    200.   0.4                          /
OP01      10   7   6   6   OPEN   1*    200.   0.4                          /
OP01      10   6   6   6   OPEN   1*    200.   0.4                          /
OP01      10   5   6   6   OPEN   1*    200.   0.4                          /
/
--
--       WELL SEGMENT SPECIFICATION DATA
--
-- WELL  NODAL       LEN     WELL    DEPH    PRESS   FLOW
-- NAME  DEPTH       TUBING  VOLM    OPTN    CALC    MODEL
WELSEGS
OP01     2512.5     2512.5  1.0E-5  ABS     HFA     HO                       /
--
--       SEG   SEG   BRAN    SEG     TUBING  NODAL   TUBE   TUBE     XSEC  VOL
--       ISTR  IEND  NO      NO      LENGTH  DEPTH   ID     ROUGH    AREA  SEG
           2     2    1       1      2537.5  2534.5  0.3    0.00010          /
           3     3    1       2      2562.5  2560.5  0.3    0.00010          /
           4     4    1       3      2587.5  2593.5  0.3    0.00010          /
           5     5    1       4      2612.5  2614.5  0.3    0.00010          /
           6     6    1       5      2637.5  2635.5  0.3    0.00010          /

           7     7    2       2      2737.5  2538.5  0.2    0.00010          /
           8     8    2       7      2937.5  2537.5  0.2    0.00010          /
           9     9    2       8      3137.5  2539.5  0.2    0.00010          /
          10    10    2       9      3337.5  2535.5  0.2    0.00010          /
          11    11    2      10      3537.5  2536.5  0.2    0.00010          /

          12    12    3       3      2762.5  2563.5  0.2    0.00010          /
          13    13    3      12      2962.5  2562.5  0.1    0.00010          /
          14    14    3      13      3162.5  2562.5  0.1    0.00010          /
          15    15    3      14      3362.5  2564.5  0.1    0.00010          /
          16    16    3      15      3562.5  2562.5  0.1    0.00010          /

          17    17    4       5      2812.5  2613.5  0.2    0.00010          /
          18    18    4      17      3012.5  2612.5  0.1    0.00010          /
          19    19    4      18      3212.5  2612.5  0.1    0.00010          /
          20    20    4      19      3412.5  2612.5  0.1    0.00010          /
          21    21    4      20      3612.5  2613.5  0.1    0.00010          /

          22    22    5       6      2837.5  2634.5  0.2    0.00010          /
          23    23    5      22      3037.5  2637.5  0.2    0.00010          /
          24    24    5      23      3237.5  2638.5  0.2    0.00010          /
```

```
          25    25    5      24      3437.5  2639.5  0.1    0.00010           /
          26    26    5      25      3637.5  2639.5  0.1    0.00010           /
/
--
--       COMPLETION SEGMENT SPECIFICATION DATA
--
-- WELL
-- NAME
COMPSEGS
OP01                                                                          /
--
--      --LOCATION--  BRAN  TUBING  NODAL   DIR  LOC    MID    COMP    ISEG
--       II  JJ  K1   NO    LENGTH  DEPTH   PEN  I,J,K  PERFS  LENGTH  NO.
         10  10   1    1    2512.5  2525.0                                  /
         10  10   2    1    2525.0  2550.0                                  /
         10  10   3    1    2550.0  2575.0                                  /
         10  10   4    1    2575.0  2600.0                                  /
         10  10   5    1    2600.0  2625.0                                  /
         10  10   6    1    2625.0  2650.0                                  /

          9  10   2    2    2637.5  2837.5                                  /
          8  10   2    2    2837.5  3037.5                                  /
          7  10   2    2    3037.5  3237.5                                  /
          6  10   2    2    3237.5  3437.5                                  /
          5  10   2    2    3437.5  3637.5                                  /

         10   9   3    3    2662.5  2862.5                                  /
         10   8   3    3    2862.5  3062.5                                  /
         10   7   3    3    3062.5  3262.5                                  /
         10   6   3    3    3262.5  3462.5                                  /
         10   5   3    3    3462.5  3662.5                                  /

          9  10   5    4    2712.5  2912.5                                  /
          8  10   5    4    2912.5  3112.5                                  /
          7  10   5    4    3112.5  3312.5                                  /
          6  10   5    4    3312.5  3512.5                                  /
          5  10   5    4    3512.5  3712.5                                  /

         10   9   6    5    2737.5  2937.5                                  /
         10   8   6    5    2937.5  3137.5                                  /
         10   7   6    5    3137.5  3337.5                                  /
         10   6   6    5    3337.5  3537.5                                  /
         10   5   6    5    3537.5  3737.5                                  /
--
--       MULTI-SEGMENT WELL ICD SEGMENT SPECIFICATION DATA
--
-- WELL  SEG   SEG   ICD      ICD  CAL  CAL   EML   EML    EML  SCAL  CAL   OPEN
-- NAME  ISTR  IEND  STRNEN   LEN  DEN  VISC  CRIT  TRANS  MAX  FAC   RATE  CLOSE
WSEGSICD
OP01       7    10  0.00025   1*  1.0  0.45  0.50  0.05   5.0   2    1*    OPEN /

OP01      12    15  0.00025   1*  1.0  0.45  0.50  0.05   5.0   2    1*    OPEN /

OP01      17    20  0.00025   1*  1.0  0.45  0.50  0.05   5.0   2    1*    OPEN /

OP01      22    22  0.00025   1*  1.0  0.45  0.50  0.05   5.0   2    1*    OPEN /
OP01      23    23  0.00025   1*  1.0  0.45  0.50  0.05   5.0   2    1*    OPEN /
OP01      24    24  0.00025   1*  1.0  0.45  0.50  0.05   5.0   2    1*    OPEN /
OP01      25    25  0.00050  10.0  1.0  0.45  0.50  0.05   5.0   2    1*    OPEN /
/
```

Branch number two uses one ICD for segments seven to ten, branch number three again uses only one ICD for segments 12 to 15 and similarly branch number three uses one ICD. The fifth and final branch has a total

of four ICDs with the last interval having a 10 foot length. Since NSCAFAC equals two for the ICDs, then the scale factor is equal to the length to ICDLEN, divided by the total length of the completions which supply the ICD. Where ICDLEN is defaulted, the default value of 39.37 ft will be used for field units and 12 m for SI units.

12.3.345 WSEGSOLV DEFINE MULTI-SEGMENT WELL ITERATIVE LINEAR SOLVER PARAMETERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The WSEGSOLV keyword defines the numerical control parameters for the iterative linear solver for multi-segment well looped flow paths, as defined by the WSEGLINK keyword in the SCHEDULE section. A looped segment results in the nodes of the two individual segments that are looped (or connected) having the same solution pressures and oil, water and gas flowing rates.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.346 WSEGTABL – ASSIGN MULTI-SEGMENT WELL VLP TABLES TO SEGMENTS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

WSEGTABL assigns previously defined Vertical Lift Performance ("VLP") tables as specified by the VFPROD keyword in the SCHEDULE section, to multi-segment well segments, as well as stipulating how the tables are to be applied.

The FLOWOPT parameter on the WELSEGS keyword in the SCHEDULE section sets the default multi-segment well model. FLOWOPT either activates the homogeneous model, that is all phases flow at the same velocity, or the Drift Flux Slip model. However, the WSEGFMOD keyword in the SCHEDULE section, can be used to set the flow model for a segment to either the homogeneous model or the Drift Flux Slip model, and addition a: VLP table allocated via the WSEGTABL keyword, or a specific model as defined by the WSEGVALV, WSEGFLIM and WSEGLABY keywords.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.347 WSEGVALV – DEFINE MULTI-SEGMENT WELL SUB-CRITICAL VALVE

Description

The WSEGVALV keyword defines a multi-segment well segment to be a sub-critical valve Inflow Control Device. ("ICD") as part of a completion for a multi-segment well. Note that the well must have been previously define by the WELSPECS and WELSEGS keywords in the SCHEDULE section and that the data for the keyword should be repeated for each multi-segment completion that contains a sub-critical valve ICD.

An ICD is a well completion component usually installed along the producing section of a well to minimize the unwanted water and gas breakthrough in an oil well, or early water production in a gas well, due to an uneven flow profile over the completed interval. Permeability variations over the producing interval cause the high permeability zones to produce higher quantities of fluids than the lower permeability zones and this uneven producing fluid profile may result in bypassed hydrocarbons. Secondly, for horizontal wells, the pressure loss from the "toe" to the "heel" of the well again results in an uneven fluid profile over the producing interval. In order to rectify this ICDs can be installed so that the well fluids have to flow through an ICD before entering the tubing; thus, creating an additional "designed" pressure loss.

A sub-critical valve ICD is a type of frictional ICD that adds an additional pressure loss by directing the fluid through a constriction before entering the tubing. The pressure drop is a function of the fluid density, the volumetric flow rate, and the diameter of the constriction. By placing various ICD's over the production interval one can design a completion that results in a more uniform producing fluid profile throughout the length of the producing interval.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WELNAME	A character string of up to eight characters in length that defines the well name for which a multi-segment well is being defined. Note that the well name (WELNAME) must have been declared previously using both the WELSPECS and WELSEGS keywords in the SCHEDULE section, otherwise an error may occur.			None
2	ISEG1	A positive integer greater than or equal to two and less than or equal to MXSEGS on the WSEGDIMS keyword in the RUNSPEC section that defines the segment containing the sub-critical valve.			None
3	ICDCV	A real positive value greater than zero that defines the dimensionless flow coefficient for the valve (Cv). This a vendor specific value for a given vendor's ICD.			
		dimensionless	dimensionless	dimensionless	None
4	AREAREST	A real positive value that defines the cross-sectional area of flow in the restricted section of the valve (Ar) and should have a minimum value of 1.0 x 10-10. AREAREST is used to convert the segment volumetric flow rate into the flow velocity at the constriction (υr).			
		ft^2	m^2	cm^2	None

No.	Name	Description			Default
		Field	Metric	Laboratory	
5	SEGLEN	A real positive value greater than or equal to zero that defines the additional pipe length for the frictional pressure drop (L). If set to zero then there is no additional pressure loss due to friction, whereas, if set to the default (1*), then the segment pipe length is calculated from the corresponding WELSEGS keyword.			
		ft	m	cm	Defined
6	ID	A real positive value that defines the pipe internal diameter of the segment used to calculate the pressure drop due to friction (D). The value is used to replace the segment pipe internal diameter defined on the WELSEGS keyword in record 2-7, also named ID. If ID is defaulted on this keyword then the equivalent value on the WELSEGS keyword will be used instead. Note for non-circular pipe segments use the equivalent diameter instead, that is: $Equivalent\ ID = \frac{4.0 \times (Cross - Sectional\ Area)}{Perimeter}$			
		feet	m	cm	Defined
7	EPSILON	A real positive value that defines the pipe absolute roughness for this segment. The value is used to replace the segment pipe absolute roughness defined on the WELSEGS keyword in record 2-8, also named EPSILON. If EPSILON is defaulted on this keyword then the equivalent value on the WELSEGS keyword will be used instead.			
		feet	m	cm	Defined
8	AREAPIPE	A real positive value that defines the cross-sectional area of flow in the pipe (Ap), as opposed to the restricted section of the valve (Ar). AREAPIPE is used to convert the segment volumetric flow rate into the flow velocity through the pipe (υp). The value is used to replace the segment pipe cross-sectional area of flow defined on the WELSEGS keyword in record 2-9, named XAREA. If AREAPIPE is defaulted on this keyword then the equivalent value on the WELSEGS keyword will be used instead.			
		ft^2	m^2	cm^2	Defined
9	STATUS	A character string of length four that defines the ICD's operational status, STATUS should be set to one of the following character strings: 1) OPEN: the ICD connection is are open to flow. 2) SHUT: the ICD connections is closed to flow (shut-in).			OPEN
10	AREAMAX	A real positive value that defines the maximum cross-sectional area of flow in the restricted section of the valve (Amax). AREAMAX is used to convert the segment volumetric flow rate into the maximum flow velocity at the constriction (υr). If defaulted then AREAPIPE will be used if defined, otherwise XAREA (item 2-9) on the WELSEGS keyword will be used.			Defined

No.	Name	Description			Default
		Field	Metric	Laboratory	
		ft^2	m^2	cm^2	

Notes:

1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".

Table 12.126: WSEGVALV Keyword Description

The total number of wells should be defined via the WELLSDIMS keyword and the number of multi-segment wells should be declared on the WSEGDIMS keyword, both keywords are in the RUNSPEC section. In addition, the WELSPECS keyword should be used to define wells, the COMPDAT keyword to define the well completions for both ordinary wells and multi-segment wells, and the COMPSEGS keyword to define a multi-segment segment completions. Finally, the WSEGVALV keyword can then be use to define ICD connections for the well. All the aforementioned keywords are described in the SCHEDULE section.

OPM flow calculates the pressure drop across the ICD using a homogeneous sub-critical flow through a constriction model. The model consists of two variables, the pressure drop due to the constriction, $\delta P_{restriction}$, and the pressure drop due to friction, $\delta P_{friction}$, as shown in equation (12.42).

$$\delta P = \delta P_{restriction} + \delta P_{friction}$$

where

$$\delta P_{restriction} = C_1 \frac{\rho \upsilon_r^2}{2 C_v^2} \qquad (12.42)$$

$$\delta P_{friction} = 2 C_2 f \frac{L}{D} \upsilon_p^2$$

Where:

C_1 and C2	=	Unit conversion constants as listed in Table 12.127.
C_v	=	Vendor supplied dimensionless flow coefficient for the valve.
D	=	Diameter of the pipe.
f	=	Fanning friction factor.
L	=	Segment pipe length.
ρ	=	Fluid mixture density.
υr	=	The flow velocity of the mixture through the constriction.
υp	=	The flow velocity of the mixture through the *segment pipe.*

Conversion Factor Constants and Variable Units					
No.	**Name**	**Field**	**Metric**	**Laboratory**	**Property**
1	C_1	2.159 x 10-4	1.0 x 10-5	9.869×10^{-7}	Constant
2	C_2	2.892×10^{-14}	1.340×10^{-15}	7.615×10^{-14}	
3	Density	lb/ft^3	kg/m^3	gm/cc	Units
4	Pressure	psia	bars	atm	
5	Velocity	ft/s	m/s	cm/s	
6	Volumetric Flow	ft^3/day	m^3/day	cc/hour	

Table 12.127: WELVALV Conversion Factor Constants and Units

In addition as both υr and υp are dependent on their respective cross-section areas then the volumetric flow (q) through the device requires that:

$$q = \upsilon_r A_r = \upsilon_p A_p \tag{12.43}$$

Substituting equation (12.43) for $\delta P_{restriction}$ in equation (12.42) one obtains:

$$\delta P_{restriction} = C_2 \frac{\rho q^2}{2 C_v^{\,2} A_r^2} \tag{12.44}$$

Where:

A_r = Cross-sectional area of the constriction.
C2 = Unit conversion constants as listed in Table 12.127.
C_v = Vendor supplied dimensionless flow coefficient for the valve.
ρ = Fluid mixture density.
q = Volumetric flow rate.

The base strength of the device, K is defined using equation (12.44) as follows:

$$K = \frac{C_2}{2 C_v^{\,2} A_r^2} \tag{12.45}$$

Note if K is greater than 0.1 then the device will be shut.

The setting of the device, that is how open the device is; is related to the restricted area and the maximum restricted area of the device, that is:

$$Seting\,of\,Device = \frac{A_r}{A_{max}} \tag{12.46}$$

Where:

A_r = Cross-sectional area of the constriction.
Amax = Maximum cross-sectional area of the constriction.

Example

The following example is based on one producing well segment oil well (OP01) using the WELSPECS, WELSEGS COMPDAT and COMPSEGS keywords, as per the WSEGSICD keyword example (*Example*), and is therefore not repeated here.

```
--
--        MULTI-SEGMENT WELL ICD SEGMENT SPECIFICATION DATA
--
-- WELL   SEG   DEVICE   AREA    PIPE    PIPE   PIPE   PIPE   OPEN   MAX
-- NAME   NO    CV       REST    LENG    ID     EPSI   AREA   SHUT   AREA
WSEGVALV
OP01        7   0.960    0.012   1*      1*     1*     1*     OPEN   1*   / DEFAULT VALUES
OP01       12   0.960    0.012   1*      1*     1*     1*     OPEN   1*   / TAKEN FROM
OP01       17   0.960    0.012   1*      1*     1*     1*     OPEN   1*   / WELSEGS KEYWORD

OP01       22   0.850    0.100   1*      1*     1*     1*     OPEN   1*   /
OP01       23   0.850    0.100   1*      1*     1*     1*     OPEN   1*   /
OP01       24   0.850    0.100   1*      1*     1*     1*     OPEN   1*   /
OP01       25   0.850    0.100   1*      1*     1*     1*     OPEN   1*   /
/
```

Here segments 7, 12 and 17 have the same type of sub-critical valves with their pipe properties taken from the WELSEGS keyword used to define well OP01 as a multi-segment well. Similarly, segments 22 to 25 have the same ICD properties, and again the pipe properties are taken from the WELSEGS keyword.

12.3.348 WSKPTAB - ASSIGN WELL POLYMER MOLECULAR MODEL SKIN TABLES

Description

The WSKPTAB keyword assigns the well polymer molecular water and polymer skin tables to water injection wells in OPM Flow's Polymer Molecular Weight Transport option, that uses the polymer molecular weight in calculating the polymer viscosity, as well as accounting for formation damage due to the water and polymer injection, by adjusting the wellbore skin pressure. This keyword should only be used if the POLYMER and POLYMW keywords in the RUNSPEC section are also activated. The keyword assigns the water SKPRWAT tables, that are defined via the SKPRWAT keyword in the PROPS section, that are used to calculable the wellbore skin pressure during water injection. As well as the polymer SKPRPOLY tables, that are defined via the SKPRPOLY keyword in the PROPS section, that are used to calculable the wellbore skin pressure during polymer injection.

Note

This is an OPM Flow specific keyword that employs an alternative polymer flood model based on a Polymer Molecular Weight Transport equation, that is not available in the commercial simulator.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WELNAME	A character string of up to eight characters in length, that defines the water injection well name, for which the well water injection skin table, SKKPRWAT, and the polymer skin injection table, SKPRPOLY, are to be assigned. Note that the well name (WELNAME) must have been declared previously using the WELSPECS keyword in the SCHEDULE section, otherwise an error may occur.			None
2	SKPRWAT	A positive integer value that defines the corresponding SKPRWAT table to be allocated to the water injection well. A value less than or equal to zero means that no SKPRWAT table is allocated to the well			0
3	SKPRPOLY	A positive integer value that defines the corresponding SKPRPOLY table to be allocated to the water injection well. A value less than or equal to zero means that no SKPRPOLY table is allocated to the well			0

Notes:

1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".

Table 12.128: WSKPTAB Keyword Description

See also the SKPRWAT and SKPRPOLY keywords in the PROPS section, that describe the relationship of the wellbore skin pressure with respect to the injected water and polymer throughput and fluid velocity, for the simulator's Polymer Molecular Weight Transport option. As well as the PLYMWINJ and PLYVMH keywords, also in the PROPS section, that are the additional keywords required for the Polymer Molecular Weight Transport option.

The WPMITABJ keyword in the SCHEDULE section may be used to assign the PLYMWINJ keyword in the PROPS section. Where PLYMWINJ, describes the relationship of the injected polymer molecular weight as a function of polymer throughput and polymer velocity, for the simulator's Polymer Molecular Weight Transport option

Note that the standard polymer property data keywords: PLYROCK, PLYADS, PLYMAX, etc., are still required to fully describe the polymer fluid.

Example

Given NTSKWAT equals two and NTSKPOLY equals three on the PINTDIMS keyword in the RUNSPEC section, then:

```
--
--       ASSIGN WELL POLYMER MOLECULAR MODEL SKIN TABLES
--
-- WELL  SKPRWAT   SKPRPOLY
-- NAME  TABLE     TABLE
WSKPTAB
WI01     1         1                                          /
WI02     1         3                                          /
WI03     2         2                                          /
/
```

Assigns SKPRWAT table one to wells WI01 and WI02 and table two to WI03, and SKPRPOLY tables one, two and three to wells WI01, WI03, and WI02, respectively.

12.3.349 WSOLVENT - DEFINE GAS INJECTION WELL SOLVENT FRACTION

Description

WSOLVENT defines a gas injection well's solvent faction in the injection stream that is to be used when the Solvent option has been activated by the SOLVENT keyword in the RUNSPEC section.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WELNAME	A character string of up to eight characters in length that defines the well name of a gas injection well for which the solvent fraction data is being defined. Note that the well name (WELNAME) must have been declared previously using the WELSPECS keyword in the SCHEDULE section, otherwise an error may occur.			None
4	SOLFRA	A real positive value greater than or equal to zero and less than or equal to one that defines the fraction of solvent in the gas well's injection stream.			
		fraction	fraction	fraction	None
Notes: 1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".					

Table 12.129: WSOLVENT Keyword Description

Gas injection wells that are not declared via this keyword have their solvent fractions set to zero.

See also the GCONINJE keyword to define a group's injection targets and constraints, and the WCONINJE keyword to define an injection well's targets and constraints. All the aforementioned keywords are described in the SCHEDULE section.

Example

The following example defines the solvent fractions for three gas injection wells for when the solvent option has been activated by the SOLVENT keyword in the RUNSPEC section.

```
--
--       DEFINE GAS INJECTION WELL SOLVENT FRACTION
--
-- WELL  SOLVENT
-- NAME  FRACTION
--       --------
WSOLVENT
GI01     0.0000                                    /
GI02     0.5000                                    /
GI03     0.5000                                    /
/
```

The solvent fraction for the GI01 gas injector is set to zero and both GI02 and GI03 gas injectors have solvent fraction values of 0.5 for their injection streams.

12.3.350 WSURFACT - DEFINE WATER INJECTION WELL SURFACTANT CONCENTRATION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

WSURFACT defines a water injection well's surfactant concentration in the injection stream that is to be used when the Surfactant phase has been activated by either the SURFACT or SURFACTW keywords in the RUNSPEC section.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WELNAME	A character string of up to eight characters in length that defines the well name of a gas injection well for which the solvent fraction data is being defined. Note that the well name (WELNAME) must have been declared previously using the WELSPECS keyword in the SCHEDULE section, otherwise an error may occur.			None
2	SURCON	A real positive value that defines the surfactant concentration of the well's injection stream.			
		lb/stb	kg/sm^3	gm/scc	None

Notes:

1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".

Table 12.130: WSURFACT Keyword Description

Water injection wells that are not declared via this keyword have their surfactant concentrations set to zero.

See also the GCONINJE keyword to define a group's injection targets and constraints, and the WCONINJE keyword to define an injection well's targets and constraints. All the aforementioned keywords are described in the SCHEDULE section.

Example

The following example defines the surfactant concentrations for three water injection wells for when the surfactant phase option has been activated by either the SURFACT or SURFACTW keywords in the RUNSPEC section. Here the surfactant concentration has been set to 0.200 for all three wells.

```
--
--       DEFINE WATER INJECTION WELL SURFACTANT CONCENTRATION
--
-- WELL  SURFACT
-- NAME  SURCON
--       --------
WSURFACT
WI01     0.2000                                          /
WI02     0.2000                                          /
WI03     0.2000                                          /
/
```

12.3.351 WTADD – ADD A CONSTANT TO A WELL TARGET OR CONSTRAINT

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, WTADD, adds a constant to a previously define well's target or constraint, as stated on the WCONPROD, WCONINJE, or WELTARG keywords, but not for the history matching wells using the WCONHIST or WCONINJH keywords. All the aforementioned keywords are in the SCHEDULE section. The constant can be positive or negative.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.352 WTEMP – DEFINE AN INJECTION WELL'S FLUID TEMPERATURE

Description

The WTEMP keyword defines the temperature of the injection fluid being injected by an injection well.

This keyword can only be used if OPM Flow's thermal option has been activated by the THERMAL keyword in the RUNSPEC section. Note this is different to the commercial simulator that uses the TEMP keyword in the RUNSPEC section to activate the black-oil thermal model.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WELNAME	A character string of up to eight characters in length that defines the well name for an injection well for which the injection well fluid's temperature data is being defined. Note that the well name (WELNAME) must have been declared previously using the WELSPECS keyword in the SCHEDULE section, otherwise an error may occur.			None
2	TEMP	A real positive value greater than zero that defines the temperature of the injected fluid.			
		°F	°C	°C	None

Notes:

1) Injection wells that are not declared via this keyword have their injection fluid temperatures set to zero degrees in the run's units.
2) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".

Table 12.131: WTEMP Keyword Description

See also the GCONINJE keyword to define a group's injection targets and constraints, and the WCONINJE keyword to define an injection well's targets and constraints. All the aforementioned keywords are described in the SCHEDULE section.

Example

The following example defines the injected fluid temperatures for three water injection wells for when the thermal option has been activated by the THERMAL keyword in the RUNSPEC section.

```
--
--       DEFINE INJECTION WELL FLUID TEMPERATURE
--
-- WELL  FLUID
-- NAME  TEMP.
--       --------
WTEMP
WI01     39.00                                    /
WI02     37.00                                    /
WI03     39.00                                    /
/
```

Here wells WI01 and WI03 inject water with a water temperature of 39 oF and well WI02's injection water temperature is 37 oF.

12.3.353 WTEMPQ – OUTPUT WELL NAMES AND WELL LISTS TO THE PRINT FILE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The WTEMPQ prints out a user defined selected list of currently defined wells and well lists to the print file (*.PRT). The keyword allows for sub-setting the well names etc., using the normal well and well list naming conventions. For example to list all wells beginning with the characters "OP" then one would use "OP*" as the well name on this keyword.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.354 WTEST – WELL TESTING CRITERIA FOR RE-OPENING CLOSED WELLS

Description

The WTEST keyword outlined the testing procedures to be applied to wells that are closed for various reason to see if the wells are capable flowing under the current operating conditions. The keyword can be applied to single wells or groups of wells.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WELNAME	A character string of up to eight characters in length that defines the well name for which the well connection data is being defined. Note that the well name (WELNAME) must have been declared previously using the WELSPECS keyword in the SCHEDULE section, otherwise an error may occur.			None
2	TIME	A real value greater than to zero that defines the period before another test is performed, for example if TIME is set equal to 365.25 (days), the test is performed every year.			
		days	days	hours	None
3	TEST	A character string of up to five characters that defines the reason the well was closed. If a well was closed for one of the criteria then the well is tested to see if it can be put back on production. The characters that can be used to define TEST are as follows: 1) P: meaning the well was closed due to a bottom-hole or tubing head pressure limit, or other physical limit then the well is tested to see if it can flow, if it can, then it is put back on production, otherwise it remains closed. 2) E: meaning the well was closed due to a well or a well connection economic constraint then the well is tested to see if it can flow, if it can then it is put back on production, otherwise it remains closed. 3) Here, if the well has at least one connection open then the well is opened. 4) If a well has no open connections, then those that have been closed due to economic limit test ("ELT"), then ALL such connections are individually tested and reopened if they meet the ELT criteria as defined by the WECON, CECON and WECONINJ keywords in the SCHEDULE section. And the well is opened if there is at least one open connection. Where the ELT applies to ORAT, GRAT, WCUT, GOR and WGR etc. 5) G: meaning the well was closed due to a group economic constraint, as per the GECON, GECONT, GCONCAL, GCONPRI, GCONPRPD, and GCONSALE, keywords in the SCHEDULE section, then the well is tested to see if it can flow, if it can, then it is put back on production, otherwise it remains closed. 6) D: not used by OPM Flow. 7) C: not used by OPM Flow. The default value is an empty string " " that switches off testing. ***Note that only the P and E options are currently supported in OPM Flow.***			" "

No.	Name	Description			Default
		Field	Metric	Laboratory	
4	NTIME	A positive integer greater than or equal to zero that define the number of times a well can be tested. The default value of zero means an infinite number of times.			0
5	START	A real positive value that defines the start up time used to prorate the rate at which the well is put back on production. If START is large compared to the time step size, then the well is brought on gradually, if it is less than, then the well is opened faster. The rate is prorated based on the following: $Q_i = Q_{final} \times \left(\frac{T_{End\ of\ Time\ Step} - T_{opened}}{START} \right)$ The default value of 0.0 means the well is opened immediately.			
		days	days	hours	0.0
Notes: 1) The keyword is followed by any number of records and each record is terminated by a "/" and the keyword should be terminated by a "/".					

Table 12.132: WTEST Keyword Description

See also the WELSPECS keyword to define a wells shut-in or stop options, WECON for setting a well's economic criteria, GCONPROD and GCONINJE for group controls, and GECON for setting a group's economic criteria. All the aforementioned keywords are described in the SCHEDULE section.

Example

The following example defines test criteria for all gas wells ("GP*') and three oil wells (OP01, OP02, and OP03).

```
--
--
--       WELL TESTING CRITERIA FOR RE-OPENING CLOSED WELLS
--
-- WELL  TST    TST    NO.    STRT
-- NAME  INTV   TYPE   TSTS   TIME
-- ----  ----   ----   ----   ----
WTEST
'GP*'    365.25 P      5      0.0
OP01     30.0   PEG    0      0.0                                    /
OP02     30.0   PEG    0      0.0                                    /
OP03     30.0   PEG    0      0.0                                    /
/
```

All the gas wells are test annually if they have been shut-in due to a bottom-hole or tubing head pressure limit, are tested five times after they have been closed, and are opened up immediately. The oil wells are tested every 30 days if they have been closes due bottom-hole or tubing head pressure limit, a well economic limit or a group economic limit. All the oil wells are tested an infinite amount of times and are opened up immediately. ***Note that only the*** P and ***E options*** are ***currently supported in OPM Flow.***

12.3.355 WTHPMAX – DEFINE A WELL'S MAXIMUM FLOWING THP FOR SHUT-IN

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

WTHPMAX stipulates a well's maximum flowing Tubing Head Pressure ("THP"), above which the well will be shut-in. The facility is useful if the THP exceeds the wellhead maximum design pressure, which can occur if excessive gas invades the wellbore. In addition to setting the maximum THP, the keyword defines the criteria for re-testing the well to see if the THP has fallen below the maximum value.

This keyword is not supported by OPM Flow but would change the results if supported so the simulation will be stopped.

12.3.356 WTMULT – MULTIPLE A WELL TARGET OR CONSTRAINT BY A CONSTANT

Description

This keyword, WTMULT, multiplies a defined well's target or constraint by a constant, for the target and constraints previously stipulated on the WCONPROD, WCONINJE, or WELTARG keywords, but not for the history matching wells using the WCONHIST or WCONINJH keywords. All the aforementioned keywords are in the SCHEDULE section. The constant should be positive value.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WELNAME	A character string of up to eight characters in length that defines the well name for which the well production targets and constraints flow streams (TARGET) are being adjusted by the CONSTANT parameter. Note that the well name (WELNAME) must have been declared previously using the WELSPECS keyword in the SCHEDULE section, otherwise an error may occur.			None
2	TARGET	A defined character string that declares the production or injection flow stream for the well, that will be adjusted by multiplying the current value by the CONSTANT parameter. TARGET should be set to one of the following character strings: 1) ORAT: the well's surface oil production rate will be adjusted. 2) WRAT: the well's surface water production rate will be adjusted. 3) GRAT: the well's surface gas production rate will be adjusted. 4) LRAT: the well's surface liquid (oil plus water) production rate will be adjusted. 5) CRAT: the well's linearly combined maximum surface rate, as per the LINCOM keyword in the SCHEDULE section, will be adjusted. 6) RESV: the well's in situ reservoir volume rate will be adjusted. 7) BHP: the well's bottom-hole pressure will be adjusted. 8) THP: the well's tubing head pressure will be adjusted. In this case a vertical performance table must have been previously assigned to the well via the WCONPROD or WCONINJE keywords. The tables are entered via the VFPINJ or the VFPPROD keywords. All the keywords are in the SCHEDULE section. 9) LIFT: the well's artificial lift quantity will be adjusted. Again, as for the THP, a vertical performance table must have been assigned to the well. 10) GUID: the well's guide rate will be adjusted. Only wells under group control and have a guide rate stipulated by the WGRUPCON keyword in the SCHEDULE section, may use this option. 11) CVAL: the well's calorific rate will be adjusted. 12) NGL: the well's natural gas liquid rate will be adjusted. Note that the CRAT, CVAL and the NGL options are not available in OPM Flow and should not be used.			None

No.	Name	Description			Default
		Field	Metric	Laboratory	
3	CONSTANT	A real value that is used to multiply the current TARGET flow stream by to get the adjusted flow stream rate.			
		dimensionless	dimensionless	dimensionless	None
4	NTIME	A positive integer greater than or equal to one that defines the number of report time steps for User Defined Arguments (UDA) variables only, that the TARGET flow stream is multiplied by to get the adjusted flow stream. The default value of one means that the multiplication is applied only at the current time step for the UDA variable. This option is not supported by OPM Flow.			
		dimensionless	dimensionless	dimensionless	1.0
Notes: 1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".					

Table 12.133: WTMULT Keyword Description

See also the WELTARG and WELCNTL keywords in the SCHEDULE section, that can be used to reset the well's target and constraints of both rates and pressures, as well as the well's control mode.

Example

The example shows three oil wells having the flow streams adjusted.

```
--
-- WELL TARGET/LIMIT MULTIPLIER
--
--  WELL    WELL   TARGET      REPORT
--  NAME    TARG   MULTIPLIER  TIMES
WTMULT
OP01        ORAT   0.90               /
OP02        BHP    0.95               /
OP03        LIFT   1.25               /
/
```

Well OP01 has its current oil rate multiplied by 0.90, well OP02 has its bottom-hole pressure multiplied by 0.95, and well OP03 has its artificial lift quantity increased by 1.25 times.

12.3.357 WTRACER – DEFINE AN INJECTION WELL'S TRACER CONCENTRATION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The WTRACER keyword defines the tracer concentration of the injection fluid being injected by an injection well. This keyword should only be used if the Tracer option has been invoked by the TRACERS keyword in the RUNSPEC section and the tracers have be declared via the TRACER keyword in the PROPS section.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WELNAME	A character string of up to eight characters in length that defines the well name of an injection well for which the tracer fraction data is being defined. Note that the well name (WELNAME) must have been declared previously using the WELSPECS keyword in the SCHEDULE section, otherwise an error may occur.			None
2	NAME	A three letter character string defining the tracer's name which has previously been defined via the TRACER keyword in the PROPS section. Note it is best to void names beginning with the letters F, S, and T as these names may create naming issues in post-processing software.			None
3	TRCON	A real positive value that defines the tracer concentration of the well's injection stream.			
		Liquid: 1/stb Gas: 1/Mscf	Liquid: 1/sm^3 Gas: 1/sm^3	Liquid: 1/scc Gas: 1/scc	None
4	TRCUM	A real positive value that defines the cumulative tracer concentration factor of the well's injection stream. This feature is currently not supported by OPM Flow.			
		Liquid: 1/stb Gas: 1/Mscf	Liquid: 1/sm^3 Gas: 1/sm^3	Liquid: 1/scc Gas: 1/scc	None
5	GRPNAME	A character string of up to eight characters in length that defines the group from which the produced tracer concentration should be used for the well's injection stream. GRPNAME must have been previously defined via the GCONPROD keyword in the SCHEDULE section, unless the FIELD group has been specified here. Note if GRPNAME is not defined then TRCON will be used for the tracer concentration of the well's injection stream. This feature is currently not supported by OPM Flow.			None

Notes:

1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".

Table 12.134: WTRACER Keyword Description

Injection wells that are not declared via this keyword have their tracer concentrations set to zero.

See also the GCONINJE keyword to define a group's injection targets and constraints, and the WCONINJE keyword to define an injection well's targets and constraints. All the aforementioned keywords are described in the SCHEDULE section.

Example

The following example defines the tracer concentrations for two gas injectors and three water injection wells, with water injection well WI02 having no 'WAT' tracer injected in the water phase.

```
--
--       DEFINE CONCENTRATION OF TRACERS IN THE INJECTION STREAMS,
--       INJECTION TRACER CONCENTRATIONS NOT DEFINED USING THE WTRACER
--       KEYWORD ARE ASSUMED TO BE ZERO.
--
-- WELL  NAME    TRACER  TRACER  TRACER
-- NAME  TRACER  CONC    CUM     GROUP
WTRACER
GI01     'GAS'   1.0                                  /
GI02     'GAS'   1.0                                  /

WI01     'WAT'   1.0                                  /
WI02     'WAT'   0.0                                  /
WI03     'WAT'   1.0                                  /
/
```

Note the terminating "/" for the keyword.

12.3.358 WVFPDP – MODIFY WELL BHP OBTAINED FROM VFP TABLES

Description

The WVFPDP keyword modifies a well's Bottom-Hole Pressure ("BHP") estimated by the simulator by interpolation of the Vertical Flow Performance ("VFP") tables. The production VFP tables are entered via the VFPPROD keyword and the injection tables by the VFPINJ keyword; both keywords are in the SCHEDULE section.

Note that simulator automatically adjusts the interpolated BHP to account for hydrostatic head using the density of the wellbore fluid and the difference between a well's BHP reference depth, as per the BHPREF parameter on the WELSPECS or WELSPECL keywords in the SCHEDULE section, and the VFPREF parameter reference depth on the VFPPROD and VFPINJ keywords. Thus, WVFPDP applies an additional adjustment in order to match a well's flow rate to a given tubing head pressure, by adjusting the BHP.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WELNAME	A character string of up to eight characters in length that defines the well name for which VFP interpolated BHP adjustment is to be applied. Note that the well name (WELNAME) must have been declared previously using the WELSPECS and WCONPROD (or WCONINJE) keywords in the SCHEDULE section, otherwise an error may occur.			None
2	DELTAP	A real positive or negative value that is added to the VFP interpolated BHP value (BHPVFP). A positive value of DELTAP increases the BHP and therefore makes a production well less productive; whereas, a negative value is subtracted from the BHP and therefore increases the productivity of a production well. Consequently, the opposite effect occurs for injection wells, that is, a positive value of BHPVFP increases the BHP and therefore increases an injection well's injectivity; whereas, a negative value is subtracted from the BHP and therefore decreases the injectivity of an injection well.			
		psia	barsa	atma	0.0
3	MULTP	MULTP is a real positive or negative value that scales the tubing pressure loss by the following equation; $BHP_{Adjusted} = THP + MULTP\left(BHP_{VFP} - THP\right)$ Thus, a MULTP value greater then 1.0 increases the BHP and therefore makes a production well less productive; whereas, a value less than 1.0 increases the productivity of a production well. Consequently, the opposite effect occurs for injection wells			
		dimensionless	dimensionless	dimensionless	1.0

Notes:

1) The keyword is followed by any number records with each record terminated by a "/" and the keyword should be terminated by a "/".

Table 12.135: WVFPDP Keyword Description

Example

The following example below shows three oils operating under THP control.

```
-- ------------------------------------------------------------------------
-- 01 JAN 2000 START OF SCHEDULE SECTION
-- ------------------------------------------------------------------------
--
--       WELL PRODUCTION WELL CONTROLS
--
-- WELL  OPEN/  CNTL   OIL    WAT    GAS    LIQ    RES    BHP    THP   VFP    VFP
-- NAME  SHUT   MODE   RATE   RATE   RATE   RATE   RATE   PRES   PRES  TABLE  ALFQ
WCONPROD
OP01     OPEN   THP    1*     1*     1*    5000   1*     750.0  500.  9      1* /
OP02     OPEN   THP    1*     1*     1*    5000   1*     750.0  500.  9      1* /
OP03     OPEN   THP    1*     1*     1*    5000   1*     750.0  500.  9      1* /
/
--
--       WELL VFP BHP-THP CORRECTION DATA
--
--  WELL BHP     BHP
--  NAME DELTAP  MULTP
WVFPDP
OP01     20.0    1*                                    /
OP01     -5.0    1*                                    /
OP01      0.0    1.10                                  /
/
```

Well OP01 has a delta pressure correction of 20 psia applied to it's BHP resulting in a reduction in the well's productivity for the given 500.0 psia THP operating target. For well OP02, the well's productivity is increased by subtracting 5.0 psia from the BHP. And finally for well OP03, the MULTP value of 1.10 decreases the well's productivity by increasing the pressure loss between the THP and BHP by 10%.

12.3.359 WVFPEXP – DEFINE WELL VFP INTERPOLATION OPTIONS

Description

This keyword, WVFPEXP, defines how Vertical Flow Performance ("VFP") tables are interpolated and can be used to resolve certain issues with wells operating under tubing head pressure control. For example, setting the VFP table to interpolate explicitly, that is using the previous time step results of the gas and water ratios for an oil well, may improve convergence. The default is to use implicit interpolation that uses the current time step values and may result in solution convergence oscillations in solving the linear equations. The WCONPROD keyword is used to allocate the VFPPROD tables to specific production wells. Note that one VFP table can be allocated to one or more wells; however, WVFPEXP is applied to a well's allocated VFP table, not to all wells that use the same table, unless specifically requested. All the aforementioned keywords are in the SCHEDULE section.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WELNAME	A character string of up to eight characters in length that defines the well name for which the well's VFP interpolation options are being defined. Note that the well name (WELNAME) must have been declared previously using the WELSPECS keyword in the SCHEDULE section, otherwise an error may occur.			None
2	IMPEXP	A defined character string of length three that defines the how the well's VFPPROD tables are to be interpolated and should be set to one of the following character strings: 1) IMP: For this option the simulator interpolates a well's VFP table implicitly, that is the latest gas fraction (GOR, GLR, etc.), water fraction (WCUT, WOR etc.), and ALQ is used to determine the required pressure. This is the most accurate option but may lead to oscillating behavior during the Newton/Linear iterations. This is the default behavior. 2) EXP: This options forces the simulator to use the previous time step values of a well's gas fraction, water fraction, and ALQ to interpolate a well's VFP table in order to determine the required pressure. This is less accurate than the IMP option but minimizes any oscillating behavior during the Newton/Linear iterations. If a well's WCUT and GOR is varying as a function of BHP inside a Newton/Linear iteration (as for example when gas cusping or water coning is occurring), the implicit lookup of the VFP tables might indicate that the well has died, due to the interpolation/extrapolation. If a well dies using implicit WCUT and GOR VFP lookup then the simulator automatically switches to explicit (previous time step) VFP lookup to prevent the well from premature closure, and writes a message to the screen and print file stating the fact. Note the explicit treatment is just for the VFP lookup only. Using the IMPEXP option allows one to select wells that should use either the implicit or explicit lookup option, which may be useful in improving run time performance. Note that if the default value of IMP is used, the simulator will still switch to explicit VFP lookup to prevent a well from premature closure if the condition occurs.			IMP

No.	Name	Description			Default
		Field	Metric	Laboratory	
3	STATUS	A defined character string that defines if the well's operating condition should be checked to see if the VFP table lookup indicates if the well is operating on the flat or horizontal part of stabilized portion of the VLP table (curve), and should be set to one of the following character strings: 1) NO: For this option the check is not performed. 2) YES: In this case the check is performed. Hence, if a well under THP control is found to operating on the horizontal stabilized portion of the VLP curve, then the well is SHUT or STOPPED depending on the AUTO parameter on the WELSPECS keyword. This option is specific to the commercial simulator's VFPi program that can generate VFPF tables where the unstable part of the VFP curves to the left of the minimum is replaced with a horizontal line at the minimum BHP value. This option is not supported by OPM Flow.			NO
4	CONTROL	A defined character string that defines the behavior of rate control wells operating on the unstabilized portion of the VFP table, and should be set to one of the following character strings: 1) NO: This option does not prevent changes to a well's operating mode when the well is constrained to be operating on the unstabilized portion of the VFP table. This option may cause the well control to oscillate between rate and THP control during the Newton/Linear iterations. 2) YES1: For this option the well is prevented from changing from rate control to THP control provided it can produce at a higher rate under THP control. In this case the well will continue to produce at the required rate even though the calculated THP may be below the well's minimum THP value. Note however, the well will still be allowed to die if the IPR does not intersect the VFP curve. In the commercial simulator this option will also print a message the first time a well is prevented from changing its control mode, whereas OPM Flow always prints a message every time a well is prevented from changing its control mode. 3) YES2: This option behaves the same as the YES1 option, except a message is printed every time a well is prevented from changing its control mode. The default value of NO allows wells to die when they are constrained by a lower rate that forces them to operate on the unstable section of the VFP curve. This can occur if a well's rate is being set to match its group target rate or constraint limit. Using either the YES1 or YES2 option will help prevent the "hunting between control modes" issue or the well prematurely being shut-in.			NO
5	EXTRAP	The EXTRAP parameter is a defined character string in the commercial composition simulator that declares how the extrapolation of a well's gas fraction, water fraction, and ALQ values is to be conducted. This option is not supported by OPM Flow.			WG

Notes:

1) The keyword is followed by any number of records with each record terminated by a "/" and the keyword should be terminated by a "/".

Table 12.136: WVFPEXP Keyword Description

Example

The following example defines two oil wells using the WELSPECS and WCONPROD keywords, together with the WVFPEXP keyword that declares how the VFPPROD table lookup should be performed.

```
--
-- WELL SPECIFICATION DATA
--
-- WELL     GROUP      LOCATION  BHP    PHASE  DRAIN  INFLOW  SHUT  CROSS  PRESS
-- NAME     NAME        I    J   DEPTH  FLUID  AREA   EQUA.   IN    FLOW   TABLE
WELSPECS
OP01      PLATFORM     14   13   1*      OIL   1*     STD     SHUT   NO    1*  /
OP02      PLATFORM     28   96   1*      OIL   1*     STD     SHUT   NO    1*  /
/
--
--       WELL PRODUCTION WELL CONTROLS
--
-- WELL  OPEN/  CNTL   OIL    WAT    GAS    LIQ    RES    BHP   THP   VFP    VFP
-- NAME  SHUT   MODE   RATE   RATE   RATE   RATE   RATE   PRES  PRES  TABLE  ALFQ
WCONPROD
OP01     SHUT   GRUP   1*     1*     1*     1*     1*     500.0 100.0   1       /
OP02     SHUT   GRUP   1*     1*     1*     1*     1*     500.0 100.0   1       /
/
--
-- WELL OPTIONS FOR PROBLEMATIC THP CONTROLLED WELLS
--
-- WELL   IMP   CLSE   RATE   VFP
-- NAME   EXP   WELL   CNTL   EXT
WVFPEXP
'OP*   '  1*    1*     YES1   1*                                              /
/
```

Here both wells are declared as initially shut on the WELSPECS keyword and use VFPPROD table number one as declared on the WCONPROD keyword. The WFVPEXP keyword declares all wells with a name beginning with OP to use implicit lookup and the wells are prevented from changing from rate control to THP control provided they can produce at a higher rate under THP control.

12.3.360 WWPAVE – WELL BLOCK AVERAGE PRESSURE CALCULATION PARAMETERS FOR INDIVIDUAL WELLS

Description

The WWPAVE keyword defines the method and parameters for calculating a well's block average pressures for individual wells that can be written to the SUMMARY and RSM files via the WBP, WBP4, WBP5 and WBP9 vectors in the SUMMARY section. The resulting average pressure can be written out to the summary file in order to compared with field observed data. The keyword is similar to the WPAVE keyword in the SCHEDULE section that has similar functionality, but is applied to all wells in the model.

Note that WWPAVE will overwrite any parameters on the WPAVE keyword for a given well, and that WWPAVE can also be overwritten by any subsequent WPAVE keyword.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WELNAME	A character string of up to eight characters in length that defines the well name for which the well average pressure calculation parameters are being defined. Note that the well name (WELNAME) must have been declared previously using the WELSPECS keyword in the SCHEDULE section, otherwise an error may occur.			None
2	WPAVE1	A real dimensionless value that defines the weighting factor between the inner block and the surrounding blocks used in the calculation of the connection factor weighted average pressure. If WPAVE1 is greater than or equal to zero and less than or equal to one, then the average pressure for each well connection is calculated based on this weighting factor. A value of zero indicates only the surrounding blocks should be used in the calculation; and a value of one indicates only the inner blocks should be used. If WPAVE1 is less than zero, then the average pressure for each well connection is weighted based on the pore volumes of the inner and surrounding blocks.			0.5
3	WPAVE2	A real dimensionless value greater than or equal to zero and less than or equal to one, that defines the weighting factor between the connection weighted average pressures and the pore volume weighted average pressures. If WPAVE2 is equal to one, then the average pressures are calculate based only using the connection factor calculated pressures. If WPAVE2 is equal to zero, then average pressures are calculate based on only using the pore volumes calculated pressures.			1.0

No.	Name	Description			Default
		Field	Metric	Laboratory	
4	WPAVE3	A defined character string that determines how the hydrostatic head calculation is performed in correcting the pressures to the BHP reference depth on the WELSPECS or WPAVEDEP keywords in the SCHEDULE section. WPAVE3 should be set to one of the following character strings: 1) WELL: the hydrostatic head is calculated using the density of the fluid in the wellbore at the well connections. 2) RES: he hydrostatic head is calculated using the density of the fluid in the reservoir with well connections and averaged over the connections. 3) NONE: no hydrostatic correction is applied to the pressures.			WELL
5	WPAVE4	A defined character string that determines which connections should be used in the calculations, WPAVE4 should be set to one of the following character strings: 1) OPEN: only open connections and associated grid blocks should be used in the calculations. This option may result in pressure discontinuities if connections are opened and closed during the run. 2) ALL: all currently defined open and closed connections and associated grid blocks are used in the calculations. The pressure discontinuities issue mentioned above can be avoided with this option and defining all the well connections for a well at the beginning of the run. Only the OPEN option is currently supported by the simulator.			OPEN
Notes: 1) The keyword should be terminated by a "/".					

Table 12.137: WWPAVE Keyword Description

The keyword is not applicable and should not be used with radial and spider grid geometries.

See also the WELSPECS keyword that defines a well and a well's bottom-hole pressure reference depth, the WPAVEDEP keyword that also defines a well's bottom-hole pressure reference depth, and the COMPDAT keyword to define a well's connections. All the aforementioned keywords are described in the SCHEDULE section.

Examples

The following example defines the default well block average pressure calculation parameters for three oil wells: OP01, OP02 and OP03.

```
--
--       DEFINE WELL BLOCK AVERAGE PRESSURE CALCULATION PARAMETERS
--
-- WELL   INNER   PORV   WELL   OPEN
-- NAME   OUTER   CONN   RES    ALL
WWPAVE
OP01      0.5     1.0    WELL   ALL                                    /
OP02      0.5     1.0    WELL   ALL                                    /
OP03      0.5     1.0    1*     1*                                     /
```

And the next example shows the parameters used in the Norne model.

```
--
--        DEFINE WELL BLOCK AVERAGE PRESSURE CALCULATION PARAMETERS
--
-- WELL   INNER   PORV   WELL  OPEN
-- NAME   OUTER   CONN   RES   ALL
WPAVE
OP01      1*      0.0    WELL  ALL                                          /
OP02      1*      0.0    WELL  ALL                                          /
OP03      1*      0.0    WELL  ALL                                          /
```

Here only pore volume weighting is used instead of connection weighting.

12.3.361 ZIPP2OFF – DEACTIVATE AUTOMATIC TIME STEP CONTROL

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The ZIPP2OFF keyword deactivates the commercial simulator's alternative automatic time step selection algorithm that assumes no prior knowledge of the problem, as opposed to the standard time step algorithm that is controlled via the TUNING keyword in the SCHEDULE section, combined with posterior knowledge gained from previous time steps.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

See section *2.2 Running OPM Flow 2023-10 From The Command Line* on how to control time stepping for OPM Flow.

12.3.362 ZIPPY2 – ACTIVATE AUTOMATIC TIME STEP CONTROL

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The ZIPPY2 keyword activates the commercial simulator's alternative automatic time step selection algorithm that assumes no prior knowledge of the problem, as opposed to the standard time step algorithm that is controlled via the TUNNING keyword in the SCHEDULE section, combined with posterior knowledge gained from previous time steps.

This keyword is not supported by OPM Flow but has no effect on the results so it will be ignored.

See section *2.2 Running OPM Flow 2023-10 From The Command Line* on how to control time stepping for OPM Flow.

APPENDIX A: KEYWORD INDEX - ALPHABETIC LISTING

A

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter A	Status
ACTDIMS – ACTION Keyword Dimensions	
ACTION – Define Action Conditions and Command Processing (Field)	
ACTIONG – Define Action Conditions and Command Processing (Groups)	
ACTIONR – Define Action Conditions and Command Processing (Regions)	
ACTIONS – Define Action Conditions and Command Processing (Well Segments)	
ACTIONW – Define Action Conditions and Command Processing (Wells)	
ACTIONX – Define Action Conditions and Command Processing	
ACTNUM – Set the Status of a Grid Block To Active or Inactive	
ACTPARAM – Define Action Facility Target and Tolerance Parameters	
ADD – Add a Constant to a Specified Array	
ADDREG – Add a Constant to an Array based on a Region Number	
ADDZCORN – Add a Constant to the ZCORN Depth Array	
ADSALNOD – Salt Concentration Based on SATNUM Array	
ADSORP – Define Generalized Langmuir Adsorption Function	
AITS – Activate Intelligent Time Stepping	
AITSOFF – Deactivate Intelligent Time Stepping	
ALKADS – Define Alkaline Adsorption Functions	
ALKALINE – Activate the Alkaline Phase and Model	
ALKROCK – Define Rock Alkaline Properties	
ALL – Export Standard Summary Variable Vectors to File	
ALPOLADS – Polymer Adsorption versus Alkaline Concentration Multipliers	
ALSURFAD – Surfactant Adsorption versus Alkaline Concentration Multipliers	
ALSURFST – Water-Oil Surface Tension versus Alkaline Concentration Multipliers	
AMALGAM – Define LGR Amalgamations	
API – Activate API Tracking	

A

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter A	Status
APIGROUP – Define API Tracking Number of Grouped Oil PVT Tables	
APILIM – Define API Tracking Grid Block Limits	
APIVD - Equilibration Oil API Gravity versus Depth Tables	
AQANNC – Define Analytic Aquifer Non-Neighbor Connections	
AQANNC – Define Analytic Aquifer Non-Neighbor Connections	
AQANTRC - Define Analytic Aquifer Initial Tracer Concentrations	
AQUALIST – Define An Analytic Aquifer Name to Aquifer Numbers	
AQUANCON – Define Analytical Connections to the Grid	
AQUCHGAS – Define Constant Pressure Gas Analytical Aquifer Properties	
AQUCHWAT – Define Constant Pressure Water Analytical Aquifer Properties	
AQUCON – Define Numerical Aquifer Connections to the Grid	
AQUCT – Define Carter-Tracy Analytical Aquifers	
AQUCWFAC – Modify Constant Pressure Water Analytical Aquifer Properties	
AQUDIMS – Define Aquifer Dimensions	
AQUFET – Define Fetkovich Analytical Aquifer and Connections	
AQUFETP – Define Fetkovich Analytical Aquifers	
AQUFLUX - Define Constant Flux Analytical Aquifer	
AQUNNC – Define Numerical Aquifer Non-Neighbor Connections	
AQUNUM – Define Numerical Aquifer Properties	
AQUTAB – Define Carter-Tracy Aquifer Influence Functions	
AUTOCOAR - Define Auto Refinement Grid Coarsen Area	
AUTOREF - Define Auto Refinement Options	

B

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter B	Status
BC – Define Boundary Conditions	OPM Flow
BCCON – Define Boundary Conditions Connections	
BCPROP – Define Boundary Conditions Properties	
BDENSITY – Define the Surface Brine Density for the Fluid	
BGGI - Define Gi Gas Formation Volume Factor Pressure Tables	
BIGMODEL – Activate Big Model Option (Retired)	
BLACKOIL – Activate Black-Oil Phases	
BOGI - Define Gi Oil Formation Volume Factor Pressure Tables	
BOUNDARY – Define a Boundary Box for Printing	
BOX - Define a Range of Grid Blocks to Enter Property Data	
BPARA – Activate Block Parallel License	
BPIDIMS – Define the Dimensions of the Interpolated Block Quantities	
BRANPROP – Define Network Branch Properties for Extended Network Option	
BRINE – Activate Brine Tracking Option	
BTOBALFA – Dual Porosity Matrix to Fracture Multiplier (All Cells)	
BTOBALFV – Dual Porosity Matrix to Fracture Multiplier (Individual Cells)	

C

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter C	Status
CALTRAC – Define a Gas Calorfic Value Tracer	
CARFIN – Define a Cartesian Local Grid Refinement	
CART – Activate Cartesian Geometry	
CBMOPTS – Define Coal Bed Methane Options	
CECON – Define Well Connections Economic Limit Criteria	
CECONT – Define Well Connections Tracer Economic Limit Criteria	
CNAMES – Define Compositional Component Names	OPM Flow
CO2STORE – Activate the CO2 Storage Model	OPM Flow
COAL – Activate the Coal Phase (CBM Model)	
COALADS – Define Gas and Solvent Relative Adsorption Tables	
COALNUM – Define the Coal Region Numbers	
COALPP – Define Gas and Solvent Partial Pressure Adsorption Tables	
COARSEN – Define Grid Coarsening Cells	
COLLAPSE – Define Compressed Vertical Equilibrium Cells	
COLUMNS – Define Input File Column Margins	
COMPDAT – Define Well Connections to the Grid	
COMPDATL – Define Well Connections to a LGR Grid	
COMPDATM – Define Well Connections to an Amalgamated LGR Grid	
COMPFLSH – Assign COMPDAT Differential-Flash Liberation Ratios	
COMPIMB – Assign Imbibition Saturation Tables to Well Connections	
COMPINJK – Assign Injection well Relative Permeability Values	
COMPLMPL – Assign Well LGR Connections to Completions	
COMPLUMP – Assign Well Connections to Completions	
COMPOFF – Deactivate Network Automatic Compressors	
COMPORD - Define Well Connection Ordering	

C

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter C	Status
COMPRIV – Define Grid Cell Connections to a River	
COMPRP – Re-Scale Fluid Saturations of Well Connections	
COMPRPL – Re-Scale Fluid Saturations of Well LGR Connections	
COMPS – Activate Compositional Modeling Formulation	OPM Flow
COMPSEGL – Define Well Connections for Multi-Segment Wells in a LGR	
COMPSEGS – Define Well Connections for Multi-Segment Wells	
COMPTRAJ – Define Well Trajectory Connections to the Grid	OPM Flow
COMPVE – Re-Define Well Connection Depths	
COMPVEL – Re-Define Well LGR Connection Depths	
COORD – Define a Set of Coordinates Lines for a Reservoir Grid	
COORDSYS – Define Coordinate Grid Options	
COPY – Copy Array Data to Another Array	
COPYBOX – Copy Array Data Defined by a Box	
COPYREG – Copy an Array to Another Array based on a Region Number	
CPIFACT – Define Well Connection Transmissibility Multipliers	
CPIFACTL – Define Well Connection Transmissibility Multipliers in a LGR	
CPR – Activate Constrained Pressure Residual ("CPR") Linear Solver	
CRITPERM – Define Minimum Permeability for Vertical Equilibrium Grid Cell Compression	
CSKIN – Re-Define Well Connection Skin Factors	

D

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter D	Status
DATE - Activate the DATE Option for the SUMMARY File	
DATES – Advance Simulation by Reporting Date	
DATUM – Define the Datum Depth for the Model	
DATUMR – Define Datum Depths for the FIPNUM Regions	
DATUMRX – Define Datum Depths for the FIP Allocated Regions	
DCQDEFN – Define Gas DCQ Units as Rate or Energy	
DEBUG – Define the Debug Data to be Printed to File	
DELAYACT - Define Delayed Action Keywords	
DENSITY – Define the Surface Oil, Water Gas Densities for the Fluids	
DEPTH - Edits the Depth at the Center of Each Cell	
DEPTHTAB – River Time and Depth Tables	
DIAGDISP – Activate Alternate Form of Tracer Dispersion	
DIFFC – Define PVT Region Molecular Diffusion Tables	
DIFFCGAS – Define PVT Region Gas Component Diffusion Coefficients	OPM Flow
DIFFCOAL – Define Coal Bed Methane Gas Diffusion Data	
DIFFCWAT – Define PVT Region Water Component Diffusion Coefficients	OPM Flow
DIFFDP – Activate Dual Porosity Molecular Diffusion for Matrix-Fracture Flow Only	
DIFFMMF – Define Diffusivity Multipliers for Matrix-Fractures	
DIFFMR – Define Grid Block Radial Direction Diffusivity Multipliers	
DIFFMR- – Define Grid Block Negative Radial Direction Diffusivity Multipliers	
DIFFMTHT – Define Grid Block Theta Direction Diffusivity Multipliers	
DIFFMTH- – Define Grid Block Negative Theta Direction Diffusivity Multipliers	
DIFFMX – Define Grid Block X-Direction Diffusivity Multipliers	
DIFFMX- – Define Grid Block Negative X-Direction Diffusivity Multipliers	
DIFFMY – Define Grid Block Y-Direction Diffusivity Multipliers	

D

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

DIFFMY- – Define Grid Block Negative Y-Direction Diffusivity Multipliers	
DIFFMZ – Define Grid Block Z-Direction Diffusivity Multipliers	
DIFFMZ- – Define Grid Block Negative Z-Direction Diffusivity Multipliers	
DIFFR – Define Grid Block Radial Direction Diffusivity Values	
DIFFTHT – Define Grid Block Theta Direction Diffusivity Values	
DIFFUSE – Activate Molecular Diffusion Option	
DIFFX – Define Grid Block X-Direction Diffusivity Values	
DIFFY – Define Grid Block Y-Direction Diffusivity Values	
DIFFZ – Define Grid Block Z-Direction Diffusivity Values	
DIMENS – Define the Dimensions of the Model	
DIMPES – Define IMPES Dynamic Solution Parameters	
DIMPLICT – Activate Fully Implicit Dynamic Solution Formulation	
DISGAS – Activate the Dissolved Gas Phase in the Model	
DISGASW – Activate Dissolved Gas in the Water Phase in the Model	OPM Flow
DISPDIMS – Define the Maximum Number of Dispersion Tables	
Error: Reference source not found	
DOMAINS – Define the Parallel Domain Properties	
DPGRID – Activate The Matrix Cell to Fracture Cell Option	
DPKRMOD – Modify Matrix Oil Relative Permeability Data	
DPNUM – Define Dual and Single Porosity Grid Block Array	
DR - Define the Size of Grid Blocks in the R Direction for All Cells	
DRILPRI – Define Prioritized Drilling Queue Priority Parameters	
DRSDT – Solution Gas (Rs) Maximum Rate of Increase Parameters	
DRSDTCON – CO2 Convective Dissolution Parameter (χ)	OPM Flow
DRSDTR – Solution Gas (Rs) Maximum Rate of Increase Parameters by Region	
DRV - Define the Size of Grid Blocks in the R Direction via a Vector	

D

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

DRVDT – Solution Oil (Rv) Maximum Rate of Increase Parameters	
DRVDTR – Solution Oil (Rv) Maximum Rate of Increase Parameters by Region	
DSPDEINT – Activate Brine Tracer Dispersion Interpolation by Water Density	
DTHETA - Define the Size of Grid Blocks in the THETA Direction for All Cells	
DTHETAV - Sets the Size of Grid Blocks in THETA Direction via a Vector	
DUALPERM – Activate Dual Permeability Model	
DUALPORO – Activate Dual Porosity Model	
DUMPCUPL – Activate Output to the Reservoir Coupling File	
DUMPFLUX – Activate Writing Out of a Flux File	
DX - Define the Size of Grid Blocks in the X Direction for All Cells	
DXV - Define the Size of Grid Blocks in the X Direction via a Vector	
DY - Define the Size of Grid Blocks in the Y Direction for All Cells	
DYNAMICR – Start of Dynamic Region Parameter Definition	
DYNRDIMS – Define Dynamic Region Dimensions	
DYV - Define the Size of Grid Blocks in the Y Direction via a Vector	
DZ - Define the Size of Grid Blocks in the Z Direction for All Cells	
DZMATRIX - Matrix Block Height for Gravity Drainage Model For All Cells	
DZMTRX - Matrix Block Height for Gravity Drainage Model for the Grid	
DZMTRXV - Matrix Block Height for Gravity Drainage Model For All Cells	
DZNET – Define Grid Block Net Thickness for All Cells	
DZV - Define the Size of Grid Blocks in the Z Direction via a Vector	

E

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter E	Status
ECHO – Activate Echoing of User Input Files to the Print File	
ECLMC – Activate Multi-Component Brine Model	
EDIT - Define the Start of the EDIT Section of Keywords	
EDITNNC – Scale Non-Neighbor Connections Between Cells Manually	
EDITNNCR – Reset Non-Neighbor Connections Between Cells Manually	
EHYSTR – Define Hysteresis Model and Parameters	
EHYSTRR – Define Hysteresis Model and Parameters via SATNUM	
END – Define the End of the Input File	
ENDACTIO – End the Definition of ACTION Commands	
ENDBOX – Define the End of the BOX Defined Grid	
ENDDYN– End of Dynamic Region Parameter Definition	
ENDFIN – End the Definition of a Local Grid Refinement	
ENDINC – Define the End of an Include File	
ENDNUM – Define the End-Point Scaling Depth Region Numbers	
ENDSCALE – Activate Relative Permeability End-Point Scaling Option	
ENDSKIP – DeActivate Skipping of Keywords and Input Data	
ENKRVD – Define Relative Permeability End-Points versus Depth Functions	
ENKRVDX	
ENKRVDX-	
ENKRVDY	
ENKRVDY-	
ENKRVDZ	
ENKRVDZ-	
ENPCVD – Define Maximum Capillary Pressure versus Depth Functions	
ENPTVD – Define Relative Permeability Saturation End-Points versus Depth	

E

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter E	Status
ENPTVDX	
ENPTVDX-	
ENPTVDY	
ENPTVDY-	
ENPTVDZ	
ENPTVDZ-	
ENSPCVD – Define Capillary Pressure End-Points versus Depth	
EPSDBGS - Write End-Point Debug Data to the DEBUG File (Multiple)	
EPSDEBUG - Write End-Point Debug Data to the DEBUG File (Individual)	
EQLDIMS – Define the Equilibration Data Dimensions	
EQLNUM – Define the Equilibration Region Numbers	
EQLOPTS – Activate the Equilibration Options	
EQLZCORN - Modify the Depth of the Corner-Point Depth Array	
EQUALREG – Sets an Array to a Constant by Region Number	
EQUALS – Sets a Specified Array to a Constant	
EQUIL – Define the Equilibration Initialization Data	
ESSNODE – Define Salt Concentration Data for Water-Oil Surface Tension	
EXCAVATE - Set the Status of a Grid Block To Active or Excavate	
EXCEL - Activate the EXCEL Option for the SUMMARY File	
EXIT – Exit Simulation from within an Action Section	OPM Flow
EXTFIN - Define an External Unstructured Local Grid Refinement	
EXTHOST - Define Host Cells for External LGRs	
EXTRAPMS – Activate Extrapolation Warning Messages	
EXTREPGL - Define Host Cells for External Unstructured LGRs	

F

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter F	Status
FAULTDIM – Define the Number of Fault Segments	
FAULTS – Define Faults in the Grid Geometry	
FBHPDEF – Define Well Default BHP Target and Constraints	
FHERCHBL – Define Herschel-Bulkley Data versus Polymer Concentration	
FIELD – Activate the Oil Field System of Units for the Model	
FILEUNIT – Activate Unit Consistency Checking	
FILLEPS – Activate Saturation End-Point Export to the INIT File	
FIP – Define the Fluid In-Place Names and Region and Numbers	
FIPNUM – Define the Fluid In-Place Region Numbers	
FIPOWG – Activate Oil, Gas, and Water FIP Zone Reporting	
FLUXNUM – Define the Flux Regions	
FLUXREG – Define Active Flux Regions	
FLUXTYPE – Defines the Flux Boundary Type	
FMTHMD – Activate The Format History Match Gradient File Option	
FMTIN – Activate The Format Input File Option	
FMTOUT – Activate The Format Output File Option	
FMWSET - Export Well Status Vectors for the Field to File	
FOAM – Activate the Foam Phase and Model	
FOAMADS - Define Foam Rock Adsorption Tables	
FOAMDCYO – Define Foam Decay versus Oil Saturation Tables	
FOAMDCYW – Define Foam Decay versus Water Saturation Tables	
FOAMFCN – Define Foam Gas Mobility Reduction versus Capillary Number	
FOAMFRM – Define Foam Gas Mobility Reduction versus Reference Mobility	
FOAMFSC – Define Foam Gas Mobility versus Surfactant Concentration Functions	
FOAMFSO – Define Foam Gas Mobility Reduction versus Oil Saturation	

F

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter F	Status
FOAMFST – Define Foam Gas-Water Surface Tension versus Surfactant Concentration	
FOAMFSW – Define Foam Gas Mobility Reduction versus Water Saturation	
FOAMMOB - Define Foam Gas Mobility versus Foam Concentration Tables	
FOAMMOBP – Define Foam Mobility Reduction versus Oil Pressure	
FOAMMOBS – Define Foam Mobility Reduction versus Shear	
FOAMOPTS - Define Foam Model Options	
FOAMROCK - Define Foam Rock Properties	
FORMFEED – Defined the Print File Form-Feed Character	
FMWSET - Export Well Status Vectors for the Field to File	
FRICTION – Activate Wellbore Friction Option	
FULLIMP – Activate Fully Implicit Solution Option	

G

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter G	Status
GAS – Activate the Gas Phase in the Model	
GASBEGIN – Define Start of Annual Scheduling Section	
GASCONC – Define the Initial Equilibration Coal Gas Concentration for All Grid Blocks	
GASDENT – Define Gas Density Temperature Coefficients	OMP Flow
GASEND – Define End of Annual Scheduling Section	
GASFCOMP – Define Automatic Gas Compressors	
GASFDECR – Define Field Gas Sales Contract Monthly Reduction	
GASFDELC – Define Gas Deliverability Calculation	
GASFIELD – Define Gas Field Operations Options	
GASFTARG – Define Field Gas Sales Contract Monthly Target	
GASJT – Define Gas Joule-Thomson Coefficient	OPM Flow
GASMONTH – Define Start of Annual Scheduling Event	
GASPERIO – Advance Simulation by Gas Contract Period	
GASSATC – Define the Initial Equilibration Saturated Coal Gas Concentration for All Grid Blocks	
GASVISCT - Define Gas Viscosity versus Temperature Functions	OPM Flow
GASWAT – Activate the Gas-Water Model Formulation	OPM Flow
GASYEAR – Advance Simulation by Gas Contract Year	
GCALECON – Group Economic Criteria for Production Calorific Groups	
GCONCAL – Group Production Calorific Targets	
GCONENG – Group Production Energy Targets	
GCONINJE – Group Injection Targets and Constraints	
GCONPRI – Group Production Priority Targets and Constraints	
GCONPROD – Group Production Targets and Constraints	
GCONSALE – Define Group Sales Gas Production Targets and Constraints	
GCONSUMP – Define Group Gas Consumption and Gas Import Targets	

G

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter G	Status
GCONTOL – Define Group Constraint Tolerance	
GCUTBACK – Define Group Cutback Limits and Parameters	
GCUTBACT – Define Group Tracer Cutback Limits and Parameters	
GCVD – Define Equilibration Coal Gas Concentration versus Depth Tables	
GDCQ – Define Group Multiple Daily Contract Quantities	
GDCQECON – Group Economic Criteria for DCQ Production Groups	
GDFILE – Load a Grid File	
GDIMS – Activate Instantaneous Gradient Option and Define Dimensions	
GDORIENT - Define Grid Orientation Parameters	
GDRILPOT – Define Group Potential Rates for Automatic Drilling	
GECON – Group Economic Criteria for Production Groups	
GECONT – Group Tracer Economic Criteria for Production Groups	
GEFAC – Define Group Efficiency	
GETDATA – Load and Assign Data Array from INIT or RESTART Files	
GETGLOB – Activate Loading of Global Grid Restart Data Option	
GI - Define the Initial Equilibration Gi Values for All Grid Blocks	
GIALL – Define Gi Values and PVT Properties versus Pressure	
GIMODEL – Activate Gi Pseudo Compositional Option	
GINODE – Define Gi Node Values	
GLIFTLIM – Group Artificial Lift Constraints	
GLIFTOPT – Define Group Gas Optimization Limits	
GMWSET - Export Well Status Vectors by Group to File	
GNETDP – Group Network Pressure and Rate Controls	
GNETINJE – Define Group Injection Network Configuration	
GNETPUMP – Standard Network Automatic Compressor and Pumps	

G

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter G	Status
GPMAINT – Define Group Pressure Maintenance Targets and Controls	
GRADGRUP – Define Group History Match Gradient File Output	
GRADRESV – Define Solution Derivative History Match Gradient Output	
GRADRFT – Define RFT Derivative History Match Gradient Output	
GRADWELL – Define Well History Match Gradient File Output	
GRAVCONS – Re-Define Gravity Constant	
GRAVDR – Activate Gravity Drainage and Imbibition for Dual Porosity Model	
GRAVDRB - Activate Vertical Discretized Gravity Drainage and Imbibition for Dual Porosity Model	
GRAVDRM - Activate Alternative Gravity Drainage and Imbibition for Dual Porosity Model	
GRAVITY– Define the Surface Oil, Water Gas Gravities for the Fluids	
GRDREACH – Define River and Grid Block Connections	
GRID - Define the Start of the GRID Section of Keywords	
GRIDFILE – Set the Grid File Output Options	
GRIDOPTS - Grid Processing Options	
GRIDUNIT – Define the Grid Units	
GRUPMAST – Define Master and Slave Groups	
GRUPNET – Define Group Standard Network Parameters	
GRUPRIG – Group Drilling and Workover Rig Specifications	
GRUPSLAV – Define Slave Groups in Slave Reservoirs	
GRUPTARG – Modify Group Targets and Constraints Values	
GRUPTREE – Define Group Tree Hierarchy	
GSATINJE – Define Group Satellite Injection Rates	
GSATPROD – Define Group Satellite Production Rates	
GSEPCOND – Assign Group Separators	
GSF - Gas Saturation Function Tables (Gas-Water Systems)	OPM Flow

G

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter G	Status
GSSCPTST – Perform Sustainable Capacity Test	
GSWINGF – Define Group Multiple Gas Contract Parameters	
GTADD – Add a Constant to a Group Target or Constraint	
GTMULT – Multiply Group Target or Constraint by a Constant	
GUIDECAL – Scale Guide Rates Based on Gas Calorific Value	
GUIDERAT – Define Group Guide Rate Formula	
GUPFREQ – Instantaneous Gradient Option Update Frequency	
GWRTWCV – Instantaneous Gradient Option Well Variables	

H

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter H	Status
H2STORE – Activate the H2 Storage Model	
HALFTRAN – Activate Half Block Transmissibility Calculations	
HA – History Match End-Point Gradient Additive Modifier	
HAKRG	
HAKRGR	
HAKRO	
HAKRORG	
HAKRORW	
HAKRW	
HAKRWR	
HAPCG	
HAPCG	
HASGLPC	
HASOGCR	
HASOWCR	
HASWL	
HASWLPC	
HBNUM – Define Herschel-Bulkley Region Numbers	
HDISP – Define Tracer Mechanical Dispersivity Parameters	
HEATCR – Define Reservoir Rock Heat Capacity for All Cells	
HEATCRT – Define Reservoir Rock Heat Capacity Temperature Dependence for All Cells	
HM – History Match End-Point Gradient Multiplicative Modifier	
HMKRG	
HMKRGR	
HMKRO	
HMKRORG	

H

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter H	Status
HMKRORW	
HMKRW	
HMKRWR	
HMPCG	
HMPCG	
HMSGLPC	
HMSOGCR	
HMSOWCR	
HMSWL	
HMSWLPC	
HM – History Match Region Gradient Parameters	
HMKRG	
HMKRGR	
HMKRO	
HMKRORG	
HMKRORW	
HMKRW	
HMKRWR	
HMPERMX	
HMPERMY	
HMPERMZ	
HMPORVM	
HMPRMXY	
HMSGCR	
HMSGL	
HMSGLPC	

H

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter H	Status
HMSIGMA	
HMSOGCR	
HMSOWCR	
HMSWCR	
HMSWL	
HMSWLPC	
HMTRANX	
HMTRANY	
HMTRANZ	
HMTRNXY	
HMAQUCT – History Match Carter-Tracy Aquifer Gradient Parameters	
HMAQUFET – History Match Fetkovich Aquifer Gradient Parameters	
HMAQUNUM - History Match Numerical Aquifer Gradient Parameters	
HMDIMS – Define History Match Gradient Parameter Dimensions	
HMFAULTS – History Match Fault Gradient Parameters	
HMMLAQUN – History Match Numerical Aquifer Gradient Multipliers	
HMMLCTAQ – History Match Carter-Tracy Aquifer Gradient Multipliers	
HMMLFTAQ – History Match Fetkovich Aquifer Gradient Multipliers	
HMMLT – History Match Grid Permeability Gradient Cumulative Multipliers	
HMMLTPR	
HMMLTPTH	
HMMLTPX	
HMMLTPXY	
HMMLTPY	
HMMLTPZ	
HMMLTWCN – History Match Well Connection and Skin Multipliers	

H

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter H	Status
HMMMREGT - History Match Region Transmissibility Gradient Cumulative Multipliers	
HMMROCK – History Match Rock Compressibility Gradient Cumulative Multipliers	
HMMROCKT – History Match Rock Compaction Gradient Cumulative Multipliers	
HMMULRGT – History Match Region Transmissibility Parameters	
HMMULT – History Match Grid Transmissibility & Pore Volume Gradient Cumulative Multipliers	
HMULTPV	
HMMULTR	
HMMULTTH	
HMMULTX	
HMMULTXY	
HMMULTY	
HMMULTZ	
HMMULTFT – History Match Fault Transmissibility Gradient Cumulative Multipliers	
HMMULTSG – History Match Dual porosity Sigma Gradient Cumulative Multipliers	
HMPROPS – History Match End-Point Section Start	
HMROCK – History Match Rock Compressibility Gradient Parameters	
HMROCKT – History Match Rock Compaction Gradient Parameters	
HMRREF – History Match Rock Table Reference Pressure Values	
HMWELCON – History Match Well Connection and Skin Parameters	
HMWPIMLT – History Match Well Productivity Index Parameters	
HRFIN - Define the Ratio of LGR Grid Blocks in the R-Direction	
HWKRO – End-Point Scaling of Grid Cell Kro(Swl) (High Salinity and Water Wet)	
HWKRORG – End-Point Scaling of Grid Cell Kro(Sgcr) (High Salinity and Water Wet)	
HWKRORW – End-Point Scaling of Grid Cell Kro(Swcr) (High Salinity and Water Wet)	
HWKRW – End-Point Scaling of Grid Cell Krw(Sw =1.0) (High Salinity and Water Wet)	

H

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter H	Status
HWKRWR – End-Point Scaling of Grid Cell KRWR(Sw=1.0) (High Salinity and Water Wet)	
HWPCW – End-Point Scaling of Grid Cell Water Capillary Pressure (High Salinity and Water Wet)	
HWSNUM – Define the Saturation Table Region Numbers (High Salinity and Water Wet)	
HWSOGCR – End-Point Scaling Grid Cell SOGCR (High Salinity and Water Wet)	
HWSOWCR – End-Point Scaling Grid Cell SOWCR (High Salinity and Water Wet)	
HWSWCR – End-Point Scaling Grid Cell SWCR (High Salinity and Water Wet)	
HWSWL – End-Point Scaling Grid Cell SWL (High Salinity and Water Wet)	
HWSWLPC – End-Point Scaling Grid Cell SWLPC (High Salinity and Water Wet)	
HWSWU – End-Point Scaling Grid Cell SWU (High Salinity and Water Wet)	
HXFIN - Define the Ratio of LGR Grid Blocks in the X-Direction	
HYDRHEAD – Define Hydraulic Head Output Reference Data	
HYFIN - Define the Ratio of LGR Grid Blocks in the Y-Direction	
HYMOBGDR – Activate Carlson and Killough Alternative Drainage Hysteresis Option	
HYST – Activate the Hysteresis Option (Retired)	
HYSTCHCK - Activate Hysteresis Imbibition and Drainage End-Point Validation	
HZFIN - Define the Ratio of LGR Grid Blocks in the Z-Direction	

I

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter I	Status
IHOST – Assign LGRs to Parallel Process Number	
IKRG – End-Point Scaling of Grid Cell Krg(Sgu) (Imbibition)	
IKRGX	
IKRGX-	
IKRGY	
IKRGY-	
IKRGZ	
IKRGZ-	
IKRGR – End-Point Scaling of Grid Cell Krgr(1-Sogcr) (Imbibition)	
IKRGRX	
IKRGRX-	
IKRGRY	
IKRGRY-	
IKRGRZ	
IKRGRZ-	
IKRO – End-Point Scaling of Grid Cell Kro(Swl) (Imbibition)	
IKROX	
IKROX-	
IKROY	
IKROY-	
IKROZ	
KROZ-	
IKRORG – End-Point Scaling of Grid Cell Kro(Sgcr) (Imbibition)	
IKRORGX	
IKRORGX-	
IKRORGY	

I

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter I	Status
IKRORGY-	
IKRORGZ	
IKRORGZ-	
IKRORW – End-Point Scaling of Grid Cell Kro(Swcr) (Imbibition)	
IKRORWX	
IKRORWX-	
IKRORWY	
IKRORWY-	
IKRORWZ	
IKRORWZ-	
IKRW – End-Point Scaling of Grid Cell Krw(Sw =1.0) (Imbibition)	
IKRWX	
IKRWX-	
IKRWY	
IKRWY-	
IKRWZ	
IKRWZ-	
IKRWR – End-Point Scaling of Grid Cell KRWR(Sowcr) (Imbibition)	
IKRWRX	
IKRWRX-	
IKRWRY	
IKRWRY-	
IKRWRZ	
IKRWRZ-	
IMBNUM – Define the Imbibition Saturation Table Region Numbers	
IMBNUMX	

I

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter I	Status
IMBNUMX-	
IMBNUMY	
IMBNUMY-	
IMBNUMZ	
IMBNUMZ-	
IMBNUMMF – Define the Imbibition Saturation Table Region Numbers (Matrix-Fracture)	
IMKRVD – Imbibition Relative Permeability End-Points versus Depth Functions	
IMKRVDX	
IMKRVDX-	
IMKRVDY	
IMKRVDY-	
IMKRVDX	
IMKRVDZ-	
IMPCVD – Imbibition Maximum Capillary Pressure versus Depth Functions	
IMPES – Activate Implicit Pressure Explicit Saturation Solution Option	
IMPLICIT – Activate Fully Implicit Solution Option	
IMPORT – Import Grid File Data at the Current Position	
IMPTVD – Imbibition Relative Permeability Saturation End-Points versus Depth	
IMPTVDX	
IMPTVDX-	
IMPTVDY	
IMPTVDY-	
IMPTVDZ	
IMPTVDZ-	
IMSPCVD – Imbibition Capillary Pressure Connate Saturations versus Depth	

I

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter I	Status
INCLUDE – Load Another Data File at the Current Position	
INIT – Activate the INIT File Option	
INRAD – Define the Inner Radius of a Radial Grid	
INSPEC – Activate the INSPEC File Option	
INTPC – Activate Dual Porosity Integrated Capillary Pressure Option	
IONROCK – Define the Ion Exchange Capacity for All the Cells	
IONXROCK - Define Ion Exchange Constant by Saturation Table Regions	
IONXSURF - Define Surfactant Ion Exchange Constant by Saturation Table Regions	
IPCG – End-Point Scaling of Grid Cell Gas Capillary Pressure (Imbibition)	
IPCW – End-Point Scaling of Grid Cell Water Capillary Pressure (Imbibition)	
ISGCR – End-Point Scaling of Grid Cell Critical Gas Saturation (Imbibition)	
ISGCRX	
ISGCRX-	
ISGCRY	
ISGCRY-	
ISGCRZ	
ISGCRZ-	
ISGL – End-Point Scaling of Grid Cell Connate Gas Saturation (Imbibition)	
ISGLX	
ISGLX-	
ISGLY	
ISGLY-	
ISGLZ	
ISGLZ-	
ISGLPC – End-Point Scaling of Grid Cell Capillary Pressure Connate Gas Saturation (Imbibition)	

I

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter I	Status
ISGU – End-Point Scaling of Grid Cell Maximum Gas Saturation (Imbibition)	
ISGUX	
ISGUX-	
ISGUY	
ISGUY-	
ISGUZ	
ISGUZ-	
ISOGCR – End-Point Scaling of Grid Cell Critical Oil Saturation to Gas (Imbibition)	
ISOGCRX	
ISOGCRX-	
ISOGCRY	
ISOGCRY-	
ISOGCRZ	
ISOGCRZ-	
ISOLNUM – Define the Independent Reservoir Regions	
ISOWCR – End-Point Scaling of Grid Cell Critical Oil Saturation to Water (Imbib.)	
ISOWCRX	
ISOWCRX-	
ISOWCRY	
ISOWCRY-	
ISOWCRZ	
ISOWCRZ-	
ISWCR – End-Point Scaling of Grid Cell Critical Water Saturation (Imbibition)	
ISWCRX	
ISWCRX-	
ISWCRY	

I

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter I	Status
ISWCRY-	
ISWCRZ	
ISWCRZ-	
ISWL – End-Point Scaling of Grid Cell Connate Water Saturation (Imbibition)	
ISWLX	
ISWLX-	
ISWLY	
ISWLY-	
ISWLZ	
ISWLZ-	
ISWLPC – End-Point Scaling Grid Cell Capillary Pressure Connate Water Saturations (Imbibition)	
ISWU – End-Point Scaling of Grid Cell Maximum Water Saturation (Imbibition)	
ISWUX	
ISWUX-	
ISWUY	
ISWUY-	
ISWUZ	
ISWUZ-	

J

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter J	Status
JFUNC - Activate the Leverett J-function Option	
JFUNCR - Activate the Leverett J-function Saturation Table Option	

K

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter K	Status
KRG – End-Point Scaling of Grid Cell Krg(Sgu) (Drainage)	
KRGX	
KRGX-	
KRGY	
KRGY-	
KRGZ	
KRGZ-	
KRGR – End-Point Scaling of Grid Cell Krgr(1-Sogcr) (Drainage)	
KRGRX	
KRGRX-	
KRGRY	
KRGRY-	
KRGRZ	
KRGRZ-	
KRNUM – Define the Directional Saturation Table Region Numbers	
KRNUMR	
KRNUMR-	
KRNUMT	
KRNUMT-	
KRNUMX	
KRNUMX-	
KRNUMY	
KRNUMY-	
KRNUMZ	
KRNUMZ-	
KRNUMMF – Define the Saturation Table Region Numbers (Matrix-Fracture)	

K

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter K	Status
KRO – End-Point Scaling of Grid Cell Kro(Swl) (Drainage)	
KROX	
KROX-	
KROY	
KROY-	
KROZ	
KROZ-	
KRORG – End-Point Scaling of Grid Cell Kro(Sgcr) (Drainage)	
KRORGX	
KRORGX-	
KRORGY	
KRORGY-	
KRORGZ	
KRORGZ-	
KRORW – End-Point Scaling of Grid Cell Kro(Swcr) (Drainage)	
KRORWX	
KRORWX-	
KRORWY	
KRORWY-	
KRORWZ	
KRORWZ-	
KRW – End-Point Scaling of Grid Cell Krw(Sw =1.0) (Drainage)	
KRWX	
KRWX-	
KRWY	
KRWY-	

K

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter K	Status
KRWZ	
KRWZ-	
KRWR – End-Point Scaling of Grid Cell KRWR(Sowcr) (Drainage)	
KRWRX	
KRWRX-	
KRWRY	
KRWRY-	
KRWRZ	
KRWRZ-	

L

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter L	Status
LAB - Activate the Laboratory System of Units for the Model	
LANGMPL – Define Langmuir Pressure Grid Cell Multiplier	
LANGMUIR – Langmuir Adsorption Isotherm Tables	
LANGSOLV – Langmuir Adsorption Isotherm Solvent Tables	
LCUNIT – Define Linear Combination Rate and Volume Units	
LGR – Define Local Grid Refinement Dimensions and Parameters	
LGRCOPY – Activate Local Grid Refinement Inheritance	
LGRFREE – Activate Local Grid Refinement Independent Time Steps	
LGRLOCK – Deactivate Local Grid Refinement Independent Time Steps	
LGROFF – Deactivate a Local Grid Refinement	
LGRON – Activate a Local Grid Refinement	
LICENSES – Define Required Licenses for Run	
LIFTOPT – Activate Gas Lift Optimization	
LINCOM – Define Linear Combination Coefficients	
LINKPERM - Assign Cell Permeabilities to Cell Faces	
LIVEOIL – Activate the Live Oil Phase (Oil with Free and Dissolved Gas)	
LKRO – End-Point Scaling of Grid Cell Kro(Swl) (Low Salinity and Oil Wet)	
LKRORG – End-Point Scaling of Grid Cell Kro(Sgcr) (Low Salinity and Oil Wet)	
LKRORW – End-Point Scaling of Grid Cell Kro(Swcr) (Low Salinity and Oil Wet)	
LKRW – End-Point Scaling of Grid Cell Krw(Sw =1.0) (Low Salinity and Oil Wet)	
LKRWR – End-Point Scaling of Grid Cell KRWR(Sowcr) (Low Salinity and Oil Wet)	
LOAD – Load a SAVE File for a Fast Restart	
LOWSALT – Activate the Low Salt Brine Phase in the Brine Model	
LPCW – End-Point Scaling of Grid Cell Water Capillary Pressure (Low Salinity and Oil Wet)	
LSALTFNC - Define Low Salt Weighting Factors versus Salt Concentration Functions	

L

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter L	Status
LSLTWNUM – Define the Low Salt Water Wet Saturation Table Region Numbers	
LSNUM – Define the Low Salt Oil Wet Saturation Table Region Numbers	
LSOGCR – End-Point Scaling Grid Cell SOGCR (Low Salinity and Oil Wet)	
LSOWCR – End-Point Scaling Grid Cell SOWCR (Low Salinity and Oil Wet)	
LSWCR – End-Point Scaling Grid Cell SWCR (Low Salinity and Oil Wet)	
LSWL – End-Point Scaling Grid Cell SWL (Low Salinity and Oil Wet)	
LSWLPC – End-Point Scaling Grid Cell SWLPC (Low Salinity and Oil Wet)	
LSWU – End-Point Scaling Grid Cell SWU (Low Salinity and Oil Wet)	
LTOSIGMA - Dual Porosity Viscous Displacement Sigma Parameters	
LWKRO – End-Point Scaling of Grid Cell Kro(Swl) (Low Salinity and Water Wet)	
LWKRORG – End-Point Scaling of Grid Cell Kro(Sgcr) (Low Salinity and Water Wet)	
LWKRORW – End-Point Scaling of Grid Cell Kro(Swcr) (Low Salinity and Water Wet)	
LWKRW – End-Point Scaling of Grid Cell Krw(Sw =1.0) (Low Salinity and Water Wet)	
LWKRWR – End-Point Scaling of Grid Cell KRWR(Sw=1.0) (Low Salinity and Water Wet)	
LWPCW – End-Point Scaling of Grid Cell Water Capillary Pressure (Low Salinity and Water Wet)	
LWSLTNUM – Define the Low Salt Oil Wet Saturation Table Region Numbers	
LWSNUM – Define the Low Salt Water Wet Saturation Table Region Numbers	
LWSOGCR – End-Point Scaling Grid Cell SOGCR (Low Salinity and Water Wet)	
LWSOWCR – End-Point Scaling Grid Cell SOWCR (Low Salinity and Water Wet)	
LWSWCR – End-Point Scaling Grid Cell SWCR (Low Salinity and Water Wet)	
LWSWL – End-Point Scaling Grid Cell SWL (Low Salinity and Water Wet)	
LWSWLPC – End-Point Scaling Grid Cell SWLPC (Low Salinity and Water Wet)	
LWSWU – End-Point Scaling Grid Cell SWU (Low Salinity and Water Wet)	
LX - Dual Porosity Viscous Displacement X Direction Matrix Size for All Cells	
LXFIN – Define Logarithmic LGR Grid Block Spacing in the X-Direction	

L

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter L	Status
LY - Dual Porosity Viscous Displacement Y Direction Matrix Size for All Cells	
LYFIN – Define Logarithmic LGR Grid Block Spacing in the Y-Direction	
LZ - Dual Porosity Viscous Displacement Z Direction Matrix Size for All Cells	
LZFIN – Define Logarithmic LGR Grid Block Spacing in the Z-Direction	

M

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter M	Status
MAPAXES - Define the Map Origin Input Data	
MAPUNITS – Define the Map Axes Units	
MASSFLOW – Define River Mass Flow versus Time Tables	
MATCORR – Activate the Material Balance Correction Option	
MAXVALUE – Sets a Maximum Value for an Array Element	
MEMORY – Define Allocated Memory (Retired)	
MESSAGE – Output User Message	
MESSAGES – Define Message Print Limits and Stop Limits	
MESSOPTS – Reset Severity Level for Forced Time Steps	
MESSSRVC - Activate or Deactivate Database Message File Output	
METRIC – Activate the Metric System of Units for the Model	
MICP – Activate the Microbially Induced Calcite Precipitation Model	OPM Flow
MICPPARA – Define Microbially Induced Calcite Precipitation Parameters	OPM Flow
MINNNCT – Set a Minimum Non-Neighbor Connection Transmissibility	
MINNPCOL - Define the Minimum Number of Newton Iterations Used to Update Well Targets	OPM Flow
MINPORV – Set a Minimum Grid Block Pore Volume Threshold for All Cells	
MINPV – Set a Minimum Grid Block Pore Volume Threshold for All Cells	
MINPVV – Set a Minimum Grid Block Pore Volume Threshold for Individual Cells	
MINVALUE – Set a Minimum Value for an Array Element	
MISC – Define Solvent Miscibility-Immiscibility Transform Functions	
MISCIBLE – Define Miscibility Todd-Longstaff Parameters	
MISNUM – Define the Miscibility Region Numbers	
MLANG – Define Langmuir Maximum Gas Concentration for All Grid Cells	
MLANGSLV – Define Langmuir Maximum Solvent Concentration for All Grid Cells	
MONITOR – Activate Output of the Monitoring Data and File	

M

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter M	Status
MPFANUM – Define Multi-Point Flux Discretization Regions	
MPFNNC – Define Multi-Point Flux Non-Neighbor Connections	
MSFN – Miscible Normalized Relative Permeability Tables	
MSGFILE – Active or Deactivate Message File Output	
MULSGGD – Multiply Matrix-Fracture Coupling for Oil-Gas Gravity Drainage for All Cells	
MULSGGDV – Multiply Matrix-Fracture Coupling for Oil-Gas Gravity Drainage for Individual Cells	
MULTFLT – Multiply the Transmissibility of a Defined Fault by a Constant	
MULTIN – Activate the Non-Unified Multiple Input File Option	
MULTIPLY – Multiply a Specified Array by a Constant	
MULTIREG – Multiply an Array by a Constant based on a Region Number	
MULTNUM – Define the Multiple Transmissibility Regions	
MULTOUT – Activate the Non-Unified Multiple Output File Option	
MULTOUTS – Activate Non-Unified Multiple Summary Output File Option	
MULTPV – Multiply Cell Pore Volumes by a Constant	
MULTR - Multiply Cell Transmissibility in the +R Direction	
MULTR- - Multiply Cell Transmissibility in the -R Direction	
MULTREAL – Activate Commercial Simulator's Multi-Realization License	
MULTREGD – Multiply Diffusivities Between Regions	
MULTREGH – Multiply Thermal Conductivities Between Regions	
MULTREGP – Multiply Pore Volumes Based On Region Number	
MULTREGT – Multiply Transmissibilities Between Regions	
MULTSIG – Multiply Matrix-Fracture Coupling for All Cells	
MULTSIGV – Multiply Matrix-Fracture Coupling for Individual Cells	
MULTTHT - Multiply Cell Transmissibility in the +Theta Direction	
MULTTHT- - Multiply Cell Transmissibility in the -Theta Direction	

M

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter M	Status
MULTX - Multiply Cell Transmissibility in the +X Direction	
MULTX- - Multiply Cell Transmissibility in the -X Direction	
MULTY - Multiply Cell Transmissibility in the +Y Direction	
MULTY- - Multiply Cell Transmissibility in the -Y Direction	
MULTZ - Multiply Cell Transmissibility in the +Z Direction	
MULTZ- - Multiply Cell Transmissibility in the -Z Direction	

N

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter N	Status
NARROW – Activate Run Summary Narrow Column Output Option	
NCOMPS – Confirm Number of Compositional Components	OPM Flow
NCONSUMP - Node Gas Consumption (Extended Network)	
NEFAC - Node Efficiency Factors (Extended Network)	
NETBALAN – Network Balancing Parameters	
NETCOMPA – Define Automatic Compressors (Extended Network)	
NETWORK – Activate the Extended Network Option and Define Parameters	
NEWTON – Activate Newton Iteration SUMMARY Output	
NEWTRAN – Activate Irregular Corner-Point Grid Transmissibilities	
NEXT – Maximum Next Time Step Size (Alias for NEXTSTEP)	
NEXTSTEP – Maximum Next Time Step Size	
NEXTSTPL – Maximum Next Time Step Size (LGR)	
NINENUM – Define the Nine-Point Discretization Region	
NINEPOIN – Activate the Nine-Point Discretization Option	
NMATOPTS – Define the Discretized Matrix Dual Porosity Parameters	
NMATRIX – Activate the Discretized Matrix Dual Porosity Option	
NMESSAGE – Export Cumulative Message Summary Variables to File	
NNC – Define Non-Neighbor Connections Between Cells Manually	
NNEWTF – Activate the Non-Newtonian Fluid Model	
NOCASC – Activate Linear Solver Tracer Algorithm	
NODEPROP – Define Network Node Properties for Extended Network	
NODPPM – Deactivate Fracture Porosity-Permeability Calculation	
NOECHO – Deactivate Echoing of User Input Files to the Print File	
NOGGF – Deactivate Output of Grid Geometry File	
NOHMD – Deactivate History Match Gradient Derivative Calculations	

N

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter N	Status
NOHMO – Deactivate History Match Gradient Derivative Calculations (Alias)	
NOHYST - Deactivate the Hysteresis Option	
NOINSPEC – Deactivate Output of the INIT Index File	
NOMONITO – Deactivate Output of the Monitoring Data and File	
NONNC – Deactivate Non-Neighbor Connections	
NORSSPEC – Deactivate Output of the RESTART Index File	
NOSIM – Activate the No Simulation Mode for Data File Checking	
NOWARN – Deactivate Warning Messages	
NOWARNEP – Deactivate End-Point Scaling Warning Messages	
NRSOUT – Defined Maximum Number of RESTART Elements	
NSTACK – Define the Stack Length for the Iterative Linear Solver	
NTG – Define the Net-to-Gross Ratio for All the Cells	
NUMRES – Define the Number of Reservoir Grids	
NUPCOL – Define the Maximum Number of Newton Iterations Used to Update Well Targets	
NWATREM – Node Water Removal (Extended Network)	
NXFIN – Define the Number of LGR Grid Blocks in the X-Direction	
NYFIN – Define the Number of LGR Grid Blocks in the Y-Direction	
NZFIN – Define the Number of LGR Grid Blocks in the Z-Direction	

O

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter O	Status
OFM – Activate OFM File Output of the SUMMARY Data	
OIL – Activate the Oil Phase in the Model	
OILAPI – Define the Initial Equilibration Oil API for All Grid Blocks	
OILDENT – Define Oil Density Temperature Coefficients	OPM Flow
OILJT – Define Oil Joule-Thomson Coefficient	OPM Flow
OILVISCT – Define Oil Viscosity versus Temperature Functions	
OLDTRAN – Activate Cartesian Regular Grid Transmissibilities	
OLDTRANR – Activate Radial Regular Grid Transmissibilities	
OPERATE – Define Mathematical Operations on Arrays	
OPERATER – Define Mathematical Operations on Arrays by Region	
OPERNUM – Define Regions for Mathematical Operations on Arrays	
OPTIONS – Activate Various Program Options	
OUTRAD - Define the Outer Radius of a Radial Grid	
OUTSOL – Define Data to be Written to the RESTART File (Retired)	
OVERBURD – Define Rock Overburden Pressure versus Depth Tables	

P

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter P	Status
PARALLEL – Define Parallel Run Configuration	
PARAOPTS – Define Parallel Run Options	
PARTTRAC – Activate and Define Partitioned Tracer Option	
PATHS – Define Filename Directory Path Aliases	
PBUB – Define the Initial Equilibration Bubble-Point Pressure for All Grid Blocks	
PBVD – Equilibration Bubble-Point versus Depth Tables	
PCG – End-Point Scaling of Grid Cell Maximum Gas Capillary Pressure (Drainage)	
PCG32D – Gas-Oil Capillary Pressure versus Oil and Water Saturation Tables	
PCW – End-Point Scaling of Grid Cell Water Capillary Pressure (Drainage)	
PCW32D – Water-Oil Capillary Pressure versus Oil and Gas Saturation Tables	
PDEW – Define the Initial Equilibration Dew-Point Pressure for All Grid Blocks	
PDVD – Define Equilibration Dew-Point versus Depth Tables	
PEBI – Activate and Defined PEBI Grid Options	
PECOEFS – Define Petro-Elastic Model Coefficients	
PEDIMS – Define Petro-Elastic Model Regions and Table Dimensions	
PEGTAB – Petro-Elastic Pressure Shear Modulus Table	
PEGTAB0	
PEGTAB1	
PEGTAB2	
PEGTAB3	
PEGTAB4	
PEGTAB5	
PEGTAB6	
PEGTAB7	
PEKTAB – Petro-Elastic Pressure Bulk Modulus Table	

P

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter P	Status
PEKTAB0	
PEKTAB1	
PEKTAB2	
PEKTAB3	
PEKTAB4	
PEKTAB5	
PEKTAB6	
PEKTAB7	
PENUM – Define the Petro-Elastic Region Numbers	
PERFORMA – Export Standard Simulator Performance Summary Variables to File	
PERMAVE – Define Average Transmissibility Coefficients	
PERMFACT – Permeability Multiplication Factor as a Function of Porosity Change	OPM Flow
PERMJFUN – Define Leverett J-Function Permeability for All Cells	
PERMR – Define the Permeability for Each Cell in the R Direction	
PERMTHT – Define the Permeability for Each Cell in the THETA Direction	
PERMX - Define the Permeability in the X Direction for All the Cells	
PERMY - Define the Permeability in the Y Direction for All the Cells	
PERMZ - Define the Permeability in the Z Direction for All the Cells	
PETGRID – Load a Generic Simulation Grid File	
PETOPTS – Define Petrel and Generic Simulation File Options	
PICOND – Define the Generalized Pseudo Pressure Parameters	
PIMTDIMS – Define Well Productivity Scaling Table Dimensions	
PIMULTAB – Define Well Productivity Index versus Water Cut Tables	
PINCH – Define Pinch-Out Layer Options	
PINCHNUM – Define Pinch-Out Regions for the PINCHREG Keyword	

P

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter P	Status
PINCHOUT - Define Pinch-Out Layers Option (Fixed)	
PINCHREG - Define Pinch-Out Region Options	
PINCHXY – Define Pinch-Out Areal Options	
PINTDIMS – Define Polymer Molecular Weight Model Table Dimensions	OPM Flow
PLMIXNUM – Define the Polymer Region Numbers	
PLMIXPAR – Define the Polymer Todd-Longstaff Mixing Parameters	
PLYADS - Define Polymer Rock Adsorption Tables	
PLYADSS - Define Polymer Rock Adsorption with Salt Dependence Tables	
PLYATEMP – Define Polymer Adsorption Table Temperature	
PLYCAMAX - Define Polymer-Rock Maximum Adsorption by Cell	
PLYDHFLF - Define Polymer Thermal Degradation Half-Life Tables	
PLYESAL – Define Polymer Effective Salinity Coefficient	
PLYKRRF – Define Polymer Rock Permeability Reduction by Cell	
PLYMAX - Define Polymer-Salt Viscosity Mixing Concentrations	
PLYMWINJ – Polymer Molecular Weight Model Throughput and Velocity Table	OPM Flow
PLYRMDEN - Define Polymer Model In Situ Rock Density	
PLYROCK - Define Polymer-Rock Properties	
PLYROCKM - Modify Polymer-Rock Properties	
PLYSHEAR – Activate and Define Polymer Shearing Parameters	
PLYSHLOG - Activate and Define the Polymer Shearing Logarithmic Parameters	
PLYTRRF – Define Polymer Rock Permeability Reduction versus Temperature	
PLYTRRFA – Define Polymer Rock Permeability Reduction versus Temperature Option	
PLYVISC – Define Polymer Viscosity Scaling Factors	
PLYVISCS – Define Polymer-Salt Viscosity Scaling Factors	
PLYVISCT – Define Polymer-Temperature Viscosity Scaling Factors	

P

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter P	Status
PLYVMH - Polymer Molecular Weight Model Polymer Viscosity Constants	OPM Flow
PLYVSCST – Define Polymer-Salt-Temperature Viscosity Scaling Factors	
PMAX – Maximum and Minimum Pressure for Total Compressibility Check	
PMISC – Define Miscibility versus Pressure Tables	
POLYMER – Activate the Polymer Phase in the Model	
POLYMW – Activate the Polymer Molecular Weight Transport Option	OPM Flow
PORO - Define the Porosity Values for All the Cells	
PORV - Define the Pore Volumes for All the Cells	
PPCWMAX – Define SWATINIT Calculated Capillary Pressure Constraints	
PRECSALT – Activate the OPM Flow Salt Precipitation Model	OPM Flow
PRESSURE – Define the Initial Equilibration Pressures for All Grid Blocks	
PRIORITY – Activate and Define Well Prioritization Coefficients	
PROPS - Define the Start of the PROPS Section of Keywords	
PRORDER – Define a Group Production Rules Sequence	
PRVD – Define the Initial Equilibration Pressures versus Depth	
PSTEADY – Activate Pseudo Steady State Flow Calculation Option	
PVCDO - Oil PVT Properties for Dead Oil (Constant Compressibility)	
PVCO - Oil PVT Properties for Live Oil	
PVDG - Gas PVT Properties for Dry Gas	
PVDO – Oil PVT Properties for Dead Oil	
PVDS - Solvent PVT Properties for the Solvent Model	
PVTG - Gas PVT Properties for Wet Gas with Vaporized Oil	
PVTGW - Gas PVT Properties for Dry Gas with Vaporized Water	OPM Flow
PVTGWO - Gas PVT Properties for Wet Gas with Vaporized Water and Oil	OPM Flow
PVTNUM – Define the PVT Regions	

P

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter P	Status
PVTO - Oil PVT Properties for Live Oil	
PVTSOL - Oil PVT Properties for Live Oil versus CO2 Mass Fraction	OPM Flow
PVTW - Define Water Fluid Properties for Various Regions	
PVTWSALT - Define Brine Water Fluid Properties for Various Regions	
PVZG - Gas PVT Properties for Dry Gas (Z-Factor)	
PYACTION – Define Python Based Action Conditions and Command Processing	OPM Flow
PYEND – End the Definition of a PYINPUT Section	OPM Flow
PYINPUT – Define the Start of a PYINPUT Section	OPM Flow

Q

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter Q	Status
QDRILL – Define Sequential Drilling Queue Wells	
QHRATING – Define River Mass Flow versus Depth Tables	
QMOBIL Activate or Deactivate LGR End-Point Mobility Correction	

R

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter R	Status
RADFIN – Define a Radial Local Grid Refinement with One Column	
RADFIN4 – Define a Radial Local Grid Refinement with Four Columns	
RADIAL – Radial Grid Activation Option	
RAINFALL – Constant Flux Aquifer Rainfall Flux by Month	
RBEDCONT – Define River Grid Block Contact Area versus Depth	
RCMASTS – Reservoir Coupling Group Minimum Time Step for Flow Restriction	
REACHES – Define River Reaches Structure	
READDATA – Read Schedule Data Based on Current Time Step	
REFINE – Start the Definition of a Local Grid Refinement	
REGDIMS – Define the Maximum Number of Regions for a Region Array	
REGIONS - Define the Start of the REGIONS Section of Keywords	
RESIDNUM – Define Vertical Equilibrium Residual Flow Region Numbers	
RESTART – Restart Run From an Existing Restart File	
RESVNUM – Define Reservoir Coordinate Data Set	
RIVDEBUG – Define the Debug Data to be Printed to File (Rivers)	
RIVERSYS - Define River System (Branch Structure and Boundary Conditions)	
RIVRDIMS – Define the River Dimensions and Associated Data	
RIVRPROP – Modify River Reaches Properties	
RIVRXSEC – Define River Cross-Section versus Depth Parameters	
RIVSALT – Define River Upstream Flow Salt Concentrations	
RIVTRACE – Define River Upstream Flow Tracer Concentrations	
RKTRMDIR - Activate ROCKTAB Keyword Directional Transmissibility Multipliers	
ROCK - Define the Rock Compressibility for Various Regions	
ROCK2D – Pore Volume Compaction versus Pressure and Sw Tables	
ROCK2DTR – Transmissibility Compaction versus Pressure and Sw Tables	

R

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter R	Status
ROCKCOMP – Activate Rock Compaction	
ROCKFRAC - Define the Rock Volume to Bulk Volume Fraction for All the Cells	
ROCKNUM – Define Rock Compaction Table Region Numbers	
ROCKOPTS – Define Rock Compaction and Compressibility Options	
ROCKPAMA – Define Coal Palmer-Mansorri Rock Model Parameters	
ROCKTAB – Rock Compaction Tables	
ROCKTABH – Rock Compaction Hysteresis Tables	
ROCKTABW – Rock Compaction Tables (Water Induced)	
ROCKTHSG – Rock Compaction Hysteresis Tables (Dual Porosity)	
ROCKTSIG – Rock Compaction Tables (Dual Porosity)	
ROCKWNOD – Water Saturation Values for Compaction Pressure-Sw Tables	
RPTCPL – Activate Couple Simulation Reporting	
RPTGRID – Define GRID Section Reporting	
RPTGRIDL – Define GRID Section Reporting for LGRs	
RPTHMD - Define Well History Match Gradient Reporting Options	
RPTHMG - Define Well History Match Gradient Reporting (Groups)	
RPTHMW - Define Well History Match Gradient Reporting (Wells)	
RPTINIT – Define Output to the INIT File	
RPTISOL – Activate Isolated Reservoir Number Reporting	
RPTONLY - Activate the Report Time Steps Only Option for the SUMMARY File	
RPTONLYO - Deactivate the Report Time Steps Only Option for the SUMMARY File	
RPTPROPS – Define PROPS Section Reporting	
RPTREGS – Define REGIONS Section Reporting	
RPTRST – Define Data to be Written to the RESTART File	
RPTRUNSP – Activate RUNSPEC Reporting	

R

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter R	Status
RPTSCHED – Define SCHEDULE Section Reporting	
RPTSMRY - Activate or Deactivate Summary List Report	
RPTSOL – Define SOLUTION Section Reporting	
RS – Define the Initial Equilibration GOR (Rs) for All Grid Blocks	
RSCONST – Define Constant GOR (Rs) for All Dead Oil PVT Fluids	
RSCONSTT – Define Constant GOR (Rs) for Each Dead Oil PVT Fluid	
RSGI – Define Gas-Oil Ratio versus Pressure and Gi Tables	
RSSPEC – Activate Output of the RESTART Index File	
RSVD – Equilibration Dissolved Gas-Oil Ratio (Rs) versus Depth Tables	
RTEMP - Define the Initial Reservoir Temperature for the Model	
RTEMPA - Define the Initial Reservoir Temperature for the Model	
RTEMPVD - Define the Initial Reservoir Temperature versus Depth Tables	
RUNSPEC - Define the Start of the RUNSPEC Section of Keywords	
RUNSUM – Activate RSM File Output of the SUMMARY Data	
RV – Define the Initial Equilibration CGR (Rv) for All Grid Blocks	
RVCONST – Define Constant CGR (Rv) for All Dry Gas PVT Fluids	
RVCONSTT – Define Constant CGR (Rv) for Each Dry Gas PVT Fluid	
RVGI – Define Condensate-Gas Ratio versus Pressure and Gi Tables	
RVVD – Equilibration Vaporized Oil-Gas Ratio (Rv) versus Depth Tables	
RVW – Define the Initial Equilibration Vaporized Water in Gas Ratio for All Grid Blocks	OPM Flow
RVWVD – Equilibration Vaporized Water-Gas Ratio (Rvw) versus Depth Tables	OPM Flow
RWGSALT – Water Vaporization versus Pressure and Salt Concentration	OPM Flow

S

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter S	Status
SALINITY – Define the Reservoir Salinity for All Cells	OPM FlowS
SALT – Define the Initial Equilibration Salt Concentration for All Grid Blocks	
SALTNODE – Salt Concentration Based PVTNUM Array	
SALTP – Define the Initial Precipitated Salt Volume Fraction for All Grid Blocks	OPM Flow
SALTPVD – Initial Precipitated Salt Volume Fraction versus Depth Tables	OPM Flow
SALTREST – Define the Restart Salt Concentration for All Grid Blocks	
SALTSOL – Define the Salt Solubility Limit by Region	OPM Flow
SALTVD – Equilibration Salt Concentration versus Depth Tables	
SAMG – Activate Algebraic Multi-Grid Linear Solver	
SATNUM – Define the Saturation Table Region Numbers	
SATOPTS – Activate Relative Permeability Assignment Options	
SAVE – Activate Output of a SAVE File for Fast Restarts	
SBIOF - Define The Initial Equilibration Biofilm Volume Fraction For All Grid Blocks	OPM Flow
SCALC – Define The Initial Equilibration Calcite Volume Fraction For All Grid Blocks	OPM Flow
SCALECRS – Define End-Point Scaling Option	
SCALELIM – End-Point Scaling versus Depth Maximum Water Saturation	
SCDATAB – Well Connection PI Multipliers versus Scale Deposit	
SCDETAB – Well Connection Karst Aquifer Properties for Scale Deposit	
SCDPDIMS – Define Scale Deposition and Damage Table Dimensions	
SCDPTAB – Well Connection Scale Deposition Tables	
SCDPTRAC – Allocate Sea Water Tracer for Scale Deposition	
SCHEDULE - Define the Start of the SCHEDULE Section of Keywords	
SCVD – Define Equilibration Coal Solvent Concentration versus Depth Tables	
SDENSITY – Define the Miscible or Solvent Surface Gas Density	
SEPARATE – Activate the Separate RSM File Output Option	

S

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter S	Status
SEPVALS – Define Separator Oil Formation Volume Factor and GOR	
SFOAM – Define the Initial Equilibration Foam Concentration for All Grid Blocks	
SGAS – Define the Initial Equilibration Gas Saturation for All Grid Blocks	
SGCR – End-Point Scaling Grid Cell Critical Gas Saturations	
SGCRX	
SGCRX-	
SGCRY	
SGCRY-	
SGCRZ	
SGCRZ-	
SGCWMIS – Miscible Critical Gas versus Water Saturation Functions	
SGF32D – Gas Saturation Tables versus Oil and Water Saturations	
SGFN – Gas Saturation Tables (Format Type 2)	
SGL – End-Point Scaling Grid Cell Connate Gas Saturations	
SGLX	
SGLX-	
SGLY	
SGLY-	
SGLZ	
SGLZ-	
SGLPC – End-Point Scaling Grid Cell Capillary Pressure Connate Gas Saturations	
SGOF – Gas-Oil Saturation Tables versus Gas (Format Type 1)	
SGOFLET– Gas-Oil LET Relative Permeability Functions	OPM Flow
SGU – End-Point Scaling Grid Cell Gas Saturation	
SGUX	
SGUX-	

S

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter S	Status
SGUY	
SGUY-	
SGUZ	
SGUZ-	
SGWFLET – Gas-Water LET Relative Permeability Functions	OPM Flow
SGWFN – Gas-Water Saturation Tables (Format Type 2)	
SHRATE - Activate and Define the Polymer Shearing Logarithmic Parameters	
SIGMA – Dual Porosity Matrix to Fracture Sigma (All Cells)	
SIGMAGD – Dual Porosity Matrix to Fracture Sigma for Gravity Drainage (All Cells)	
SIGMAGDV – Dual Porosity Matrix to Fracture Sigma Gravity Drainage (Individual Cells)	
SIGMAV – Dual Porosity Matrix to Fracture Sigma (Individual Cells)	
SIMULATE - Activate the Simulation Mode	
SKIP – Activate Skipping of All Keywords and Input Data	
SKIP100 – Activate Skipping of Black-Oil Keywords and Input Data	
SKIP300 – Activate Skipping of "Compositional" Keywords and Input Data	
SKIPREST – Activate Skipping of Restart Schedule Data	
SKPRPOLY – Polymer Molecular Weight Model Polymer Injection Skin Table	OPM Flow
SKPRWAT – Polymer Molecular Weight Model Water Injection Skin Table	OPM Flow
SKRO – End-Point Scaling of Grid Cell Kro(Swl) (Surfactant)	
SKRORG – End-Point Scaling of Grid Cell Kro(Sgcr) (Surfactant)	
SKRORW – End-Point Scaling of Grid Cell Kro(Swcr) (Surfactant)	
SKRW – End-Point Scaling of Grid Cell Krw(Sw =1.0) (Surfactant)	
SKRWR – End-Point Scaling of Grid Cell KRWR(Sowcr) (Surfactant)	
SLAVES – Define Slave Reservoir Simulation Parameters	
SLGOF – Gas-Oil Saturation Tables versus Gas (Format Type 1)	

S

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter S	Status
SMICR – Define The Initial Equilibration Microbial Concentration For All Grid Blocks	OPM Flow
SMRYDIMS – Define Maximum Number of Summary Vectors to be Written	
SMULTX - Multiply Cell Transmissibility in the +X Direction (Auto-Refinement)	
SMULTY - Multiply Cell Transmissibility in the +Y Direction (Auto-Refinement)	
SMULTZ - Multiply Cell Transmissibility in the +Z Direction (Auto-Refinement)	
SOCRS – End-Point Scaling Grid Cell Miscible Critical Oil Saturation with Respect to Water	
SOF2 – Oil Saturation Tables with Respect to Gas or Water (Format Type 2)	
SOF3 – Oil Saturation Tables with Respect to Gas and Water (Format Type 2)	
SOF32D – Oil Saturation Tables with Respect to Water and Gas (Three Phase)	
SOGCR – End-Point Scaling Grid Cell Critical Oil Saturation with Respect to Gas	
SOGCRX	
SOGCRX-	
SOGCRY	
SOGCRY-	
SOGCRZ	
SOGCRZ-	
SOIL – Define the Initial Equilibration Oil Saturation for All Grid Blocks	
SOLUTION - Define the Start of the SOLUTION Section of Keywords	
SOLVCONC – Define the Initial Coal Solvent Concentration for All Grid Blocks	
SOLVDIMS – Define PEBI Grid Nested Factorization Solver Dimensions	
SOLVDIRS – Define Linear Solver Principal Directions	
SOLVENT – Activate the SOLVENT Phase in the Model	
SOLVFRAC – Define the Initial Gas Solvent Fraction for All Grid Blocks	
SOLVNUM – Define PEBI Grid Correspondence to Solver Order	
SOMGAS – STONE1 Model Minimum Oil Saturation versus Gas Saturation	

S

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter S	Status
SOMWAT – STONE1 Model Minimum Oil Saturation versus Water Saturation	
SORWMIS – Miscible Residual Oil versus Water Saturation Functions	
SOWCR – End-Point Scaling Grid Cell Critical Oil Saturation with Respect to Water	
SOWCRX	
SOWCRX-	
SOWCRY	
SOWCRY-	
SOWCRZ	
SOWCRZ-	
SOXYG - Define The Initial Equilibration Oxygen Concentration For All Grid Blocks	OPM Flow
SPECGRID- Define the Dimensions of a Corner-Point Grid	
SPECHEAT – Define the Specific Heat of Oil, Water and Gas	
SPECROCK – Define the Specific Heat of the Reservoir Rock	
SPIDER – Spider Grid Activation Option	OPM Flow
SPOLY – Define the Initial Equilibration Polymer Concentration for All Grid Blocks	
SPOLYMW – Define The Initial Equilibration Polymer Molecular Weights For All Grid Blocks	OPM Flow
SSFN – Solvent and Gas Relative Permeability Tables	
SSGCR – End-Point Scaling Grid Cell Surfactant Critical Gas Saturations	
SSGL – End-Point Scaling Grid Cell Surfactant Connate Gas Saturations	
SSOGCR – End-Point Scaling Grid Cell Surfactant Critical Oil Saturation with Respect to Gas	
SSOL – Define the Initial Equilibration Solvent Saturation for All Grid Blocks	
SSOWCR – End-Point Scaling Grid Cell Surfactant Critical Oil Saturation with Respect to Water	
SSWCR – End-Point Scaling Grid Cell Critical Water Saturation	
SSWL – End-Point Scaling Grid Cell Surfactant Connate Water Saturation	
SSWU – End-Point Scaling Grid Cell Surfactant Maximum Water Saturation	

S

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter S	Status
START – Simulation Start Date	
STOG Define Capillary Pressure Oil-Gas Surface Tension versus Pressure	
STONE – Activate Stone's Second Three Phase Oil Relative Permeability Model (Alias for STONE2)	
STONE1 – Activate Stone's First Three Phase Oil Relative Permeability Model	
STONE1EX – Define Stone's First Three Phase Oil Relative Permeability Parameter	
STONE2 – Activate Stone's Second Three Phase Oil Relative Permeability Model	
STOW Define Capillary Pressure Oil-Water Surface Tension versus Pressure	
STWG Define Capillary Pressure Water-Gas Surface Tension versus Pressure	
SUMMARY - Define the Start of the SUMMARY Section of Keywords	
SUMTHIN – Define SUMMARY Data Reporting Time Steps	
SUREA - Define The Initial Equilibration Urea Concentration For All Grid Blocks	OPM Flow
SURF – Define the Initial Equilibration Polymer Concentration for All Grid Blocks	
SURFACT – Activate the Surfactant Phase in the Model	
SURFACTW – Activate the Surfactant Phase with Wettability Changes in the Model	
SURFADDW – Defined Surfactant Adsorbed Concentration versus Wettability Fraction	
SURFADS - Define Surfactant Rock Adsorption Tables	
SURFCAPD – Capillary Number versus Miscibility Tables	
SURFESAL – Define Surfactant Effective Salinity Coefficient	
SURFNUM – Define the Surfactant Miscible Saturation Table Region Numbers	
SURFROCK - Define Surfactant-Rock Properties	
SURFST - Surfactant Water-Oil Surface Tension versus Surfactant Concentration	
SURFSTES - Surfactant Water-Oil Surface Tension versus Surfactant and Salt Concentrations	
SURFVISC – Surfactant Solution Viscosity versus Concentration	
SURFWNUM – Define the Saturation Table Region Numbers (High Salinity and Water Wet)	
SWAT – Define the Initial Equilibration Water Saturation for All Grid Blocks	

S

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter S	Status
SWATINIT – Define the Initial Water Saturation Array for Capillary Pressure Scaling	
SWCR – End-Point Scaling Grid Cell Critical Water Saturation	
SWCRX	
SWCRX-	
SWCRY	
SWCRY-	
SWCRZ	
SWCRZ-	
SWF32D – Water Saturation Tables with Respect to Oil and Gas (Three Phase)	
SWFN – Water Saturation Tables (Format Type 2)	
SGWFLET – Gas-Water LET Relative Permeability Functions	OPM Flow
SWINGFAC – Define Field Gas Contract Parameters	
SWL – End-Point Scaling Grid Cell Connate Water Saturation	
SWLX	
SWLX-	
SWLY	
SWLY-	
SWLZ	
SWLZ-	
SWLPC – End-Point Scaling Grid Cell Capillary Pressure Connate Water Saturations	
SWOF – Water-Oil Saturation Tables (Format Type 1)	
SWU – End-Point Scaling Grid Cell Maximum Water Saturation	
SWUX	
SWUX-	
SWUY	
SWUY-	

S

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter S	Status
SWUZ	
SWUZ-	

T

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter T	Status
TABDIMS – Define the Number of Tables and the Table Dimensions	
TBLK – Define Tracer Initial Grid Block Concentrations	
TEMP – Activate the Temperature Modeling Option	
TEMPI – Define the Initial Temperature Values for All Cells	
TEMPNODE - Temperature Table for Polymer Solution Viscosity	
TEMPTVD – Activate Temperature Flux Limited Transport Option	
TEMPVD - Define the Initial Reservoir Temperature versus Depth Tables	
THCGAS – Define Gas Phase Thermal Conductivity for All Cells	
THCOIL – Define Oil Phase Thermal Conductivity for All Cells	
THCONR – Define Rock and Fluid Thermal Conductivity for All Cells	
THCONSF – Define Gas Saturation Dependent Thermal Conductivity Scaling Factor for All Cells	
THCROCK – Define Reservoir Rock Thermal Conductivity for All Cells	
THCWATER – Define Water Phase Thermal Conductivity for All Cells	
THERMAL– Activate the Thermal Modeling Option	
THPRES - Define Equilibration Region Threshold Pressures	
THPRESFT - Define Fault Threshold Pressures	
TIGHTEN – Tighten and Relax Numerical Controls	
TIGHTENP – Tighten and Relax Numerical Controls Individually	
TIME – Advance Simulation by Cumulative Reporting Time	
TITLE – Define the Title for the Input Deck	
TLMIXPAR – Define the Miscible Todd-Longstaff Mixing Parameters	
TNUM – Define Passive Tracer Concentration Regions	
TOLCRIT – Define The Critical Saturation Tolerance	
TOPS - Define the Depth at the Center of the Top Face for Each Cell	
TPAMEPS – Volumetric Strain versus Coal Gas Concentration Tables	

T

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter T	Status
TPAMEPSS - Volumetric Strain versus Coal Solvent Concentration Tables	
TRACER – Define Passive Tracer Variables	
TRACERKM – Multi-Partitioned Tracer Option K(P) Tables	
TRACERKP – Standard Partitioned Tracer Option K(P) Tables	
TRACERS – Activate Tracer Options and Set Tracer Array Dimensions	
TRACITVD – Activate and Define Tracer Implicit Flux Limited Transport Option	
TRACTVD – Activate Tracer Explicit Flux Limited Transport Option	
TRADS – Environmental Tracer Adsorption Tables	
TRANGL – Define Non-Neighbor Connections Between Global and LGR Cells Manually	
TRANR - Define the Transmissibility in the +R Direction for All the Cells	
TRANTHT - Define the Transmissibility in the +Theta Direction for All the Cells	
TRANX - Define the Transmissibility in the X Direction for All the Cells	
TRANY - Define the Transmissibility in the Y Direction for All the Cells	
TRANZ - Define the Transmissibility in the Z Direction for All the Cells	
TRDCY – Environmental Tracer Decay Tables	
TRDIF – Tracer Diffusion Tables	
TRDIS – Tracer Dispersion Table Number Allocation	
TRKPF – Define Partitioned Tracer Regions	
TRNHD – Activate Dispersion Non-Homogeneous Diffusion Option	
TRPLPORO – Activate the Triple Porosity Model Option	
TRROCK – Environmental Tracer-Rock Property Data	
TSTEP – Advance Simulation by Reporting Time	
TUNING - Numerical Tuning Control	
TUNINGDP – Numerical Tuning Control for High Throughput Cases	
TUNINGH – Numerical Tuning Control for History Match Gradient Calculations	

T

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter T	Status
TUNINGL - Numerical Tuning Control for All LGRs	
TUNINGS - Numerical Tuning Control for Individual LGRs	
TVDP – Define the Initial Equilibration Tracer Saturation versus Depth Functions	
TZONE – End-Point Scaling Transition Zone Options	

U

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter U	Status
UDADIMS – Define the Dimensions of the User Defined Arguments	
UDQ - Declare User Define Quantities ("UDQ")	
UDQDIMS – Define the Dimensions of the User Defined UDQ Feature	
UDQPARAM – Define Parameters for the User Defined Quantity Feature	
UDT - Declare User Define Tables ("UDT")	
UDTDIMS – Define the Dimensions of the User Defined Tables	
UNCODHMD – Activate History Match Gradient Unencoded Output	
UNIFIN – Activate The Unified Input File Option	
UNIFOUT – Activate The Unified Output File Option	
UNIFOUTS – Activate The Unified Output Summary File Option	
UNIFSAVE – Activate The Unified Output Save File Option	
USECUPL – Load a Reservoir Coupling File	
USEFLUX – Activate Flux Boundary Model and Define Flux File	
USENOFLO – Activate Flux Boundary Model Without a Flux File	

V

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter V	Status
VAPOIL – Activate the Vaporized Oil in Wet Gas Phase in the Model	
VAPPARS – Oil Vaporization Parameters	
VAPWAT – Activate Vaporized Water in the Dry and Wet Gas Phases	OPM Flow
VDFLOW – Velocity Dependent Flow Coefficient for Grid Block Flow (Grid)	
VDFLOWR – Velocity Dependent Flow Coefficient for Grid Block Flow (Region)	
VE – Activate Vertical Equilibrium Model (Global)	
VEDEBUG – Vertical Equilibrium Debug Data Output	
VEFIN – Activate Vertical Equilibrium Model (LGR)	
VEFRAC – Vertical Equilibrium Relative Permeability Fraction (Grid)	
VEFRACP – Vertical Equilibrium Capillary Pressure Fraction (Grid)	
VEFRACPV – Vertical Equilibrium Capillary Pressure Fraction (Cell)	
VEFRACV – Vertical Equilibrium Relative Permeability Fraction (Cell)	
VFPCHK – Define Production Vertical Flow Performance BHP Check	
VFPIDIMS – Injection Vertical Flow Performance Table Dimensions	
VFPINJ – Define Injection Vertical Flow Performance Tables	
VFPPDIMS – Production Vertical Flow Performance Table Dimensions	
VFPPROD – Define Production Vertical Flow Performance Tables	
VFPTABL – Define Production Vertical Flow Performance ALQ Interpolation	
VISAGE - Activate External Reservoir Geo-Mechanics VISAGE Option	
VISCD – Activate Dual Porosity Viscous Displacement Option	
VISCREF - Define Viscosity-Temperature Reference Conditions	
VISDATES – Define External Reservoir Geo-Mechanics VISAGE Stress Dates	
VISOPTS – Define External Reservoir Geo-Mechanics VISAGE Options	

W

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter W	Status
WAGHYSTR – Define Water-Alternating-Gas Hysteresis Parameters	W
WAITBAL – Wait On Network Balance Before Allowing Further Actions	
WALKALIN – Define Water Injection Alkaline Concentration	
WALQCALC – Define Well VFP Surface ALQ Phase Density	
WAPI – Define Oil Well Injection API Gravity	
WARN – Activate Warning Messages	
WATDENT – Define Water Density Temperature Coefficients	
WATER – Activate the Water Phase in the Model	
WATJT – Define Water Joule-Thomson Coefficient	OPM Flow
WATVISCT – Define Water Viscosity versus Temperature Functions	
WBHGLR – Define Well Bottom-Hole GLR Constraint	
WBOREVOL – Define Effective Wellbore Storage Volume	
WCALCVAL – Define Gas Well Calorific Value	
WCONHIST – Define Well Historical Production Rates and Pressures	
WCONINJ – Well Injection Targets and Constraints	
WCONINJE – Well Injection Targets and Constraints	
WCONINJH – Well Historical Observed Injection Rates and Pressures	
WCONINJP – Define Well Injection Targets and Constraints for Pattern Flood Wells	
WCONPROD – Define Well Production Targets and Constraints	
WCUTBACK – Define Well Cutback Limits and Parameters	
WCUTBACT – Define Well Tracer Cutback Limits and Parameters	
WCYCLE – Define Automatic Well Opening and Closing Cycling Parameters	
WDFAC – Define Gas Flow Dependent Skin Factor	
WDFACCOR – Gas Flow Dependent Skin Factor (Correlation)	
WDRILPRI – Add Wells to the Drilling Priority Drilling Queue	

W

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter W	Status
WDRILRES – Activate Prevention of Multi-Completions in the Same Cell for Queued Wells	
WDRILTIM – Define Drilling Parameters for Automatic Drilling of New Wells	
WECON – Well Economic Criteria for Production Wells	
WECONINJ – Well Economic Criteria for Injection Wells	
WECONT – Well Economic Tracer Criteria for Production Wells	
WEFAC – Define Well Efficiency	
WELCNTL – Modify Well Control and Targets	
WELDEBUG – Define the Well Debug Data to be Printed to File	
WELDRAW – Define Maximum Draw Down for Production Wells	
WELEVNT – Define Well WPWEM Summary Mnemonic Output Value	
WELLDIMS – Define the Wells and Group Dimensions	
WELLSTRE – Define Injection Stream Composition	OPM Flow
WELMOVEL – Move Global Well Into an LGR	
WELOPEN – Define Well and Well Connections Flowing Status	
WELOPENL – Define Well and Well Connections Flowing Status (LGR)	
WELPI – Define Well Productivity and Injectivity Indices	
WELPRI – Assign Well Priority	
WELSEGS – Define Multi-Segment Wells and Their Segment Structure	
WELSOMIN – Define Well Connection Minimum Oil Saturation for Opening	
WELSPECL – Define Well Specifications for Local Grid Refinements	
WELSPECS – Define Well Specifications	
WELTARG – Modify Well Target and Constraint Values	
WELTRAJ – Define Well Trajectory Data	OPM Flow
WFOAM - Define Well Foam Injection Concentrations	
WFRICSEG – Convert Friction Well to Multi-Segment Well	

W

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter W	Status
WFRICSGL – Convert Friction Well to Multi-Segment Well (LGR)	
WFRICTN – Define Well as a Friction Well	
WFRICTNL – Define Well as a Friction Well (LGR)	
WGASPROD – Define Sale Gas Well Production Targets	
WGORPEN – Define Well GOR Penalty Parameters	
WGRUPCON – Define Well Guide Rates for Group Control	
WH2NUM – Define WAG Hysteresis Saturation Table Region Numbers (Two Phase)	
WH3NUM – Define WAG Hysteresis Saturation Table Region Numbers (Three Phase)	
WHEDREFD – Define Well Hydraulic Head Reference Depth	
WHISTCTL - Define Well Historical Target Phase	
WHTEMP – Define Well Tubing Head Temperature Parameters	
WINJCLN – Clean a Fraction of a Deposited Filter Cake	
WINJDAM – Define Well Filter Cake Properties	
WINJFCNC – Define Injection Well Filtrate Concentration	
WINJGAS – Define Gas Injection Properties for a Well	OPM Flow
WINJMULT – Define Well Pressure Dependent Injectivity Multipliers	
WINJTEMP – Define Injection Fluid Thermal Properties	
WLIFT – Define Well Re-Tubing, THP and Lift Switching Workover Operations	
WLIFTOPT – Define Well Gas Lift Optimization Parameters	
WLIMTOL – Define Well Constraint Tolerance	
WLIST – Define Well Lists (Static)	
WLISTARG – Modify Well List Target and Constraint Values (Static)	
WLISTNAM – Define Well Lists (WLISTARG)	
WMICP – Define Water Injection Well's Microbial, Oxygen, And Urea Concentrations	OPM Flow
WNETCTRL – Define Well Control for Network Control Option	

W

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter W	Status
WNETDP – Define Well THP to Network Pressure Drop	
WORKLIM – Define Well Workover Time	
WORKTHP – Define Well Workover Options for THP Killed Wells	
WPAVE – Well Block Average Pressure Calculation Parameters for All Wells	
WPAVEDEP – Define Well Reference Depth for Pressure Calculations	
WPIMULT – Define Well Connection Multipliers	
WPIMULTL – Define Well Connection Multipliers (LGR)	
WPITAB - Assign Well Productivity Index versus Water Cut Tables	
WPLUG – Define Well Plug Back Length	
WPMITAB - Assign Well Polymer Molecular Model Injection Tables	OPM Flow
WPOLYMER - Define Water Injection Well Polymer and Salt Concentrations	
WPOLYRED – Define Well Polymer-Water Viscosity Reduction Factor	
WPOTCALC – Well Potential Calculation Options	
WREGROUP – Automatic Re-Assignment of Wells to Groups	
WRFT – Activate Well RFT Reporting to the RFT File	
WRFTPLT – Activate Well RFT and PLT Reporting to the RFT File	
WSALT - Define Water Injection Well Salt Concentrations	
WSCCLEAN – Well Deposited Scale Adjustment	
WSCCLENL – Well Deposited Scale Adjustment (LGR)	
WSCTAB – Assign Well Scale Deposition and Scale Damage Tables	
WSEGAICD – Define Multi-Segment Well Autonomous ICD Connections	
WSEGDFIN – Define Multi-Segment Well Drift Flux Slip Model Input Data	
WSEGDFMD – Define Multi-Segment Well Drift Flux Slip Model	
WSEGDFPA – Define Multi-Segment Well Drift Flux Slip Model Parameters	
WSEGDIMS – Define Multi-Segment Well Dimensions	

W

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter W	Status
WSEGEXSS – Define Multi-Segment Well Import-Export Segment Volumes	
WSEGFLIM - Define Multi-Segment Well Artificial Choke Connections	
WSEGFMOD – Define Multi-Segment Well Model	
WSEGINIT - Define Multi-Segment Well Initial Conditions	
WSEGITER – Define Multi-Segment Wells Iteration Parameters	
WSEGLABY - Define Multi-Segment Well Labyrinth ICD Connections	
WSEGLINK - Define Multi-Segment Well Looped Flow Paths	
WSEGMULT – Define Multi-Segment Well Frictional Pressure Loss Multipliers	
WSEGPROP – Modify Multi-Segment Wells and Their Segment Structure	
WSEGPULL – Define a Multi-Segment Well Down-Hole Separator Pump	
WSEGSEP – Define a Multi-Segment Well Down-Hole Separator	
WSEGSICD – Define Multi-Segment Well Spiral ICD Connections	
WSEGSOLV Define Multi-Segment Well Iterative Linear Solver Parameters	
WSEGTABL – Assign Multi-Segment Well VLP Tables to Segments	
WSEGVALV – Define Multi-Segment Well Sub-Critical Valve	
WSF - Water Saturation Tables versus Water Saturation (Gas-Water and CO2STORE Systems)	OPM Flow
WSKPTAB - Assign Well Polymer Molecular Model Skin Tables	OPM Flow
WSOLVENT - Define Gas Injection Well Solvent Fraction	
WSURFACT - Define Water Injection Well Surfactant Concentration	
WTADD – Add a Constant to a Well Target or Constraint	
WTEMP – Define An Injection Well's Fluid Temperature	
WTEMPQ – Output Well Names and Well Lists to the Print File	
WTEST – Well Testing Criteria for Re-Opening Closed Wells	
WTHPMAX – Define a Well's Maximum Flowing THP for Shut-In	
WTMULT – Multiple a Well Target or Constraint by a Constant	

W

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter W	Status
WTRACER – Define An Injection Well's Tracer Concentration	
WVFPDP – Modify Well BHP Obtained from VFP Tables	
WVFPEXP – Define Well VFP Interpolation Options	
WWPAVE – Well Block Average Pressure Calculation Parameters for Individual Wells	

X

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter X	Status
There are No Keywords Beginning with the Letter X	

Y

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter Y	Status
There are No Keywords Beginning with the Letter Y	

Z

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter Z	Status
ZCORN – Define the Depth of Each Corner-Point of a Grid Block	
ZIPP2OFF – Deactivate Automatic Time Step Control	
ZIPPY2 – Activate Automatic Time Step Control	
ZMFVD – Define Compositional Components versus Depth	

APPENDIX B: OPM FLOW RELEASE HISTORY

APPENDIX TABLE OF CONTENTS

B.1 RELEASE 2023-10

We dedicate this OPM 2023-10 release to our colleague and friend David Baxendale. David passed away in late June 2023 after a short severe illness. Our thoughts are with his wife and son. The OPM community is very thankful for all his contributions to OPM and the fruitful discussions with him about issues with and advancements of the simulator.

David started contributing to OPM in 2016 as *OPMUSER* on github and continued his good work until his very last days. We owe the OPM Reference manual to him. He started this heroic effort in 2017 and it now has thousands of pages. We, his colleagues and friends, are and will be surely missing him with his reservoir engineering expertise and know-how, his enthusiasm, and humor.

The 2023-10 release consists of some new features and various improvements and bug fixes. Our main target was to support more keywords used for relevant fields and reduce differences between OPM flow and the commercial simulator. These improvements include:

- Added support for temperature (THERMAL) plus salt precipitation (PRECSALT) modeling in gas-water-brine (GAS-WATER-BRINE) systems.
- Added support for modeling dissolved gas in water (DISGASW) and vaporized water in the gas phase (VAPWAT) in the thermal-gas-water simulator.
- Support for modeling FOAM combined with SOLVENT.
- Partial support for WAGHYSTR keyword (Water-Alternating-Gas hysteresis).
- Improvements to many user-facing error messages.
- More graceful exits for problems in parallel runs.
- Temperature is output to the RESTART file if requested via RPTRST.
- Added support for WBP, WBP4, WBP5 and WBP9 in the SUMMARY section to output well block averaged pressures for open completions.
- Added support for initializing constant flux aquifers from a restart.
- Faster two-point flux-approximation introduced in the last release is now also used for linearizing gas-oil cases with energy (with diffusion) and gas-oil diffusion.

B.1.1 NEW AND DEPRECATED COMMAND LINE OPTIONS

The major command line changes made for this release are summarized in Table B.1

OPM Flow 2023-10 New And Deprecated Command Line Options			
No.	**Variable Name**	**Description**	**Default**
1	--linear-solver	Added option to use the experimental hybrid solver configuration. Valid options are: ilu0 (default), cprw, cpr (an alias for cprw), cpr_quasiimpes, cpr_trueimpes, amg or hybrid (experimental). Alternatively, you can request a configuration to be read from a JSON file by giving the filename here, ending with '.json.'.	ilu0
2	--linear-solver-print-json-definition	Added option to write the JSON definition of the linear solver setup to the DBG file.	true

OPM Flow 2023-10 New And Deprecated Command Line Options			
No.	**Variable Name**	**Description**	**Default**
3	--linear-solver-reduction	The minimum reduction of the residual which the linear solver must achieve.	0.01
4	--local-domain-ordering-measure	Added parameter to specify the domain ordering measure as either residual or pressure when using the NLDD solver (#4738).	pressure
5	--local-domains-partitioning-imbalance	Subdomain partitioning imbalance tolerance for the NLDD solver. 1.03 is 3 percent imbalance.	1.03
6	--local-domains-partitioning-method	Subdomain partitioning method for the NLDD solver. Allowed values are 'zoltan', 'simple', and the name of a partition file ending with '.partition'.	zoltan
7	--local-solve-approach	Choose local solve approach for the NLDD solver. Valid choices are jacobi and gauss-seidel.	jacobi
8	--local-tolerance-scaling-cnv	Set lower than 1.0 to use stricter convergence tolerance for local solves when using the NLDD solver.	0.01
9	--local-tolerance-scaling-mb	Set lower than 1.0 to use stricter convergence tolerance for local solves when using the NLDD solver.	1
10	--local-well-solve-control-switching	Added trigger to enable (true) or disable (false) well control/status switching during local well equation solves when using the NLDD solver (#4895).	false
11	--max-local-solve-iterations	Max iterations for local solves with NLDD nonlinear solver.	20
12	--maximum-water-saturation	Maximum water saturation.	1
13	--network-max-iterations	Maximum number of iterations in the network solver before giving up.	200
14	--network-max-strict-iterations	Maximum iterations in network solver before relaxing tolerance.	100
15	--nldd-num-initial-newton-iter	Added option to specify number of global non-linear (Newton) iterations performed by the NLDD solver before starting the local non-linear iterations. The default value of 1 preserves the current behavior (#4922).	1
16	--nonlinear-solver	Choose nonlinear solver. Valid choices are newton or nldd.	newton

OPM Flow 2023-10 New And Deprecated Command Line Options			
No.	**Variable Name**	**Description**	**Default**
17	--num-local-domains	Number of local domains for NLDD nonlinear solver. Note this is an experimental feature in the current release 2023.10 that is expected to be more complete and tested by the 2024.10 release.	0
18	--save-step	Added new options to keep only the last step. Save serialized state to .OPMRST file. Either a specific report step, "all" to save all report steps or ":x" to save every x'th step. Use negative values of "x" to keep only the last written step, or "last" to save every step, keeping only the last (#4807).	""
19	--water-only-threshold	Cells with water saturation above or equal to this value are considered one-phase water only.	1
20	--load-file	FileName for .OPMRST file used to load serialized state. If empty (""), CASENAME.OPMRST is used.	""
21	--num-pressure-points-equil	Number of pressure points (in each direction) in tables used for equilibration (#4718).	2000
22	--save-file	FileName for .OPMRST file used for saving serialized state. If empty, CASENAME.OPMRST is used.	""
23	--use-average-density-ms-wells	Approximate segment densitities by averaging over segment and its outlet.	false
24	--ecl-max-time-step-size-after-well-event	Maximum time step size after a well event (seconds). Default is equivalent to 1 year. Depreciated.	3.15576e+07
25	--ecl-restart-shrink-factor	Factor by which the time step is reduced after convergence failure. Depreciated	3

Notes:

1) Cells colored in green in the "No." column indicate a new command line option or a change to an existing option for this release.
2) Cells colored in orange in the "No." column indicate the command line option is available but is now "hidden" from the main help listing, --help. These options can be listed using the --help-all command line option, that lists all the command line options included in the release, including experimental, obsolete, hidden and deprecated options.
3) Cells colored in red in the "No." column indicate a deprecated command line option for this release.

Table B.1: OPM Flow 2023-10 New and Deprecated Command Line Options

B.1.2 NEW FEATURES

In addition to the above the following new features have been added to the simulator:

1) When the general well specification data is defined by the WELSPECS keyword in the SCHEDULE section the well can now be assigned directly to the FIELD group in item 2 GRPNAME (#3485 and #4608). Previously there was a restriction preventing wells from being parented directly to FIELD. Although wells are now allowed to be parented directly to FIELD, this is discouraged and a warning message will be issued. Mixing wells and groups as children of a single group is still forbidden.

2) Added support for dissolved gas in water (DISGASW) in the gas-water simulator with salt precipitation (PRECSALT) and vaporized water in the gas phase (VAPWAT) (#4623). However, input of gas solubility in water is not currently supported so this option is only useful in combination with CO2STORE.

3) Added support for modeling dissolved gas in water (DISGASW) and vaporized water in the gas phase (VAPWAT) in the thermal-gas-water simulator (#4661).

4) Added support for temperature (THERMAL) plus salt precipitation (PRECSALT) modeling in gas-water-brine (GAS-WATER-BRINE) systems as specified in the RUNSPEC section (#4650). This has to date only been tested in combination with CO2STORE. Note that generally, if the PRECSALT keyword has been activated in the input deck then the VAPWAT keyword should also be activated.

5) Added support for modeling foam (FOAM) plus solvent (SOLVENT) in the simulator (#4654, #3523 and #805). In addition, gas or water is now allowed as the transport phase for the foam as specified by FOAMOPTS item 1 (previously only gas was supported as the foam transport phase).

6) Added support for WVFPDP in the SCHEDULE section (#4620 and #3504). The WVFPDP keyword modifies a well's Bottom-Hole Pressure ("BHP") estimated by the simulator by interpolation of the Vertical Flow Performance ("VFP") tables.

7) Added support for GCONPROD item 7 ACTION equal to NONE in the SCHEDULE section, which specifies that no action is to be taken if the oil, water, gas or liquid rate constraints are violated. Previously only the RATE option was supported by the simulator (#4658).

8) Added partial support for WAGHYSTR keyword in the PROPS section (#4710 and #3542). This keyword defines the parameters for the Water-Alternating-Gas ("WAG") hysteresis option, when the hysteresis option has been activated by the HYSTER variable on the SATOPTS keyword in the RUNSPEC section. Only gas phase hysteresis is currently supported by the WAGHYSTR keyword. The residual oil modification fraction, which would only be active when the STONE1 three-phase oil relative permeability model is used, is not currently supported.

9) Support added for PPCWMAX to limit the maximum capillary pressure scaling when initializing the model using SWATINIT, and the option to modify the connate water saturation to match the input water saturation if the capillary pressure is exceeded (#4707 and #3570).

10) Added support for WINJMULT in the SCHEDULE section to define pressure dependent injectivity multipliers for injection wells and can be used to approximate the change in injectivity due to hydraulic fracturing (#4686).

11) Added partial support for WPAVE and WWPAVE defining the method and parameters for calculating a well's block average pressures for either all wells or specific wells (#4695, #4694 and #4693). Calculation of block averages pressures is currently only supported for OPEN completions (not for ALL completions) as specified in WPAVE item 4 and WWPAVE item 5.

12) Aquifer keywords have been enabled when H2STORE is specified in the RUNSPEC section with brine modeled by the OIL phase (#4791). This is similar to the behavior when CO2STORE is specified.

13) Added a model for formation damage due to suspended solids in the injection water (#4346 and #3313). New OPM specific well keywords WINJDAM, WINJFCNC and WINJCLN have been added to the SCHEDULE section. WINJDAM sets up the filter cake properties for the specified water

injection well, WINJFCNC defines the filtrate concentration in the injection water, and WINJCLN specifies that a fraction of the filter cake has been cleaned up.

14) User defined arguments (UDA) have been enabled for the WINJFCNC keyword item 2 (FCONCPPM) the injection concentration parameter (*#4763* and *#3614*).

15) Added partial support for GRUPNET, which defines the standard group network parameters used to model the flow and pressure behavior within the network (*#4760*, *#3609*, *#4815* and *#3651*). GRUPNET item 5 OPTION1 equals NO only is supported (defines how the groups production target should be achieved). Item 6 OPTION2 equals NO or FLO is supported but ALQ is not supported (defines how gas lift gas flows through the group's pipeline). Item 7 OPTION3 equals NONE only is supported (defines if the ALQ-PIPE variable should be reset).

16) Added partial support for GECON (*#4819* and *#3657*). Item 7 WORKOVER only supports NONE, workover procedures are not currently implemented. Item 8 ENDRUN only supports NO, end run is not currently implemented. Item 9 MXWELLS is not supported and must be defaulted.

17) Support has been added for the CSKIN keyword in the SCHEDULE section to modify the connection skin factor (*#4871* and *#3681*).

18) Support for WSF and GSF keywords in the PROPS section has been added for H2STORE runs (as for CO2STORE) (*#4528*). The WSF and GSF keywords define the water relative permeability data versus water saturation tables and the gas relative permeability data versus gas saturation tables for when only gas and water are present in the input deck. These keywords can only be used with either the CO2STORE or H2STORE models.

19) Support has been added for setting the WGRUPCON keyword Item 2 STATUS in the SCHEDULE section equal to NO (*#4575*).

20) Support has been added for setting the PINCH item 2 PINCHOPT parameter equal to NOGAP in the GRID section. This parameter is now fully supported by the simulator. Previously only the default GAP option was supported (*#4603*).

21) Added support for initializing constant flux aquifers from a restart (*#4520* and *#4519*). In particular, form constant flux aquifer objects from the restart step's collection if available and properly initialize their total produced volume.

22) Added support for gas-water-solvent systems (*#4568* and *#4548*).

23) Added partial support for setting items 11, 12 and 13 of GCONPROD in the SCHEDULE section: ACTWAT, ACTGAS and ACTLIQ respectively define the action to be taken if the water rate (WRAT), gas rate (GRAT) and liquid rate (LRAT) constraints defined by GCONPROD are violated. Supported options are now NONE or RATE. Options CON, +CON, WELL, and PLUG are not currently supported (*#4748*).

24) Added support for Non-linear Domain Decomposition (NLDD) as the non-linear solver.

25) The option has been added to allow well control or well status to be updated during the iteration process during local solve for the well equations (*#4895*). In this way the converged well is given the correct control/status for the current reservoir state. A command line argument *–local-well-solve-control-switching=true* has been added to trigger the use of the function (this is *false* by default).

B.1.3 IMPROVEMENTS

Improvements include:

1) The correct version of flow is now selected for two-phase gas-water systems with either water vaporized in the gas phase (VAPWAT) and/or gas dissolved in the water phase (DISGASW) as defined in the RUNSPEC section (*#4592*). Previously the wrong version of flow was selected for gas-water systems with water vaporized in the gas phase (but no gas dissolved in the water phase).

2) The boundary conditions keyword BC has been deprecated and been split into two parts: BCCON in the GRID section where the block indices and direction are set, and BCPROP in the SCHEDULE section where the type and value of the boundary condition are set (*#3482*, *#802* and *#949*).

3) The maximum number of allowable EQLNUM regions has been increased from 255 to 65525 (*#4726*).

4) A stricter convergence tolerance has been imposed on standard (not multi-segment) wells with a zero rate target. This change improves the results of a reported case with zero reservoir volume rate (RESV) control and potentially improves the material balance of the whole system (*#4572*).

5) When testing a gas lift well under THP control, if the well does not converge with the maximum artificial lift quantity (ALQ) then the simulator will now try to reduce ALQ in increments to check if the well equations converge with a smaller ALQ (*#4579*).

6) The bisection algorithm has been updated to keep track of the size of its search interval. The algorithm will now stop iterating if the minimum interval size is reached (*#4617*).

7) The simulator has been modified to ensure that THP constraints are only applied to prediction wells and not history match wells (*#4615*).

8) The simulator has been updated to avoid requesting the ALQ value for injectors, the ALQ value is only requested for producers (*#4648*).

9) The simulator has been modified to output temperature information to the restart file if TEMP is specified in the RPTRST keyword or if THERMAL is set in the RUNSPEC section even if the command line option *--enable-opm-rst-file=false* is set (*#4646*). The command line option *--enable-opm-rst-file* is a Boolean value that controls the output of OPM specific data sets to the commercial simulator's restart file to enable restart of OPM Flow runs by OPM Flow (*true*), or not to output the data (*false*).

10) Update the calculation of immobile gas saturation to take into the account trapped saturation (calculated when the hysteresis model is used) in the calculation of mobile and immobile fluid volumes in place (*#4642* and *#3517*). The hysteresis model is activated by the specifying the HYSTER parameter of the SATOPTS keyword in the RUNSPEC section.

11) An error message will now be generated if invalid region numbers SATNUM, PVTNUM, IMBNUM or EQLNUM are input (*#4705*). Valid region numbers are positive integers less than or equal to the maximum number of regions specified in the RUNSPEC section.

12) Added setVapPars() at the start of the report step to properly handle oil vaporization parameters (VAPPARS) updates in the SCHEDULE section (*#4677*).

13) If vaporised water (VAPWAT) is present in the model but the initial equilibrium vaporized water in gas ratio (RVW) for an equilibration region is not explicitly defined by a value versus depth table (RVWVD), and the datum depth is not at either the gas-oil contact (if oil is present) or the gas-water contact (if oil is not present) then the values of RVW will be initialised to zero (*#4647* and *#4688*). This makes it easier to include VAPWAT in CO2STORE cases.

14) The mass balance limit (XXXMBE) from the keyword TUNING is now used if it is explicitly specified (not defaulted) and the command line argument *--enable-tuning=true* is used (*#4621* and *#3522*).

15) The simulator now gives a more informative error message if the input grid has no active cells at all. Often this points to an error in the input data and this change might help in finding the problem (*#4735*).

16) A more user-friendly error message is now reported when the time step is cut too often or too much. (*#4746*).

17) Perform a more graceful exit instead of MPI_Abort for expected exceptions in parallel runs. Instead of unconditionally issuing MPI_Abort if a fatal exception is encountered, the simulator will try to test whether all processes have experienced this exception and if this is the case just terminate normally with an exit code that signals an error. MPI_Abort is still used if not all processes get an exception as

this is the only way to ensure that the program aborts (#4750). This approach also works around issues in some MPI implementations that might not correctly return the error.

18) Hydrostatic and acceleration pressure losses have been included for well segments representing an inflow control device (ICD) (#4824). Valves may be placed in long segments with significant depth differences (for example, at branch inlets), where the hydrostatic contribution in particular may be significant.

19) An error message is now reported if a negative oil or water saturation is passed to the RV and RVW initialization routine (#4675).

20) When approximately zero well rates are encountered during iterations (pre-convergence), gas/water fractions become highly inaccurate which in turn may lead to the solver getting stuck or the well getting shut prematurely. The code has been updated to switch to explicit VFP table lookup whenever the rate drops below the lowest value in the table. The logic around explicit lookup is also updated so that this approach also works for just opened wells (#4669).

21) Information about linear system sizes is now output to the DBG file (#4734 and #4754).

22) For stopped or zero-rate-target wells the *alternative_well_rate_init* procedure previously returned unscaled well-rates resulting from a zero-bhp condition. This could lead to convergence failures for network balancing since the initialized rates could be off by orders of magnitude. The code now skips this procedure for stopped or zero-rate-target wells reducing the risk of convergence failures (#4817).

23) The two-point flux-approximation (TPFA) has been added as a linearizer for gas-oil cases with energy (with diffusion) and gas-oil diffusion (#4825 and #4816). All energy cases now have diffusion enabled.

24) Added RESTART file output for the geomechanical module (#4803 and #4588).

25) Relaxation factors slightly outside the interval [0, 1] in the standard well model are reset to the interval limits, while keeping the assertion for factors further outside the interval to possibly pick up failure cases (#4862).

26) Connections between reservoir cells and numerical aquifer cells, or between numerical aquifer cells when multiple such cells define a single numerical aquifer, are now always treated as NNCs for output purposes (#4821).

27) Regional transmissibility multipliers such as those entered in the MULTREGT keyword are now applied to explicit input non-neighbor connections (NNC). This now implements all known connection behaviors for inter-region connections. In the commercial simulator, if the region numbers specified in MULTREGT items 1 and 2 are equal and positive then the transmissibilities within the region as well as any transmissibilities connecting any other regions are multiplied by the given value. The option to specify transmissibility multipliers in this way using MULTREGT is not yet supported (#4822 and #4821).

28) The name of the missing OPM restart file is reported in the error message when it cannot be located (#4870).

```
Error locating serialized restart file PATH/FILENAME.OPMRST
```

29) Added PRT file output when group economic criteria for production groups GECON is activated (#4866).

30) Reduced repetitive output of network information when running in parallel (#4879).

31) The lower limit for bottom hole pressure (BHP) in Newton updates has been reduced slightly from 1 bar. This allows for cases that might have a defaulted BHP constraint of 1 bar (#4877).

32) The region set name matching algorithm has been changed to using unique prefixes. This enables the simulator to recognize that the region set name FIPUNI should match the user defined region name FIPUNIT (#4868).

33) Debug output sent to the DBG file is no longer also echoed to the console as well (#4955 and #4941).

34) Updated the derivatives and also the Jacobian matrix for the multisegment well pressure equations (#4640).

35) The intensive quantities for timeIdx==1 corresponding to the start of the current timestep are now updated if we do not use the storage cache, or if we cannot recycle the first iteration storage (#4662).

36) Improved message issued when shutting a well because it fails to converge (#4741).

37) Damaris was updated in version 1.8.0 to support the HDF5 *H5Sselect_elements()* capability to rewrite data in memory to another order on disk. This allows (MPI decomposed) local simulation data to be written back to its original global position on disk. To support this a new element was added to the Damaris XML <variable...> type, named "select-file", along with some other options (not required by OPM Flow). This keyword is added to the basic in-built Damaris XML file of OPM Flow (#4830).

In addition, the following new SUMMARY keywords are now recognized as described by the comments in Table B.2.

No.	Summary Keyword	Comment
1	WBP, WBP4, WBP5 and WBP9	Added support for SUMMARY vectors to output well block averaged pressures for open completions (#4695, #4694 and #4693).
2	WINJFVR, WINJFVT and WINJFC	Added support for SUMMARY keywords relating to filtration injection modelling (#4790, #3628 and #3626). Added injection well SUMMARY vectors for filtrate volume injection rate (volume rate), filtrate volume injection total (volume), and filtrate injection concentration (ppm).
3	CINJFVR, CINJFVT, CFCSKIN, CFCWIDTH, CFCPERM, CFCPORO, CFCRAD and CFCAOF	Added support for SUMMARY keywords relating to filtration injection modelling (#4790, #3628 and #3626). Added injection well connection SUMMARY vectors for filtrate volume injection rate (volume rate), filtrate volume injection total (volume), skin factor due to filtration (dimensionless), thickness of filter cake (Length), permeability of filter cake (MD), porosity of filter cake (dimensionless), well bore radius used in the filtration modeling (Length), and well bore area of flow used in the filtration modeling (Area).
4	SxDEN, SDENM and SMDEN	Added support for SUMMARY vectors to output fluid densities at the block and well segment levels (#4744, #3594 and #3593). Added well segment SUMMARY vectors for phase density of phase x (segment conditions), fluid mixture density weighted by phase flowing fractions, and fluid mixture density weighted by phase flowing fractions with exponents. Phase x is one of O (oil), G (gas), or W (water).
5	BxDEN and BDENx	Added support for SUMMARY vectors to output fluid densities at the block and well segment levels (#4744, #3594 and #3593). Added block SUMMARY vectors for phase density of phase x. Phase x is one of O (oil), G (gas), or W (water).
6	BFLOWI, BFLOWJ, and BFLOWK	Added support for SUMMARY vectors BFLOWI, BFLOWJ and BFLOWK to request block oil/gas/water flow rates at surface conditions (#4867, #3675 and #827). This does not require FLOWS to be specified in RPTRST.

Table B.2: New SUMMARY Keywords for the 2023-10 Release

B.1.4 BUG FIXES

The following bug fixes and improvements have been incorporated into this release.

1) Fixed the issue of incorrectly using the gas-oil contact depth for initialization of vaporized water (RVW) in the special case of a two-phase gas-water system. The gas-water contact depth is now used in this case (*#4688* and *#4647*).

2) Fixed a bug in the group pressure maintenance (GPMAINT) code (*#4664*). Previously, if negative group pressure maintenance rates were calculated, they would first be used to incorrectly update the GPMAINT error integral, before the rates were later set to zero. Now the simulator only calculates rates for injectors if the pressure is below the regional pressure target (or for producers if the pressure is above the regional pressure target).

3) Bug fixes related to the handling of gas dissolved in water ratio (RSW) and water vaporized in gas ratio (RVW) in the standard well model (*#4591*).

4) The max size of the next time step is now only used when specified in TUNING or NEXTSTEP (*#4660*).

5) If the data file contains the CPR keyword then the "CPR" preconditioner should be used unless it is overridden by a command line argument. Previously this was incorrectly overridden by the command line argument --linear-solver-max-iter, the correct command line argument --linear-solver is now used (*#4700*).

6) The user is now able to specify any number of threads, this prevents only one thread being used on some hardware where the number of processors would always be reported as one when using MPI and OpenMP (irrespective of the actual number). The behaviour is now: (a) if nothing is specified then 2 threads are used, (b) if OMP_NUM_THREADS is specified then this number is used regardless of the hardware and the command line argument --threads-per-process is ignored, and (c) if --threads-per-process is used and OPM_NUM_THREADS is not set then the number specified on the command line is used (*#4709*). There is a check to make sure that the number of threads used by the linearizer is consistent (*#811*).

7) Fixed a bug related to indexing in the temperature boundary condition (*#4761*).

8) Gas dissolved in oil (RS) and oil vaporized in gas (RV) ratio initialized using value versus depth tables (RSVD, RVVD) should be limited by their respective saturated values. Fixed a bug where this limit was not applied at depths outside the range of depths in the RSVD or RVVD table (*#4723*).

9) For producers where all perforations have zero rates the perforation mixture fraction is approximated using the (inverse formation volume factor times mobility) ratio, and weight the perforation rates using the well transmissibility. The perforation mixture fraction was previously approximated using only the mobility ratio (*#4681*).

10) A case under investigation has some cells containing only oil and water (zero gas saturation), which have zero relative permeability (and mobility) for all the three phases. This caused problems in part of the code which assumed the total mobility was non-zero. For perforations having zero mobility for all the phases, the simulator now uses a small value to generate small perforation rates for those perforations, at the same time, the simulator can use these rates to recover the mixing ratios for those perforations (*#4682* and *#4681*).

11) The correct formation volume factors are now used in the case of zero phase rates resulting in zero RS, RV, RSW or RVW. Previously the saturated formation volume factor was used in these cases (*#4590*).

12) The reservoir volume rate constraint (RESV) in GCONPROD was not honoured. This has been fixed (*#4687*).

13) Previously, in the case of zero threshold pressure and zero pressure difference, the code would set the pressure difference explicitly to zero. This would also set any derivatives to zero, which could disconnect the corresponding matrix rows. The code no longer sets the pressure difference to zero when not necessary (#4701).

14) Code has been updated to avoid dividing by zero when scaling the well rates in updateWellStateRates() (#4715 and #4649).

15) The code has been updated to avoid round off errors in phase saturations leading to the initialization of dissolved or vaporized fluid ratios (for example saturated RS) with saturated values instead of taking values from the input ratio versus depth table (for example RSVD) (#4720).

16) Code has been updated to avoid writing beyond array limits (#4753).

17) The well closure reason is now set to GROUP rather than ECONOMIC if a group economic constraint (GECON) is reached. This prevents the closed well being reopened with WTEST item 3 TEST equal to 'E' (#4854).

18) The simulator now sets the well THP to be zero in the WellState only if there is no VFP table associated with the well (#4932). Previously, this would be done if the well had no target THP or THP limit.

19) The code now checks whether LIFTOPT is active first to avoid unnecessarily running routines relating to gas lift optimization (#4956).

20) MPI is now initialized before creating the communicator and a couple of fields are now set to ROOT_ONLY in *opmrst_inspect()* (#4601).

21) The *updateWaterMobilityWithPolymer()* method updates values, it does not rewrite them. Hence we have to feed it the scalar values not zeros. Scalar values are now used (#4655).

22) The restriction on the length of the next timestep following an event will now be applied following production or injection updates (WCONPROD, WCONINJE keywords) (#4781). The maximum length of the next timestep following an event can be set using either the TUNING keyword item 10 (TMAXWC) with the command line argument *--enable-tuning=true*, or by using the command line argument *--time-step-after-event-in-days=x* (where *x* is the number of days).

23) Added perforation data (perf_data) comparison in the equality operator for SingleWellState (#4783).

24) Added check that rock compaction table indices (ROCKNUM) are within the bounds of the tables defined by ROCKTAB (#4788).

25) Code modified to avoid a possible segmentation fault in the cleanup routine if the simulator has not been set up (#4794).

26) Fixed a bug where the code could attempt to access entries in a zero element array due to not dereferencing a pointer to inspect array elements when deciding whether to apply TRANX, TRANY, etc. keywords (#4801).

27) Code has been modified to prevent slightly negative oil fractions in well segments occurring due to round-off. This could have lead to failure if the segment represented for example an autonomous Inflow Control Device (ICD) valve in a multi-segment well (#4834).

28) Fixed a bug with the DIRICHLET option in the boundary condition property definition (BCPROP) not working in gas-water runs. Additionally, fixed a bug where BCPROP had to be defined at every report step (#4835).

29) If a well was SHUT due to economic or physical reasons the well-state quantities were mostly set to zero (including the BHP). The code has been modified to set the BHP to the BHP limit when its value has not been initialized when under well target rate control (#4841). This helps to prevent the Newton update from stagnating under certain circumstances (for example, a reported issue introduced by #4772).

30) Fixed a bug which occurred when using non-neighbor connections (NNC) in thermal simulations (#4900).

31) Fixed computation of temperature for distributed wells (#4888). This was not an issue for non-distributed wells.

32) Code has been modified to always allocate buffers for storing flows if BFLOWI, BFLOWJ or BFLOWK is requested. This fixes a segmentation fault that occurred if these keywords were requested without RPTRST in the SOLUTION section (#4904).

33) Code has been modified so that there will be no vertical connection between cells if the layers in between are inactive or collapsed unless the PINCH keyword has been specified (#4901).

34) Fixed a bug which occurred in parallel runs where all processes other than rank zero would keep iterating beyond the maximum number of allowed iterations (#4909).

35) Fixed a bug where regional multipliers (MULTREGT) were applied twice to non-neighbor connections entered in EDITNNC (#4921).

36) Fixed a bug where if *--enable-vtk-output=true* and *--enable-write-all-solutions=false* (it is false by default), then Visualization Toolkit (VTK) files were written for all solutions. The code has been modified so that VTK files are only written for report steps (unless *–enable-write-all-solutions=true*) (#4916).

37) After DUNE version 2.7, the method *getCollectiveCommunication()* was changed to *getCommunication()*. The *getCollectiveCommunication()* method issues the deprecation warning and assumes all process take part in the communication. However, this might not be the case, namely, only a subset might be used. This fix consists of getting the communication object used by the grid as this uses only processes that are actually computing and not only outputting (#4962).

38) Fixed computation of dissolved gas in water ratio (Rsw) used in calculation of wellbore density for standard wells (#4976).

B.1.5 KNOWN ISSUES

1) There is an error with the RSM header for summary vectors whose NUMS entry in the SMSPEC file is derived from more than a single number source (e.g., single region or segment ID). This applies to all block vectors (BGPV, BOPV, BWPV, etc.), connection level quantities (COPT, etc.), and inter-region flows such as ROFT etc (#3078). The work around is to plot the data in OPM ResInsight and right-click on the plot to view and copy the data.

2) OPM Flow does not support using LIQ as a well's preferred phase with the WELSPECS. keyword, that is WELSPECS(TYPE) equals LIQ. This is a long-standing bug/omission in the simulator stemming from a somewhat naive internal notion of phases so we don't have an entry for a liquid phase, only for the distinct oil and water phases. For producing wells this mostly matters if you plot the WPI summary vector (productivity index for well's preferred phase). In the current treatment WPI will not have contributions from the water phase if the declared preferred phase is LIQ. For injecting wells WELSPECS's preferred phase doesn't really matter at all since the preferred phase is (typically) reset to the injected phase in WCONINJE/WCONINJH anyway. See issue #3075.

3) If the simulator finds well connection being declared as connections via the COMPDAT keyword in the SCHEDULE section, then it writes out a warning message:

```
Warning: Problem with COMPDAT keyword
In SIM_PEE_Basic_Simulator.DATA line 535
The cell (0,6,4) in well INJ1 is not active and the connection will be ignored

Warning: Problem with COMPDAT keyword
In SIM_PEE_Basic_Simulator.DATA line 535
The cell (16,0,4) in well INJ2 is not active and the connection will be ignored
```

However, the reported cell references are offset by minus one, meaning the correct warning messages should be:

```
Warning: Problem with COMPDAT keyword
In SIM_PEE_Basic_Simulator.DATA line 535
The cell (1,7,5) in well INJ1 is not active and the connection will be ignored

Warning: Problem with COMPDAT keyword
In SIM_PEE_Basic_Simulator.DATA line 535
The cell (17,1,5) in well INJ2 is not active and the connection will be ignored
```

See issue #3167 for details.

4) The GDFILE keyword in the GRID section loads a grid file in various formats, with the FMTOPT parameter setting the format type of the file. If the variable FMTOPT is omitted then the default is for binary file input for the commercial simulator; whereas, OPM Flow derives FMTOPT from the file extension (*.EGRID or *.FEGRID), making FMTOPT superfluous. However, if the extension is lower case then OPM Flow may incorrectly determine the file type. The work around is change the extension to upper case.

5) As per previous releases of the radial model, the COORDSYS keyword item three must be set to COMP to complete the circle, this has not been implemented in this release. Also there appears to be a bug for full radial models when a well goes on BHP control that causes the well not to respect the BHP constraint, this eventually causes the well to die prematurely. See #2640 for a discussion on the topic.

6) As in previous releases there are some issues with the OPERATE and OPERATER keywords associated with the input parsing; for various reasons a few of the fields require special case treatment in the grid processing, including (at least) MULTZ, PORV and ACTNUM, and for those keywords the OPERATE/OPERATER keyword doe not work. The work around is to use the MULTIPLY keyword instead.

7) For the UDQ ASSIGN operator after the terminating "/" normally any comments can be entered; however, if there is "/" within the comment field, as per:

```
ASSIGN FUNGLYLD 1.196   /    Condensate Yield (63.5/56.7)/(1.0 – 0.065)
```

then the simulator will abort. The work around is to manually place the comment characters "--" after the ASSIGN terminating "/", like so:

```
ASSIGN FUNGLYLD 1.196   / -- Condensate Yield (63.5/56.7)/(1.0 – 0.06)
```

8) At the moment, one cannot initialize tracers using the EQUALS keyword. Instead use the array format, that is the keyword followed by the required number of values, or the TVDP keyword in the SOLUTION section to set the initial tracer concentrations as a function of depth.

9) Currently, gas tracers cannot be used if the dissolved gas phase, as per the DISGAS keyword in the RUNSPEC section, is active in the model.

10) The summary vector RTIPTHEA, that defines the energy in-place between the initial and the current time for regions, is not supported unlike the FTIPTHEA and BTIPTHEA vectors. Secondly, the error message:

```
Warning: Problem with summary keyword RTIPTHEA
In RSM-THERMAL.data line 492
FIP region FIPHEA not defined in REGIONS section - RTIPTHEA ignored
```

is incorrect, as the message indicates that it is being treated like a named region, as per the FIP keyword, when it is actually a SUMMARY vector (#3870).

11) If there are cells that are very distorted, which can occur near fault planes, then the simulator may abort because it cannot calculate the pore volume of such cells. The work around is to re-generate

the grid in the static model, taking care that the cells around the fault planes are more or less orthogonal (*#2992* and *#3770*).

12) Currently the OPERATER keyword in the EDIT section does not work with the DEPTH, TRANX, TRANY and TRANZ property arrays (*#2994* and *#748*).

13) If a standard well is fully declared in an ACTIONX block which is then activated at a later date, and later the well is modified to be a multi-segment well using the WELSEGS and COMPSEGS keywords, then this will cause the simulator to abort with an assert failure. The solution to this issue is to not use this type of work flow in declaring wells (*#2891* and *#2895*).

14) Although the ACTIONX EXIT command works as expected, it does not write out the requested RSM file at the end of the run. However, the other SUMMARY and RESTART files are written out (*#2877*).

15) Although the GCONSUMP keyword in the SCHEDULE section is fully implemented as documented, it is not possible to verify the output as the associated SUMMARY vectors are not written out, that is the SUMMARY sales gas vectors FGSR, FGST, GGSR and GGST, and fuel vectors FGCR, FGCT, GGCR, and GGCT have not been implemented (*#2679*).

16) There are small differences in the behavior of the NEXTSTEP keyword in the RUNSPEC section between OPM Flow and the commercial simulator that remain unresolved (*#3745*).

17) **There is a unit handling issue associated with OPERATE keyword. If the OPERATE(X) parameter has units, as for example PERMX, then the conversion is always done in SI units, despite the input deck declaring the deck to be fields units, as per FIELD keyword in the RUNSPEC section. Note that OPM Flow performs all of its calculations internally in SI and performs unit handling only when inputting the *.DATA file and when outputting result files. Thanks to *lrijkels* for reporting the issue. See *#4597* for details.**

18) There is an issue associated with restarting from a restart file with the solution gas (Rs) maximum rate of increase, as defined by the DRSDT keyword in the SCHEDULE, that has been set to zero. This is because, the simulator does not save/restore this setting in simulator's restart files, which means that simulator misses the essential value zero upon restarting the case. As a work-around one can use the option --sched-restart=true, when running the restart case. This will initialize the restarted simulation based on information from the complete SCHEDULE section, instead of just the parts that we're going to simulate and the rest from the restart file. Thanks to *goncalvesmachadoc* for reporting the issue. See *#4272* for details.

19) OPM cannot be built with dune-fem version 2.9 or later. Please use a previous version.

20) The simulator uses an irregular corner-point grid geometry with adjusted pore volumes to represent radial grids so it is not possible to create a full ring (360 degree disk) with only one cell in the theta direction (NY=1). The work around is to model a slice (say DTHETA=60 degrees). Note that as the angle increases larger pore volume adjustments are required (*#4755*).

21) In principle the PYACTION code can control anything, however the simulator generally deals poorly with wells not explicitly controlled by the input file (for example opening or closing wells). It is recommended to utilize the normal ACTIONX machinery for well controls by constructing an ACTIONX block in the Python code and then passing that back to the Schedule object (*#4810* and *#4813*).

22) If the *--save-step* command line option is used to request the serialized state is saved to an .OPMRST file and HDF5 support is missing then an error is generated at the report step where the save was requested. This error should be generated during startup to save time and resources (*#4812*).

23) In some cases with the network option the simulator can wrongly report that a well has no THP constraints, for example

```
GLIFT WTEST: Well S-3H does not have THP constraints
```

when THP constraints have been defined (*#4887*).

B.2 RELEASE 2023-04

This release is dedicated to the memory of Ove Sævareid, who regrettably passed away suddenly this April. Ove was a long-standing contributor to OPM, and active up to the last. We are thankful for his contributions to the OPM community. His enthusiasm, know-how and scientific expertise will be missed by colleagues and friends.

A major change for this release is the validation process, that now is more strict, in that all keywords that would effect the simulation results will now cause an error message resulting with the simulator terminating with an error. Previously, the simulator would issue a warning message instead, thus allowing the simulator to continue. A new command line parameter, *--parsing-strictness*, has been implemented to give better flexibility over the validation process. To restore the previous behavior, use *--parsing-strictness="low"*. This will allow unsupported keywords that would change the simulator results if supported, and even if marked critical, for the simulator to continue.

Note that Dune version 2.7 is now required.

The 2023-04 release consists of some new features and various improvements and bug fixes. Highlights for this release include:

- For gas-water systems, the simulator now supports Equilibrium Initialization using the EQUIL and associated keywords in the SOLUTION section.
- Added support to specify diffusion coefficients for CO2STORE cases using the DIFFCGAS and DIFFCWAT keywords for gas-water cases, thus allowing the CO2STORE option to be used with gas-water systems. Using CO2STORE for gas-oil systems, one still needs to use the DIFFC keyword.
- The simulator now supports the EDITNNCR keyword in the EDIT section.
- For two-phase gas-water systems, the hysteresis option is now supported.
- Added support for the commercial compositional simulator keywords GSF, that define the gas relative permeability and gas-water capillary pressure data as a function of gas saturation, and WSF that defines the water relative permeability as function of water saturation.
- Implemented support for the AQUFLUX keyword both in the SOLUTION and SCHEDULE sections.
- Added grid independent well specification using well trajectory data. The WELTRAJ keyword in the SCHEDULE section defines the well path and the COMPTRAJ keyword in the SCHEDULE section defines the well trajectory connections to the grid, that is the perforations intervals.
- The WRFTPLT keyword in the SCHEDULE section has been extended to include both Production Logging Tool ("PLT") data as well as multi-segment well data.
- Added support for writing out inter-block flows, including non-neighbor connection flows using the RPTRST(FLOWS) parameter, and inter-block flows at reservoir conditions using the RPTRST(FLORES) parameter, in both the SOLUTION and SCHEDULE sections.

For the 2023.04 release the opm-material module has been folded and merged into opm-common, as show in Figure B.1 This further simplifies building the simulator from source.

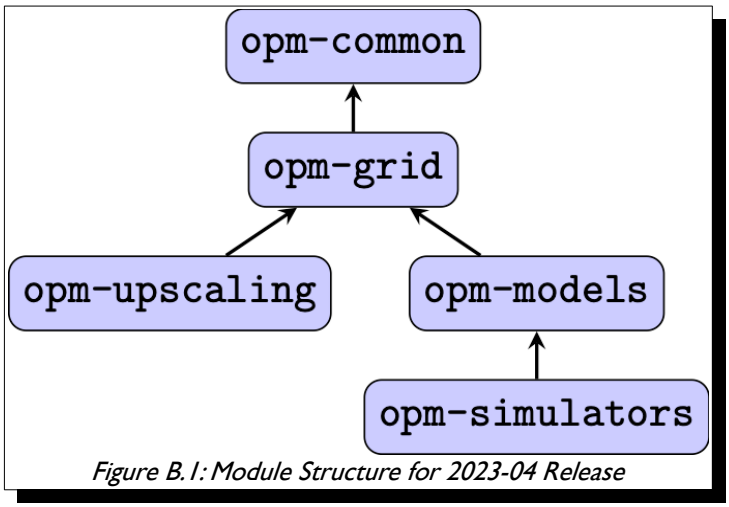


Figure B.1: Module Structure for 2023-04 Release

B.2.1 NEW AND DEPRECATED COMMAND LINE OPTIONS

The major command line changes made for this release are summarized in Table B.3

OPM Flow 2023-04 New And Deprecated Command Line Options			
No.	**Variable Name**	**Description**	**Default**
1	--cpr-reuse-interval	A positive integer that sets the reuse preconditioner interval. Used when *--cpr-reuse-setup* is set to 4, then the preconditioner will be fully recreated instead of reused every N linear solve, where N is this parameter. The default has been increased from 10 to 30 (#4338).	30
2	--cpr-reuse-setup	A positive integer that defines if the CPR solver should re-use the AMG setup. Valid options are: 0 : Recreate the preconditioner for every linear solve. 1 : Recreate once every time step. 2 : Recreate if last linear solve took more than 10 iterations. 3 : Never recreate. 4 : Recreate every N linear solves, where N is the parameter *--cpr-reuse-interval*. Changed the default value from three to four (#4338).	4
3	--linear-solver-reduction	A real positive double precision value that sets the minimum reduction of the residual for the linear solver. The linear solver convergences when the residual is reduced sufficiently. The simulator now overrides the default reduction to be 0.005 instead of 0.01, if the linear solver has been set to one of the cpr options, in a similar manner as how the default maximum number of linear iterations for the cpr and cprw options are changed to 20 instead of 100, unless specified by the command line option by the user (#4338).	0.01

OPM Flow 2023-04 New And Deprecated Command Line Options			
No.	**Variable Name**	**Description**	**Default**
4	--linear-solver	A defined quoted character string that sets the configuration of the linear solver, valid values are: 1) ilu0 (default), 2) cpr (an alias for cprw)[343] [344] and [345] 3) cpr_quasiimpes, 4) cpr_trueimpes, 5) cprw, 6) amg,[346] or 7) a file name that has the extension ".json", that contains the linear solver configuration parameters. Option (5) extends the existing Constrained Pressure Residual ("CPR") preconditioner to include wells. This option can also be invoked via the CPR keyword in the RUNSPEC section; however, the command line parameter takes precedence. For option (6) one enters a character string enclosed in quotes that defines the filename of a JSON configuration file for the flexible linear solver system. In this case the file extension should be '.json'. Note that the *.PRT file contains the "Property tree for the linear solver" listing, which is the JSON specification of the current case, and can be used to configure a user specific linear solver JSON file. The default is "ilu0". **The option "cpr" now is an alias for "cprw" instead of "cpr_trueimpes".**	"ilu0"

[343] *Wallis, J. R., Little, T. E., and Nolen, J. S.: "Constrained Residual Acceleration of Conjugate Residual Methods," paper SPE 13536 presented at the SPE Reservoir Simulation Symposium, Dallas, Texas, USA (February 10-13, 1985).*

[344] *R. Scheichl, M. Roland, J. Wendebourg, Decoupling and block preconditioning for sedimentary basin simulations, Computational Geosciences 7 (2003) 295{318.*

[345] *Klemetsdal, Ø.S., Møyner, O. & Lie, KA. Accelerating multiscale simulation of complex geomodels by use of dynamically adapted basis functions. Comput Geosci 24, 459–476 (2020). https://doi.org/10.1007/s10596-019-9827-z.*

[346] *M. Blatt, A parallel algebraic multigrid method for elliptic problems with highly discontinuous coefficients, Ph.D. thesis, Ruprecht-Karls-Universität Heidelberg (2010).*

OPM Flow 2023-04 New And Deprecated Command Line Options			
No.	**Variable Name**	**Description**	**Default**
5	--load-step	An integer value that determines if the simulator should load the serialized state from OPM Flow's version of the restart file. And should be set to either a specific report step, or 0 to load the last stored report step. The default value of -1 does not load the data from the OPM Flow specific restart file. OPM Flow's version of the restart file, is written using the *--save-step=N* option.	-1
6	--opencl-ilu-parallel	A Boolean value set to true or false that if set to true then parallelize the ILU decomposition and application on GPU, or not (false).	WAHR
7	--output-extra-convergence-info	Provides additional convergence output to separate files for diagnostic purposes. The available options are: 1) "none" results in no extra output and overrides all other options. 2) "steps" writes out convergence information per time step, to a CASENAME.INFOSTEP file. The file is useful for identifying numerical issues. 3) "iterations" writes out non-linear convergence metrics, i.e., the MB and CNV values, per phase, for each non-linear iteration in each time step, to a CASENAME.*INFOITER* file. Options may be combined with commas, e.g."steps,iterations" for multiple outputs. The default value of "none" prevents the two files from being written out, for better compatibility with the commercial simulator (#4322).	"none"

OPM Flow 2023-04 New And Deprecated Command Line Options			
No.	**Variable Name**	**Description**	**Default**
8	--parsing-strictness	Set strictness of parsing process. Available options are: 1) "normal": stop for critical errors, and for unsupported keywords that would would change the simulator results if supported. 2) "high": stop for all errors, that is even for unsupported keywords that do not effect the results, for example ECHO and NOECHO. 3) "low": same as normal, except do not stop due to unsupported keywords that would change the simulator results if supported, and even if marked critical. Default: "normal" See #4607 and #4604 for details.	"normal"
9	--relaxed-linear-solver-reduction	A real positive value that defines the minimum reduction of the residual which the linear solver need to achieve for the solution to be accepted.	0.01
10	--save-step	A character string that determines if the simulator should save the serialized state of the OPM Flow simulator at one or more report steps to a special *.OPMRST file. This is in addition to the normal restart files written, and consumes significantly more space than the normal restart files, but restarting OPM Flow from this file using the *--load-step* option deviates less from the original run, compared to restarting using the normal restart features. The files produced are not compatible with other simulators, and also will not be compatible between different releases of OPM Flow The parameter should be set to one of the following: 1) "all" to save all report steps, 2) ":x" to save every x'th step, or 3) "x" to save a specif time step. The default value of "" does not write anything to the OPM Flow specific restart file.	""
11	--cpr-max-ell-iter	A positive integer that sets the maximum number of Iterations for the elliptic pressure part of the Constrained Pressure Residual ("CPR") solver. Depreciated (#4338).	20

OPM Flow 2023-04 New And Deprecated Command Line Options			
No.	**Variable Name**	**Description**	**Default**
12	--ecl-strict-parsing	A Boolean value set to true or false that switches on (true) or off (false) strict parsing mode for parsing - all errors are collected before the application exists. Useful for debugging initial runs. Depreciated (#4607 and #4604).	FALSCH
13	--separate-sparse-source-terms	Note that the *--separate-sparse-source-terms* is only available when running the *--flow_blackoil* simulator, not the main simulator, flow. It is therefore not possible to change it when running *flow*, and the default will always apply (the default is to keep existing behavior). This is because the feature is experimental, but may be incorporated at a later day into *flow* after sufficient testing. To use the experimental *--flow_blackoil simulator* one must build OPM from source (#776 and #4347).	

Notes:

1) Cells colored in green in the "No." column indicate a new command line option or a change to an existing option for this release.
2) Cells colored in gray in the "No." column indicate a new command line option or a change to an existing option for this release, but is only available for the flow_black-oil simulator, that is it is not available for the main simulator, flow.
3) Cells colored in orange in the "No." column indicate the command line option is available but is now "hidden" from the main help listing, --help. These options can be listed using the --help-all command line option, that lists all the command line options included in the release, including experimental, obsolete, hidden and deprecated options.
4) Cells colored in red in the "No." column indicate a deprecated command line option for this release.

Table B.3: OPM Flow 2023-04 New and Deprecated Command Line Options

B.2.2 NEW FEATURES

In addition to the above the following new features have been added to the simulator:

1) Implemented support for the AQUFLUX keyword both in the SOLUTION and SCHEDULE sections (#3378, #4429 and #4480).

2) Added support for dissolved gas in water with OPM Flow specific keyword DISGASW in the RUNSPEC section, for use with the CO2STORE model and gas-water systems. The formulation incorporates diffusion. However, currently only standard wells are supported, but multi-segment wells support is ongoing, as well as implementation of the RSW keyword in the SOLUTION section to define the initial gas solubility in water ratio (#3217, #4292, #763 and #3246).

3) Added support for vaporized water to the gas phase in the CO2STORE- BRINE model. The PVT properties are assumed to be independent on the water content (#3404).

4) The Thermal model can now be used with gas-water systems, including with the CO2STORE model, to support temperature dependent properties. Currently, this is implemented as a standalone simulator, but will eventually merged into OPM Flow (#4482).

5) Added parser support for the following keywords: GASWAT, COMPS, HWELLS, SOLID in the RUNSPEC section, CNAMES, NCOMPS and ZMFVD in the PROPS section, FIELDSEP in the SOLUTION section, and WELLSTRE and WINJGAS in the SCHEDULE section (#3442).

6) Implemented support for the CPR keyword in the RUNSPEC section, that invokes the Constrained Pressure Residual linear solver. When used in the RUNSPEC section the keyword is equivalent to using the command line parameter *--linear-solver= "cprw"*. Note that if the command line *--linear-solver* parameter has been used, then this will take precedence over the CPR keyword in the RUNSPEC section (#4400).

7) Added support for using the Damaris library (#4309, #4432, #4414, and #4481). See also #4399 for a bug fix.

   Damaris is a middle-ware for asynchronous I/O and data management targeting large-scale, MPI-based High Performance Computer ("HPC") simulations. Initially the software used dedicate cores for asynchronous I/O in multi-core nodes of HPC platforms, with an emphasis on ease of integration in existing simulations, efficient resource usage (with the use of shared memory) and simplicity of extension through plug-ins. Over the years, the software has evolved into a more elaborate system, providing the possibility to use dedicated cores or dedicated nodes to carry out in situ data processing and visualization. It proposes a seamless connection to the *VisIt visualization framework* to enable in situ visualization with minimum impact on run time. Damaris provides a simple API and can be easily integrated into the existing large-scale simulations (*Damaris*).

8) Added support to specify diffusion coefficients for CO2STORE cases using the DIFFCGAS and DIFFCWAT keywords for gas-water cases. Using CO2STORE for gas-oil systems, one still needs to use the DIFFC keyword. Note that if neither the DIFFCGAS and DIFFCWAT keywords are declared when the DIFFUSE keyword is active, then the simulator will use default values computed based on temperature and pressure in the reservoir using correlations in the literature (#3455).

9) Added ISTL linear solver support for the Dune ALUGrid module, that provides an adaptive, load balancing, and unstructured implementation of the DUNE grid interface in two or three space dimensions supporting either simplices (triangle or tetrahedron) or cube elements (#4269).

10) The simulator now supports the EDITNNCR keyword in the EDIT section (#4600, #4578 and #3459).

11) Added the flexiblegmres linear solver, similar to bicgstab (the default linear solver), and gmres that were already available. The flexiblegmres solver is a variant of the gmres algorithm but allows changes in the preconditioning at every step in the iterative process (#4564).

12) Added Energy support for two-phase gas-water systems (#4574).

13) Previously for gas-water systems, only Enumeration Initialization was available. The simulator now supports Equilibrium Initialization using the EQUIL and associated keywords in the SOLUTION section (#4316). See also #4372 that changed the reported pressures for gas-water systems to the gas phase pressures, in order to be compatible with the commercial simulator.

14) For two-phase gas-water systems, the hysteresis option is now supported (#3451).

15) Added grid independent well specification using well trajectory data. The WELTRAJ keyword in the SCHEDULE section defines the well path and the COMPTRAJ keyword in the SCHEDULE section defines the well trajectory connections to the grid, that is the perforations intervals. WELTRAJ defined wells must use the COMPTRAJ keyword to define the connections to the grid, that is one cannot use COMPDAT for these type of wells. Secondly, the WELSPECS(I) and the WELSPECS(J) parameters must be defaulted with 1*, for these type of wells (#3384 and #3466).

16) Added support for the commercial compositional simulator keywords GSF, that define the gas relative permeability and gas-water capillary pressure data as a function of gas saturation, and WSF that defines the water relative permeability as function of water saturation. The keywords should only be used if only the gas and water phases are present in the run, and can therefore also be used with the CO2STORE model (#3406 and #4485).

17) Added support for directional relative permeabilities with hysteresis, using the IMBNUMX, IMBNUMY, and IMBNUMZ keywords in the REGION section. Note that the SATOPTS(IRREVERS) parameter in the RUNSPEC section, that sets the relative permeability assignment to non-reversible, and results in different sets of relative permeability tables being applied for flow from the xi to xi_{+1} direction and the xi to the xi_{-1} direction, for all directions (x, y, z), is currently not supported. To be clear, the IMBNUMX-, IMBNUMY-, and IMBNUMZ- keywords are currently not supported (#3364 and #4384).

18) Added support for the GPU rocsparseSolver, the feature is experimental and will require a certain kind of GPU, not a usual consumer one. Use the *--help-all* command line parameter to see the available options (#4178 and #4406).

19) The OPM Flow specific keyword RWGSALT has now been implemented. The keyword defines the relationship of water vaporization versus pressure and salt concentration, for when the VAPWAT keyword has be declared in the RUNSPEC section indicating that vaporized water is present in the gas phase. In addition, RWGSALT may also be used if the Salt Precipitation model has been activated via the BRINE and PRECSALT keywords in the RUNSPEC section (#4189, #3187, #744 and #538).

20) Added support and implementation of salt density on the OPM Flow SALTSOL keyword in the PROPS section. The keyword originally defined a grid block's maximum salt solubility for each PVTNUM region, and now supports in addition the salt density for the grid blocks (#745 and #2981).

21) Implemented support for the CVIR, well connection reservoir volume injection rate, and CVPR, well connection reservoir volume production rate (#4370).

22) Added various variables for OPM Flow's black-oil CO2STORE Model, that are based on the commercial simulator's compositional vectors, as the CO2STORE model is only available in the commercial simulator's compositional simulator (#3408 and #4477).

23) Added multi-segment well segment vaporized oil rate (SOFRF) and dissolved gas rate (SGFRF) SUMMARY vectors (#4252 and #3215).

24) Added vaporized water (VAPWAT) to two-phase Gas-Water formulations, together with exporting the vapor phase mole factions to the restart file (YMFWAT). In addition, use the correct molar mass for liquid phase mole fractions of CO_2 (that is, the mole factions of CO_2 in the water phase) when converting and writing out liquid phase mole fractions (XMFCO2) to the restart file (#4537).

25) Previously, the WRFTPLT keyword in the SCHEDULE section only wrote the standard RFT data set (pressure and fluid saturations versus depth) for the connected grid blocks, the same as the WRFT keyword in the SCHEDULE section. The keyword has now be extended to include both Production Logging Tool ("PLT") data as well as multi-segment well data (#3151, #3173, #3179, #3184, #3145, #3155, #4261, #4168 and #4180).

B.2.3 IMPROVEMENTS

Improvements include:

1) The OPM Flow BC keyword that sets the model's boundary conditions in the SOLUTION section, has been extended to include DIRICHLET boundary conditions, to enable user defined boundary conditions to be defined. In addition, the BC(PRESS) and BC(TEMP) parameters have been added to allow the simulator's equilibrium calculated values at the boundary to be used, if these two parameters are defaulted. Note that BC keyword syntax and structure is likely to changed in future releases of the simulator (*#739*, *#3185*, *#3212* and *#4182*).

2) The OPM Flow BC keyword using the DIRICHLET boundary conditions has been extended with the THERMAL option. The temperature is kept constant at the given boundary by allowing for heat conduction only. In other words no fluid flow only heat (*#3457* and *#4562*).

3) The OPM convertECL utility program converts various commercial simulator and OPM Flow generated output files to various other formats, as well as being able to list and extract data from user supplied time steps. The utility has now been extended with -g option, that will output GRDECL files[347]. In this case the input file should be formatted or unformatted simulation files. The -o option can be used to specify output file name and is only valid with -g option. The program requires the input name of file to be converted, if this is a binary file the output is a formatted, and vice versa. The current options include:

```
-h Print help and exit.
-l List report step numbers in the selected restart file.
-g Convert file to grdecl format.
-o Specify output file name (only valid with grdecl option).
-i Enforce IX standard on output file.
-r Extract and convert a specific report time step number from a unified restart
file.
```

   Use the -help option to get further details on the program's features (*#3201*).

4) Improved the Distributed Well code base by supporting variable length connection data for this type of formulation (*#4201* and *#599*).

5) Dune version 2.7 is now required (*#376*, *#631*, *#788*, *#3414*, and *#4490*).

6) Previously, when increasing and allocating the artificial lift quantity (ALQ), if a given ALQ value resulted in a well's rate being invalid, then the simulator stopped allocating additional values of ALQ. Now the simulator continues to increase and allocate ALQ values with the intention that higher ALQ values may allow the well to flow again. At the same time the simulator checks that rate limits are not violated (*#4571*).

7) Improved gas lift optimization allocation for wells under gas lift and group control (*#4214*).

8) OPM Flow now applies the grid unit transformation when writing out the EGRID file (*#3233*).

9) Improved the methodology for the ILU preconditioner by only copying the data, and avoiding the need for allocation and matrix construction, when the matrix already exists and has the correct sparsity structure. Expected performance gain for the linear solver setup part for ILU0 is estimated be approximately 15% based on the Norne model using eight processors (*#4427*).

10) Improved the handling on how the simulator reads left-handed grids, that fixes some of the problems previously experienced with with this type of grid. There is no guarantee that this improvement will solve this particular problem, however; but if a user previously experienced this problem, then they should re-test the model with this release. If this release does not fix the issue then please report this via GitHub, together with an example model showing the failure (*#625*).

[347] *GRDECL files is the extension commonly used for exporting the commercial simulator array data in fixed format by earth modeling software. These type of files include the keyword followed by the data, and can be directly loaded into both OPM Flow and the commercial simulator.*

11) The OPM Flow specific keyword MINPVFIL has now be depreciated, the keyword has not been documented in the manual, and has not been used for a long time (*#3461*).

12) Improved the error reporting for when the fault name was not found in the MULTFLT keyword, that is MULTFLT(FLTNAME) contains a fault that has not be defined by the FAULTS keyword. Previously, the error message was:

```
Applying MULTFLT in MULTFLT.DATA line 237

Error: [./src/opm/....../EclipseState/EclipseState.cpp:380] Could not set fault
transmissibility multiplier 0 for fault FAULT01: Key FAULT01 not found.

Error:
An error occurred while creating the reservoir properties
Internal error: [./src/opm/input/eclipse/EclipseState/EclipseState.cpp:380] Could
not set fault transmissibility multiplier 0 for fault FAULT01: Key FAULT01 not
found.

Error: Unrecoverable errors while loading input:
[./src/opm/input/eclipse/EclipseState/EclipseState.cpp:380] Could not set fault
transmissibility multiplier 0 for fault FAULT01: Key FAULT01 not found.
```

And now the new error message is:

```
Applying MULTFLT in MULTFLT.DATA line 237

Error:
MULTFLT: Cannot set fault transmissibility multiplier
MULTFLT(FLTNAME) equals FAULT01 and MULT(FLT-TRS) equals 0
Error creating reservoir properties: Key FAULT01 not found.

Error:
An error occurred while creating the reservoir properties
Internal error: Error Processing MULTFLT
```

See *#3077* and *#3062* for details.

13) Added additional time step convergence reporting of MB and CNV quantities per iteration, intended for diagnostic and analysis purposes. The data will not be written out to the CASENAME.INFOITER file unless specifically requested via the --output-extra-convergence-info command line parameter.

ReportStep	TimeStep	Time	Iteration	MB_Oil	CNV_Oil	MB_Water	CNV_Water	MB_Gas	CNV_Gas	WellStatus
82	0	9.1200e+02	0	2.3292e-05	4.5453e-02	1.1772e-05	2.5904e+01	9.7696e-06	1.9537e-02	CONV
82	0	9.1200e+02	1	1.5357e-07	2.4792e+00	3.6332e-06	4.1195e+00	1.8721e-07	2.9106e+01	CONV
82	0	9.1200e+02	2	3.8775e-07	6.7989e-01	3.1721e-06	1.1337e+00	3.2625e-07	3.9179e+00	CONV
82	0	9.1200e+02	3	2.3706e-07	2.1720e-01	3.3442e-06	9.9475e-01	1.8359e-07	2.7495e-01	CONV
82	0	9.1200e+02	4	8.0505e-08	1.3516e+00	3.2876e-06	7.8305e-02	1.0693e-07	4.7519e-01	CONV
82	0	9.1200e+02	5	2.3315e-08	9.0378e-03	2.7611e-06	8.3702e+00	3.4747e-08	2.3653e-02	CONV
82	0	9.1200e+02	6	1.8666e-08	3.0498e-03	2.6078e-06	7.1568e-01	2.5702e-08	1.0469e-03	CONV
82	0	9.1200e+02	7	8.2979e-09	1.2817e-03	9.1366e-07	6.9700e-02	4.8841e-09	7.1927e-04	CONV
83	0	9.1300e+02	0	3.2710e-05	2.3169e-01	1.6583e-05	2.4644e-01	1.3703e-05	2.4830e-01	CONV
[...]										
83	0	9.1300e+02	1	2.2321e-07	6.9474e-02	5.0324e-06	1.5035e-01	2.1392e-07	6.8448e-02	CONV
83	0	9.1300e+02	2	7.9707e-08	1.7758e-02	4.5233e-06	1.4823e+01	1.8688e-08	8.4370e-02	CONV
83	0	9.1300e+02	3	7.1908e-08	1.0376e-03	4.4680e-06	1.6808e+00	1.2909e-08	2.3520e-03	CONV
83	0	9.1300e+02	4	1.5509e-08	6.7821e-05	2.0725e-06	3.0567e-02	7.7076e-09	5.2698e-04	CONV
83	0	9.1300e+02	5	1.4586e-11	1.4220e-05	2.3775e-07	4.4014e-05	2.7961e-10	6.3101e-06	CONV
[...]										
[...]										
120	1	2.5105e+03	4	1.9859e-08	6.5491e-04	8.7468e-06	1.0563e-03	5.3570e-08	7.6632e-04	CONV
120	1	2.5105e+03	5	1.7167e-10	5.7449e-03	1.3679e-07	5.5678e-04	2.8410e-09	4.9974e-03	CONV
121	0	2.5560e+03	0	4.4468e-06	1.1437e-03	1.6915e-06	1.3708e-03	2.9207e-06	1.3201e-03	CONV
121	0	2.5560e+03	1	1.1517e-09	5.9272e-06	4.1542e-07	4.1135e-05	1.3881e-09	3.1044e-06	CONV
121	0	2.5560e+03	2	5.0131e-12	1.3745e-08	3.7373e-08	4.7599e-06	8.0810e-12	1.8838e-08	CONV

Figure B.2: INFOITER Convergence Report Example

See *#4312*, *#4311*, *#4322*, and *#4323* for further details.

14) Improved numerical performance of the tpfalinearizer routine by using direct access to matrix elements in the routine, that is, instead of using matrix[row, col] to access the matrix block to be

modified, the code now stores the pointers to the relevant block. Thus, avoiding a search on the relevant row to find the correct block matching the requested column number. The change gives a speedup of approximately 10% for this part off assembly code. This, together with not using local well assembly, will further increase numerical performance (#770).

15) Recent testing on a large full field ensemble model has led to improvements in the default CPR parameters used by the simulator, including, combining a number of minor adjustments for simpler and more consistent and robust use of the CPR options. Changes include:

    a) Removed two unused command line parameters, --cpr-ell-solvetype and --cpr-max-ell. The former was never used and the latter was not used correctly, as it should should have changed the maximum iterations of the coarse solver, not the repeats of the preconditioner, and should now be used instead with the JSON file input.

    b) Made the default AMG parameter setup for the *cpr* (including *cpr_trueimpes* and *cpr_quasiimpes*) option of the command line parameter --linear-solver, match the setup for *cprw*. In particular, beta = 0.0 (not 1e-4) and prolongationdamping = 1.0 (not 1.6).

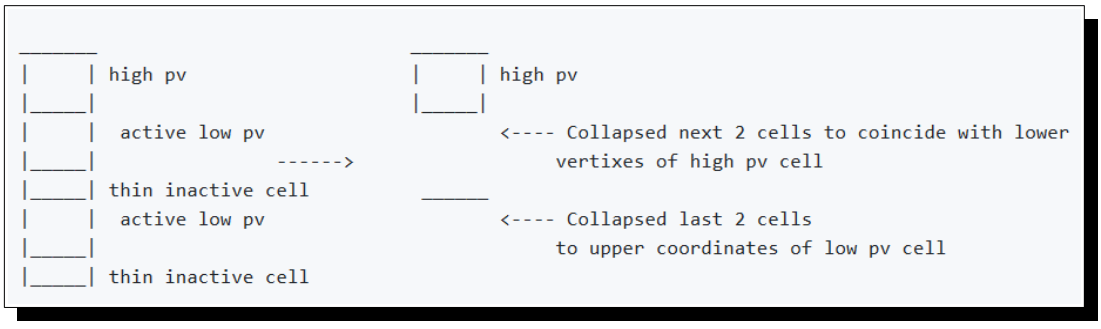
    c) The simulator now overrides the default reduction to be 0.005 instead of 0.01, if the linear solver has been set to one of the cpr options, in a similar manner as how the default maximum number of linear iterations for the cpr and cprw options are changed to 20 instead of 100, unless specified by the command line option by the user.

    d) Changed the default value for --c*pr*-r*euse*-s*etup* to 4 (was 3). The old default never recreates the preconditioner from scratch, i.e. the hierarchy that is built for AMG is kept unchanged (the numbers in the matrices and therefore also smoothers etc. are updated every solve). The new default will recreate it every --c*pr*-r*euse*-i*nterval* solves instead. The default for --c*pr*-r*euse*-i*nterval* has been increased from 10 to 30 (but recall the interval was essentially infinite when --c*pr*-r*euse*-s*etup* was 3).

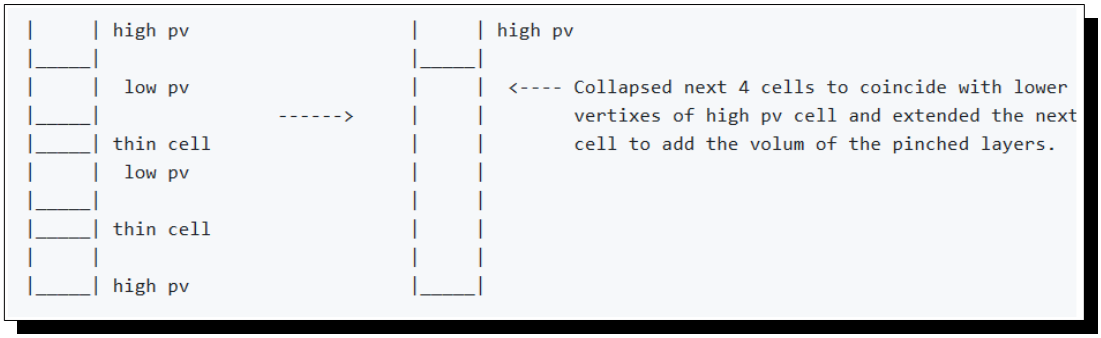
    See also B.2.1 New and Deprecated Command Line Options for details (#4338).

16) Refactored the handling of PINCH and MINPV keywords in preparation to support more of the PINCH test cases. Previously, the logic on how to detect inactive cells would not work correctly if the ACTNUM array was empty. This now been fixed. The PINCH(PINCHOPT) parameter controls the generation of pinch-outs when the MINPV keyword has been used to deactivate cells with small pore volumes. Only the default value of GAP is supported that allows for the generation of Non-Neighbor Connections (NNCs) across cells that have been made inactive with the MINPV keyword when the pinch-out threshold thickness, PINCH(PINCHTHK), is greater than the cells thickness.

    For example, given two active cells that get deactivated via the PINCH keywords, would result in:



    Another example, with an additional bottom cell at the base, would result in:



See #639 for details.

17) Improved input checking of keywords and keyword options so that in general, unsupported keywords which effect the results, will cause the simulator to issue an error message and to stop.

```
Error: Unsupported keywords or keyword items:

  SKIP: keyword not supported
  In file: TEST1.DATA, line 147

  ENDSKIP: keyword not supported
  In file: TEST1.DATA, line 149
```

However, keywords, that do not influence the result, for example: DEBUG, ECHO, NOECHO and RPTSOL etc., will result in a warning message and the simulation will continue.

```
Warning: Unsupported keywords or keyword items:

  RPTGRID: keyword not supported
  In file: TEST1.DATA, line 120

  NOECHO: keyword not supported
  In file: TEST1.DATA, line 132
```

Secondly, if an unsupported keyword parameter influences the results, for example WECON(WGR), then the simulator will now stop with an error message. Previously, the simulator would issue a warning message and continue (#4278).

18) The restart file's enumeration of active cells connected to analytic aquifers follows a column-based scheme. Recent work on compatibility prompted by real field models with the constant flux analytic aquifer type, has shown that the enumeration of the columns also must account for the horizontal model dimension. If $DIMENS(NX) \geq DIMENS(NY)$, then the columns are enumerated with the J index cycling faster than the I index. On the other hand, when $DIMENS(NY) > DIMENS(NX)$ the I index is supposed to cycle more quickly than the J index. Thus, we now guarantee that the innermost loop is always across the model layers, while ensuring that the 'outer' loop always iterates over an index range that is at least as large as that of the 'middle' loop (#3409).

19) For runs using the CO2STORE option, the simulator now unconditionally writes out the cell pressures to the restart file (#4540).

20) The simulator already supports constant flux aquifers (AQUFLUX keyword) entered both in the SOLUTION and in the SCHEDULE sections. Support is now extended to read and write the data to the restart file, compatible with the commercial simulator restart file.

    The primary structural change is that we have to support analytic aquifers being introduced dynamically, whence the connection information (xCAQ) must be captured without knowing the full

set of analytic aquifers. We therefore allocate those output arrays purely from the maximum sizes entered in the RUNSPEC section whereas we defer determining the maximum aquifer ID to the point of capturing the dynamic values (#3423, #3424 and #3437).

21) Improved RESTART file support by writing out inter-block flows, including non-neighbor connection flows using the RPTRST(FLOWS) parameter, and inter-block flows at reservoir conditions using the RPTRST(FLORES) parameter, in the SOLUTION and SCHEDULE sections (#758, #3224, and #4275). In addition, fixed a regression (issue #4428) associated with the implementation (#4430 and #781).

22) When using *--load-step* to restart from a *.OPMRST file, the simulator will now check for the *.OPMRST file to be loaded in both the input file directory and the output file directory (#4478).

23) Added support for the molar fraction of CO_2 in brine array to be written to the RESTART file. The output only occurs if CO2STORE and DISGASW keywords are present in the RUNSPEC section, and the command line parameter, -enable-opm-rst-file is set to true (#4529, #4535, #3449 and #3448).

24) Improved restart capability associated with well perforation data, by storing previous well states on the RESTART file (#4479).

25) Improved the writing out the restart file using the *--enable-opm-rst-file=true* command line option by adding additional checks to improve consistency (#4613 and #4606).

26) The base arrays written to the restart file are compatible in both name and interpretation to those emitted in the SOLUTION section from other simulators, whereas the extended arrays are specific to OPM Flow. Added a tag to the extended arrays as OPM_EXTENDED instead of AUXILIARY as the latter is deprecated and will be removed at some point in the future. Also re-tag the TEMP array as OPM_EXTENDED when not required for thermal simulations and add support for the per-phase fluid density via the RPTRST request keys (DENx, with x being O, G, or W), in addition to the collective key DEN that the simulator already supports (#4533 and #4530).

27) The rock fraction is usually taken as $1-PORO$, but if the pore volume has been adjusted using the PORV keyword, to effectively modify the porosity, $PVMULT*PORO$, then the rock volume will be incorrect. The simulator now applies the same PORV multiplier ($PVMULT$) to the rock fraction, that is, $PVMULT*(1-PORO)$. This is to avoid negative rock volume for $PVMULT*PORO>1$. See #795 for details.

28) Improved the surface rate computation by restricting the values of Rs and Rv used in the calculation, to lie in the range of zero to the maximum known values of Rs and Rv (#4335).

29) Improved the error messages for multi-column keywords (SWOF, etc.), for when a column of data must be monotonically increasing or decreasing. Previously if there was an error, then the message would be of the form:

```
Error:
An error occurred while creating the reservoir properties
Internal error: Incorrect ordering of values in column: SW

Error: Unrecoverable errors while loading input: Incorrect ordering of values in
column: SW
```

Making it difficult to isolate the cause of the error. Now the message will be of the form:

```
Error:
An error occurred while creating the reservoir properties
Internal error: Non-monotonic values in keyword SWOF, at row: 11.
SW need to be entered in strictly increasing order.

Error: Unrecoverable errors while loading input: Non-monotonic values in keyword
SWOF, at row: 11.
SW need to be entered in strictly increasing order.
```

See #3391 for details.

30) Improved reporting by printing the inter-region threshold pressures set via the THPRES keyword in the SOLUTION section. The simulator now prints the following report.

```
LIST OF ALL NON-ZERO THRESHOLD PRESSURES
----------------------------------------

    FLOW FROM REGION   TO REGION   THRESHOLD PRESSURE
    ----------------   ---------   ------------------
         1                  2         0.588031       BARSA
         1                  3         0.787619       BARSA
         1                  4         7.00083        BARSA
    ----------------   ---------   ------------------
```

See #4505 and #3422 for further information.

31) Currently, the simulator adds the source terms cell by cell, and when a perforated cell is reached, all wells and perforations are iterated over to find the correct term to add. So with K total perforations, the process takes approximately K^2 of amount of work. Using a new experimental feature, the simulator treats the well source terms separately, iterating over all wells and perforations just once, so now the process takes K amount of work. The difference for the assembly performance is modest: ~3% improvement observed with SPE9, which has a fairly high number of wells for its size. However, there are more opportunities that can improve performance, for example, using a similar approach for aquifers may give a similar boost in performance. The feature is activated by the command line parameter *--separate-sparse-source-terms* set to true (#776 and #4347).

    Note that the *--separate-sparse-source-terms* is only available when running the *flow_blackoil* simulator, not the main simulator, flow. It is therefore not possible to change it when running *flow*, and the default will always apply (the default is to keep existing behavior). This is because the feature is experimental, but may be incorporated at a later day into *flow* after sufficient testing (#776 and #4347).

32) Re-factored the friction pressure calculation for multi-segment wells to make the code more robust (#4251).

33) Support for User Defined Quantities, UDQ variables, for multi-segment well segments (SU* SUMMARY vectors) has been implemented (#3235).

34) Improved the treatment of production wells controlled by zero rate and STOPPED wells. The issue was, although the simulator calculated rates very close to zero, the water cut and GOR values where spurious and random, due to the non-zero flow rates. Now the simulator sets the rate to zero during the initialization (from well state) at the beginning of each time step iteration, as well as the update process during the Newton solution of the wells (#4531).

35) Improved the handing of production wells when switching to THP control, that should help remedy the premature shutting of the production wells under these circumstances (#4409).

36) For wells under zero rate control, including STOPPED wells, the THP is now set to zero (#4563).

37) Given the following SCHEDULE section keywords:

```
WCONPROD
  'PROD-2'   'SHUT'  'RESV'  4*  0.0 1.0 1* 0 1.0 5* /
/

DATES
 1 'FEB' 2000 /
/

WELOPEN
  'PROD-2' 'OPEN'  5* /
/

WELTARG
   'PROD-2' 'RESV'  2000.0 /
/

-- 31.000000 days from start of simulation ( 1 'JAN' 2000 )
DATES
 1 'JUN' 2000 /
/
```

Previously, the simulator would not open the well PROD-2 with the WELOPEN keyword, since the well PROD-2 disallows cross-flow and also has a zero rate constraint. And WELOPEN does not consider the later WELTARG keyword is changing the rate constraint to be non-zero. Now the simulator will allow WELOPEN to open the well without checking the rate constraint and cross-flow specification. Secondly, the behavior is similar to how the simulator treats the WCON*** keywords, in that that the simulator does not check the rate constraint and cross-flow setup; whereas, checking the rate constraint and cross-flow setup only for the WELOPEN keyword, is introducing an inconsistency (#3463).

In addition, the following new SUMMARY keywords are now recognized as described by the comments in Table B.4.

No.	Summary Keyword	Comment
1	CVIR and CVPR	Implemented support for the CVIR, well connection reservoir volume injection rate, and CVPR, well connection reservoir volume production rate (#4370).
2	FGCDI, FGCDM, FWCD, FWIPG, FWIPL, RGCDI, RGCDM, RWCD, RWIPG, and RWIPL	Added various variables for OPM Flow's black-oil CO2STORE Model, that are based on the commercial simulator's compositional vectors, as the C02STORE model is only available in the commercial simulator's compositional simulator (#3408 and #4477).
3	SGFRF and SOFRF	Added multi-segment well segment vaporized oil rate (SOFRF) and dissolved gas rate (SGFRF) SUMMARY vectors (#4252 and #3215).
4	FGIPL and RGIPL	Added gas in-place in the liquid phase, only for cases that have dissolved gas in the water phase (#4551).

Table B.4: New SUMMARY Keywords for the 2023-04 Release

B.2.4 BUG FIXES

The following bug fixes and improvements have been incorporated into this release.

1) Fixed a bug with AQUCT and AQFETP keywords that resulted in only the last entry of the keywords being active int run. Now all occurrences are taken into consideration (*#3469* and *#3468*). Currently work is on going to implement analytical aquifers in the SCHEDULE section via the AQUCT and AQUFETP keywords, until this work is complete, the simulator will issue an error message if the keywords are found in the SCHEDULE section, in order to avoid incorrect results (*#3469*, *#3468* and *#3397*).

2) In degenerate geometries, that is when a three-dimensional grid cell is essentially flat, or it may have three or two-dimensional coordinates but be essentially a line in space, then the grid cell face may reduce to a line. In this case the area of the cell is zero and dividing by the total area leads to centroids with NaN coordinates. This scenario has now been fixed by using the 'inpoint', which is typically the arithmetic average of the face's vertex coordinates (*#624*).

3) Fixed two bugs there were noticed during close inspection of the code. The first bug was associated using the mass fraction instead of the mole fraction, and the second was a sign error in the thermal Brine density calculations. Thanks to *@svenn-t* for noticing the sign bug (*#3452*).

4) Fixed a bug introduced with *#4292* for when both VAPWAT and DISGASW keywords are active in the RUNSPEC section (*#4342*).

5) Fixed an issue when the CO2STORE keyword is combined with a two phase gas-water system using Carter Tracy aquifers (*#4580*).

6) Corrected a regression with the Damaris implementation introduced with commit *#4396*, which has now been fixed with *#4399*.

7) There was an issue with writing out the summary ESMRY files when two (or more) concurrent simulations were run in the same folder. Occasionally, this caused the temporary export file (TMP_.ESMRY) to be simultaneously accessed by more than one run, which resulted in errors in writing out the file. The temporary export file name has now been updated to include both root name and seconds since epoch, to resolve the issue (*#3202*).

8) OPM Flow's support for Field-Programmable Gate Array ("FPGA")[348] devices is deprecated due to currently not functioning correctly (*#4271*).

   A FPGA is an integrated circuit that consists of internal hardware blocks with user-programmable interconnects to customize operation for a specific application. The interconnects can readily be reprogrammed, allowing an FPGA to accommodate changes to a design or even support a new application during the lifetime of the part.

9) Fixed a bug associated with gas lift optimization for parallel runs. Now group controls and constraints are fully implement in stage two optimization, and information is communicated to all nodes (*#4310*).

10) Fixed a bug in re-calculating the gas lift gradient at stage two of the gas lift calculation (*#4573*).

11) Previously, the simulator did not correctly support the GCONINJE(VREP) option, that controls the voidage replacement ratio, if the group contained multiple subgroups with both water and gas injectors. This has now been fixed by correctly calculating the group reduction rate (i.e., the total rate of wells not under group control but in the group), and also for sub-groups (*#4460*).

12) For the GCONPROD(ACTION) parameter in the SCHEDULE, only the RATE options is supported by the simulator. The simulator will now issue an error message, if GCONPROD(ACTION) is set to a value other than RATE (*#4593* and *#4595*).

[348] *A FPGA is an integrated circuit that consists of internal hardware blocks with user-programmable interconnects to customize operation for a specific application. The interconnects can readily be reprogrammed, allowing an FPGA to accommodate changes to a design or even support a new application during the lifetime of the part.*

13) Fixed an issue with the GPMAINT keyword, when GPMAINT(FIPNUM) parameter had been set to zero for setting the pressure maintenance for the whole field (#4259).

14) Previously, if the GPMAINT keyword was used without the GCONPROD and GCONINJE keywords, then the simulator would throw an exception. This has now been corrected, such that simulator will now proceed as normal if the GPMAINT keyword is active, but the GCONPROD and GCONINJE keywords are not active or present in the deck (#4359).

15) Fixed a bug with the group control output to the terminal, so that the last group control is now correctly reported (#4235).

16) Disabled HDF5 compression if the version of HDF5 is less than or equal to 1.8, as compression for these versions does not work, especially on Red Hat 7 systems (#4534).

17) Fix a regression caused by pull request #3364, associated with the JFUNC keyword, used to scale oil-water capillary pressure (#3403).

18) Some build issues on macOS have been addressed (#3385).

19) If the MINPV keyword was used without the PINCH keyword, the simulator silently assumed that the PINCH(PINCHOPT) parameter was set to "PINCH GAP", when processing MINPV. Thus, the simulator would generate Non-Neighbor Connections ("NNC"), where none should have been created. Secondly, when one cell was deactivated the simulator missed checking whether the thickness of this cell was below the threshold, as set with the PINCH(PINCHGAP) parameter, for when PINCH(PINCHOPT) is equal to "PINCH NOGAP", before creating an NCC over it. Both issues have now been addressed (#622).

20) Using the MULTFLT keyword at the start of the SCHEDULE section resulted in the simulator throwing an exception. This has now been fixed (#761).

21) Previously, only the last MULTREGP keyword in the input deck was processed, instead of all occurrences of the keyword. This has now been corrected by processing all MULTREGP keywords (#3395).

22) Previously, using the MULTZ- keyword to adjust the vertical transmissibility (TRANZ) was not honored, and was completely ignored when calculating TRANZ combined with the PINCH keyword in the GRID section. This has now been rectified such that MULTZ- now behaves the same as the MULTZ keyword (#4320).

    Note that OPM Flow does not require the GRIDOPTS(TRANMULT) parameter in the RUNSPEC section to be set to YES, in order to use this and other negative directional dependent multiplier keywords in the input deck. Whereas, the commercial simulator will terminate with an error if the keyword is present, and the GRIDOPTS(TRANMULT) parameter has not been set to YES.

23) The NODEPROP keyword in the SCHEDULE section, that defines the node properties for the Extended Network model, had an issue if the keyword was repeated for the same node, NODEPROP(NODE). If this occurred then the second, and subsequent NODEPROP keywords, would be ignored. This has now been rectified (#3377).

24) If a numeric aquifer defined by the AQUNUM keyword had no connections defined on the AQCON keyword, that connects numeric aquifers to the grid, then the simulator would throw an exception and stop. Now the simulator will issue a warning message and continue (#3196).

25) Previously, the porosity and permeability salt precipitation tables, as defined by the PERMFACT keyword in the PROPS section, were incorrectly allocated to the grid blocks using the EQLNUM array instead of the PVTNUM array. This has now been rectified (#3349).

26) If the gap created by pinched out cells is larger than the threshold, set via the PINCH(PINCHGAP) parameter, then the simulator would create Non-Neighbor Connections (NNCs). This has now been fixed by preventing of NNCs under these circumstances (#3456).

27) Previously, during processing the PINCH keyword, the simulator would compute the index of a cell in the next non-existing layer and sometimes would index cells which have ACTNUM set to zero or

zero thickness, thus creating an index out of bounds error. Secondly, when processing a cell at the bottom of the grid, then there should be no lower neighbor for a Non-Neighbor Connection. Both issues have now been addressed (#650 and #649).

28) Fixed a regression that caused the print header not to be printed in the *.PRT file (#4418).

29) Fixed an issue with dune-fem and quad precision (#3435 and #792).

30) There was an issue associated with groups and wells under RESV control for undersaturated systems, that has now been corrected, Thanks to @tskille for reporting the issue (#4549).

31) Fixed a couple of bugs associated with writing out the restart file using the *--enable-opm-rst-file=true* command line option, when running in parallel mode (#4602 and #4601).

32) When running in parallel mode the region threshold pressures from all processes were not fully written out to the restart file, this has now been corrected (#4503).

33) Fixed a bug associated with the ROCK keyword in the PROPS section, not using the default PVTNUM region, when either ROCKOPTS keyword is absent from the deck, or when ROCKOPTS(ROCKOPT3) has been defaulted with 1* (#3428).

34) A minor bug with the SUMMARY utility program associated with opening the *.SMSPEC file has been fixed. The program will now print a warning when it fails to open the file, and will then continue processing the next summary file (#3319).

35) The well flux for injection perforations depends on the connected grid cell mobility in the cell, and not the upstream grid cell value. In thermal runs, this can lead to numerical problems if the mobility strongly depends on temperature, as reported in issue #4244. The solution was to remove the derivative for thermal flux for injector perforations in the Solver. Note that this fix will also affect isothermal cases as well; however, testing indicates that this only has a minor negative impact on convergence for non-isothermal runs (#4262, #4281 and #4282).

    Note, the commercial simulator requires that the well must be declared as an injector using the WECONINJE keyword before entering the fluid's injection temperature with the WTEMP keyword. OPM Flow's behavior is more robust, due to storing the temperature unconditionally for all wells (#3218 and #4265). Thus, making the order of the WCONINJE and WTEMP keywords in the input deck irrelevant. Both keywords are in the SCHEDULE section..

    Thanks to Edmund Stephens for discovering the issue and providing an example model for analyzing the underlying issue.

36) For the Thermal Model, the simulator supports both the HEATCR and HEATCRT keywords in the GRID section; however, the implementation was incomplete, in that using HEATCR keyword would result in the simulation failing. This has now been addressed (#3410).

37) There was a problem with the adaptive time step algorithm, which uses the change of results between time steps, to estimate the next time step. In the case were there was no change in the solution between time steps, the algorithm may fail to calculate the next step, resulting in an assertion error and the simulator aborting. This has now been fixed (#4415).

38) Fix an error in the Todd-Longstaff model such that now the simulator only modifies the density, that is the effective gravity contribution, and not the volumes. The issue only effected the Solvent Model. The simulator now duplicates the results from test Case 2.1 in "A benchmark study on problems related to CO_2 storage in geologic formations" https://link.springer.com/article/10.1007/s10596-009-9146-x. See #773 for details.

39) Fix an issue with the well modeling code for injection wells looking for an artificial lift quantity (ALQ), which has now been addressed (#4461).

40) Fixed an issue when comparing a well's calculated BHP against a well's BHP constraint, for both producers and injectors (#4584).

41) Under some circumstance production wells under a THP constraint may converge with the well injecting instead of producing, due to how the vertical flow performance table was evaluated (see issue #4476). This has now been addressed (#4483).

42) There was an error in the code that resulted in referencing the wrong indices for well connections, this has now been corrected (#4304).

43) If the command line parameter *--allow-distributed-wells* has been set or defaulted to false, indicating that distributed wells are not allowed. Then the simulator attempts to move all the all perforated cells for the wells to one process. Previously, the simulator would throw an exception if this was not successful; however, the exception was only thrown on one process causing OPM Flow to hang afterwards. This has now been fixed by throwing on all processes. In addition, the error message has been improved to indicate the reason for the simulator stopping (#619).

44) Fixed an issue associated with how distributed wells were allocated across the available processes (#620). Secondly, we only previously moved the perforated cells of one well to the new partition. In the case that a cell was also perforated by another well we might have moved only a few cells of that other well. The rest would remain where they were previously. Hence the well might still be distributed between multiple processors and the user would see the following exception:

```
Error: [opm/grid/common/ZoltanPartition.cpp:132] Well is distributed between
processes, which should not be the case!
```

To solve this problem we now make a full search in the well graph starting with a cell of the well to be moved. This will effectively move the cells of all wells whose cells are reachable via the well graph. As a result there should be no distributed wells across processes (#621).

45) There was a bug associated with when the WTEST keyword executed at the beginning of (the first time step of) a report step, if the well was also created at that report step. This led to an exception being thrown due to the simulator trying to reference the potentials array. This has now been fixed (#4611 and #4609).

46) Fix bug in multi-segment well assembly where derivatives get overwritten whenever the segment flow direction is reversed, that is, when seg_upwind==outlet_segment_index (#4538).

47) There was an issue with writing out the segment connection pressures for multi-segment wells. Previously, the simulator incorrectly wrote the grid block pressures corrected to perforation depth, instead of the actual connection pressures. This has now been corrected (#4468).

48) For multi-segment wells, if there were declared connections not connected to the well's corresponding segments, then the simulator would crash. This has now been fixed for wells declared in the main input deck. However, for well specifications within ACTIONX and ACTIONW blocks, this may still be a problem, as like the commercial simulator, it is not possible to fully check all conditions within these blocks at the start of the run (#4329).

49) The simulator now checks for missing COMPSEGS keywords for declared multi-segment wells when reading the input deck, instead of during the simulation at the report step where the problem occurs. This is an improvement on #4329, that checked for the error during the simulation. The error message is now of the form:

```
unknown location(0): fatal error: in "MissingCOMPSEGS": Opm::OpmInputError:
Problem with keyword WELSEGS
In <memory string> line 38
Missing COMPSEGS keyword for the following multisegment well:
    PROD01 in <memory string> at line 38.
```

See #3293 for details.

50) For multi-segment wells with Inflow Control Devices ("ICD"), if the liquid fraction flowing through a ICD segment was negligible, then this would introduce singular matrices problems in the Solver, resulting in numerical issues. This has now been resolved by placing a threshold on the liquid fraction,

such that it is not used in the connection viscosity calculations if it is below the threshold value (*#4274*).

51) There was a bug in the handling of multi-segment wells when the number of segments was altered during the run. The bug cause the simulator to throw an exception and to terminate. This has now been fixed (*#4239*).

52) Refactored the standard well bottom-hole calculation to ensure that the BHP limit is honored for standard wells. The multi-segment equivalent code was working correctly (*#4500*).

53) Corrected a regression associated with getting the THP limit for producers in the Network Model (*#4289*).

54) Fixed a bug associated with a well's voidage injection rate calculation that did not take into account a well's water injection voidage rate (*#3346*).

55) In the SCHEDULE section, if the current value of WCONPROD(THP) is acting as a constraint instead of a target, and has been defaulted, then WCONPROD(THP) was skipped when checking for constraints in the Network Model. This has now been corrected (*#4407*).

56) For wells that have been STOPPED, the well mode of control indicator (WMCTL) should be set to zero, same as for SHUT well. Previously, this was not the case, but has now be corrected (*#3438*).

B.2.5 KNOWN ISSUES

1) There is an issue with the GCONPROD keyword when GCONPROD(TARGET) is set to LIQ and both phases are not present in the input deck. This will cause the simulator to abort. The work around is to set GCONPROD(TARGET) equal to a phase that is present in the deck. See *#3970* for a discussion.

2) There is an error with the RSM header for summary vectors whose NUMS entry in the SMSPEC file is derived from more than a single number source (e.g., single region or segment ID). This applies to all block vectors (BGPV, BOPV, BWPV, etc.), connection level quantities (COPT, etc.), and inter-region flows such as ROFT etc. The work around is to plot the data in OPM ResInsight and right-click on the plot to view and copy the data. See issue *#3078*.

3) OPM Flow does not support using LIQ as a well's preferred phase with the WELSPECS. keyword, that is WELSPECS(TYPE) equals LIQ. This is a long-standing bug/omission in the simulator stemming from a somewhat naive internal notion of phases so we don't have an entry for a liquid phase, only for the distinct oil and water phases. For producing wells this mostly matters if you plot the WPI summary vector (productivity index for well's preferred phase). In the current treatment WPI will not have contributions from the water phase if the declared preferred phase is LIQ. For injecting wells WELSPECS's preferred phase doesn't really matter at all since the preferred phase is (typically) reset to the injected phase in WCONINJE/WCONINJH anyway. See issue *#3075*.

4) If the simulator finds well connection being declared as connections via the COMPDATA keyword in the SCHEDULE section, then it writes out a warning message:

```
Warning: Problem with COMPDAT keyword
In SIM_PEE_Basic_Simulator.DATA line 535
The cell (0,6,4) in well INJ1 is not active and the connection will be ignored

Warning: Problem with COMPDAT keyword
In SIM_PEE_Basic_Simulator.DATA line 535
The cell (16,0,4) in well INJ2 is not active and the connection will be ignored
```

However, the reported cell references are offset by minus one, meaning the correct warning messages should be:

```
Warning: Problem with COMPDAT keyword
In SIM_PEE_Basic_Simulator.DATA line 535
The cell (1,7,5) in well INJ1 is not active and the connection will be ignored

Warning: Problem with COMPDAT keyword
In SIM_PEE_Basic_Simulator.DATA line 535
The cell (17,1,5) in well INJ2 is not active and the connection will be ignored
```

See issue *#3167* for details.

5) Pull request *#3192* fixed an incorrect JSON definition of the GASVISCT keyword in the PROPS section, that prevented a thermal model from running. However, thermal models containing the GASVISCT keyword will still not run, and instead will write out the following error message:

```
Error:
An error occurred while creating the reservoir properties
Internal error: Tried to get back() from empty DeckView

Error: Unrecoverable errors while loading input: Tried to get back() from empty
DeckView
```

Commenting out the keyword will enable the case to run. See issue *#3200* for details.

6) The tracer related output to summary file does currently not respect the command line variable - enable-tracer-mode, this means we may have the situation in which the input deck specifies tracer output, and the simulator is run without enabling the tracer model (enable-tracer-mode=false). In this scenario the trace SUMMARY vector output will consists of irrelevant tracer values.

7) The GDFILE keyword in the GRID section loads a grid file in various formats, with the FMTOPT parameter setting the format type of the file. If the variable FMTOPT is omitted then the default is for binary file input for the commercial simulator; whereas, OPM Flow derives FMTOPT from the file extension (*.EGRID or *.FEGRID), making FMTOPT superfluous. However, if the extension is lower case then OPM Flow may incorrectly determine the file type. The work around is change the extension to upper case.

8) As per previous releases of the radial model, the COORDSYS keyword item three must be set to COMP to complete the circle, this has not been implemented in this release. Also there appears to be a bug for full radial models when a well goes on BHP control that causes the well not to respect the BHP constraint, this eventually causes the well to die prematurely. See *#2640* for a discussion on the topic.

9) As in previous releases there are some issues with the OPERATE and OPERATER keywords associated with the input parsing; for various reasons a few of the fields require special case treatment in the grid processing, including (at least) MULTZ, PORV and ACTNUM, and for those keywords the OPERATE/OPERATER keyword doe not work. The work around is to use the MULTIPLY keyword instead.

10) For the UDQ ASSIGN operator after the terminating "/" normally any comments can be entered; however, if there is "/" within the comment field, as per:

```
ASSIGN FUNGLYLD 1.196   /    Condensate Yield (63.5/56.7)/(1.0 – 0.065)
```

then the simulator will abort. The work around is to manually place the comment characters "--" after the ASSIGN terminating "/", like so:

```
ASSIGN FUNGLYLD 1.196   / -- Condensate Yield (63.5/56.7)/(1.0 – 0.06)
```

11) At the moment, one cannot initialize tracers using the EQUALS keyword. Instead use the array format, that is the keyword followed by the required number of values, or the TVDP keyword in the SOLUTION section to set the initial tracer concentrations as a function of depth.

12) Currently, gas tracers cannot be used if the dissolved gas phase, as per the DISGAS keyword in the RUNSPEC section, is active in the model.

13) In some cases when the program stops the error message is written to the terminal but not to the *.PRT or *DBG files. This is because the failing code is not aware of the C++ logging system. For example, this will occur if there is an error in parsing the grid data as the corner-point processing code is written in C and at the moment cannot call the C++ logging system. (*#3896*).

14) The summary vector RTIPTHEA, that defines the energy in-place between the initial and the current time for regions, is not supported unlike the FTIPTHEA and BTIPTHEA vectors. Secondly, the error message:

```
Warning: Problem with summary keyword RTIPTHEA
In RSM-THERMAL.data line 492
FIP region FIPHEA not defined in REGIONS section - RTIPTHEA ignored
```

is incorrect, as the message indicates that it is being treated like a named region, as per the FIP keyword, when it is actually a SUMMARY vector (*#3870*).

15) If there are cells that are very distorted, which can occur near fault planes, then the simulator may abort because it cannot calculate the pore volume of such cells. The work around is to re-generate the grid in the static model, taking care that the cells around the fault planes are more or less orthogonal (*#2992* and *#3770*).

16) Currently the OPERATOR keyword in the EDIT section does not work with the DEPTH, TRANX, TRANY and TRANZ property arrays (*#2994* and *#748*).

17) If a standard well is fully declared in an ACTIONX block which is then activated at a later date, and later the well is modified to be a multi-segment well using the WELSEGS and COMPSEGS keywords, then this will cause the simulator to abort with an assert failure. The solution to this issue is to not use this type of work flow in declaring wells (*#2891* and *#2895*).

18) Although the ACTIONX EXIT command works as expected, it does not write out the requested RSM file at the end of the run. However, the other SUMMARY and RESTART files are written out (*#2877*).

19) Although the GCONSUMP keyword in the SCHEDULE section is fully implemented as documented, it is not possible to verify the output as the associated SUMMARY vectors are not written out, that is the SUMMARY sales gas vectors FGSR, FGST, GGSR and GGST, and fuel vectors FGCR, FGCT, GGCR, and GGCT have not been implemented (*#2679*).

20) There are small differences in the behavior of the NEXTSTEP keyword in the RUNSPEC section between OPM Flow and the commercial simulator that remain unresolved (*#3745*).

21) **There is a unit handling issue associated with OPERATE keyword. If the OPERATE(X) parameter has units, as for example PERMX, then the conversion is always done in SI units, despite the input deck declaring the deck to be fields units, as per FIELD keyword in the RUNSPEC section. Note that OPM Flow performs all of its calculations internally in SI and performs unit handling only when inputting the *.DATA file and when outputting result files. Thanks to *lrijkels* for reporting the issue. See *#4597* for details.**

22) **There is an issue associated with restarting from a restart file with the solution gas (Rs) maximum rate of increase, as defined by the DRSDT keyword in the SCHEDULE, that has been set to zero. This is because, the simulator does not save/restore this setting in simulator's restart files, which means that simulator misses the essential value zero upon restarting the case. As a work-around one can use the option --sched-restart=true, when running the restart case. This will initialize the restarted simulation based on information from the complete SCHEDULE section, instead of just the parts that we're going to simulate and the rest from the restart file. Thanks to *goncalvesmachadoc* for reporting the issue. See *#4272* for details.**

Jostein Alvestad, Kai Bao, David Baxendale, Markus Blatt, Joshua Charles Bowden, Paul Egberts, Arne Morten Kvarving (Release Manager), David Landa-Marbán, Cintia Goncalves Machado, Halvor Møll Nilsen, Håkon

Hægland, Ove Sævareid, Atgeirr Flø Rasmussen, Alf Birger Rustad, Tor Harald Sandve, Bård Skaflestad, Torbjørn Skille, and Pieter J. Verveer.

B.3 RELEASE 2022-10

The 2022-10 release consists of some new features and various improvements and bug fixes. Highlights for this release include: implementation of alternative boundary conditions to the model via the BC keyword in the SOLUTION section, directional relative permeability support, implementation of Killough's hysteresis model together with hysteresis modeling for both relative permeability and capillary pressure, incorporation of the Joule-Thomson effect in energy calculations, added partial support for NETBALAN keyword, added support for the LETI[349] family of relative permeability and capillary pressure functions that can be used as a replacement for Corey[350] type curves, and implemented the introduction of a new preconditioner to the linear solver by extending the existing Constrained Pressure Residual ("CPR") preconditioner to include wells via the -linear-solver="CPRW" command line option. In addition, more SUMMARY vectors are now supported, plus various enhancements to the RESTART file to improve compatibility and robustness. Significant work has also been conducted in fixing the number of reported bugs.

When building OPM Flow from source, the default is now to compile a parallel binary if MPI is installed, this was already the default for the binary packages that were previously distributed. Secondly, support for building OPM Flow with the new current version of DUNE (2.9) has been added.

B.3.1 NEW AND DEPRECATED COMMAND LINE OPTIONS

The major command line changes made for this release are summarized in Table B.5

OPM Flow 2022-10 New And Deprecated Command Line Options

No.	Variable Name	Description	Default
1	--linear-solver	A defined quoted character string that sets the configuration of the solver, valid values are: 1) ilu0 (default), 2) cpr (an alias for cpr_trueimpes)[351] [352] and [353] 3) cpr_quasiimpes, 4) cpr_trueimpes, 5) cprw, 6) amg,[354] or 7) a file name that has the extension ".json", that contains the linear solver configuration parameters. Option (5) extends the existing Constrained Pressure Residual ("CPR") preconditioner to include wells.	"ilu0"

[349] *Lomeland F., Ebeltoft E. and Thomas W.H., 2005. A New Versatile Relative Permeability Correlation. Paper SCA2005-32 presented at the International Symposium of the Society of Core Analysts held in Toronto, Canada, 21-25 August, 2005.*

[350] *Corey, A. T.: "The Interrelation Between gas and Oil Relative Permeabilities", Production Mon., 19. 38. (1954).*

[351] *Wallis, J. R., Little, T. E., and Nolen, J. S.: "Constrained Residual Acceleration of Conjugate Residual Methods," paper SPE 13536 presented at the SPE Reservoir Simulation Symposium, Dallas, Texas, USA (February 10-13, 1985).*

[352] *R. Scheichl, M. Roland, J. Wendebourg, Decoupling and block preconditioning for sedimentary basin simulations, Computational Geosciences 7 (2003) 295{318.*

[353] *Klemetsdal, Ø.S., Møyner, O. & Lie, KA. Accelerating multiscale simulation of complex geomodels by use of dynamically adapted basis functions. Comput Geosci 24, 459–476 (2020). https://doi.org/10.1007/s10596-019-9827-z.*

[354] *M. Blatt, A parallel algebraic multigrid method for elliptic problems with highly discontinuous coefficients, Ph.D. thesis, Ruprecht-Karls-Universität Heidelberg (2010).*

OPM Flow 2022-10 New And Deprecated Command Line Options			
No.	**Variable Name**	**Description**	**Default**
		For option (6) one enters a character string enclosed in quotes that defines the filename of a JSON configuration file for the flexible linear solver system. In this case the file extension should be '.json'. Note that the *.PRT file contains the "Property tree for the linear solver" listing, which is the JSON specification of the current case, and can be used as a guide to configure a user specific linear solver JSON file. See *#4125*.	
2	--linsolver	Renamed to --linear-solver See *#4125*.	"ilu0"
3	--linear-solver-verbosity	A positive integer value that defines the output from linear solver: 0 : No extra output, 1 : output per solution iteration, or 2 : output per iteration. See *#4130*.	0
4	--flow-linear-solver-verbosity	Renamed to --linear-solver-verbosity. See *#4130*.	0
5	--linear-solver-require-full-sparsity-pattern	A Boolean value set to true or false that if set to true the simulator will produce the full sparsity pattern for the linear solver. Removed, see *#4243* and *#4255*.	false
6	--min-strict-cnv-iter	A positive integer that sets the minimum number of Newton iterations before relaxed tolerances are used for the CNV convergence criterion. See *#4086*.	0
7	--max-strict-iter	Renamed to --min-strict-cnv-iter See *#4086*.	0
8	--newton-max-iterations	A positive integer that defines the maximum number of Newton iterations per time step used by the simulator. See *#4234*, *#4185*, *#4223*, *#741* and *#749*.	20
9	--flow-newton-max-iterations	Renamed to --newton-max-iterations. See *#4234*, *#4185*, *#4223* and *#741*.	
10	--newton-min-iterations	A real positive integer that sets the minimum number of Newton iterations per time step used by the simulator. See *#4141*.	1

OPM Flow 2022-10 New And Deprecated Command Line Options			
No.	**Variable Name**	**Description**	**Default**
11	--flow-newton-min-iterations	Renamed to --newton-min-iterations See *#4141*.	
12	--cpr-reuse-interval	A positive integer that sets the reuse preconditioner interval. Used when --cpr-reuse-setup is set to 4, then the preconditioner will be fully recreated instead of reused every N linear solve, where N is this parameter.	10
13	--regularization-factor	A real positive value that defines the "regularization factor" for wells. Previously, --regularization-factor-msw was applied to only multi-segment wells, and is now replaced by this new variable for all well types.	100
14	--relaxed-pressure-tol-msw	A real positive value that sets the relaxation tolerance for the multi-segment well pressure solution in Pascals.	10000
15	--relaxed-well-flow-tol	A real positive value that sets the relaxation tolerance for the well flow residual in reservoir cubic metres (rm3).	0.001
16	--shut-unsolvable-wells	A Boolean value that determines if the simulator should shut unsolvable wells (true), or not (false).	wahr
17	--vtk-write-diffusion-coefficients	A Boolean value that switches on (true) or off (false) the output of the molecular diffusion coefficients to the VTK[355] output files.	false
18	--vtk-write-effective-diffusion-coefficients	A Boolean value that switches on (true) or off (false) the output of the effective molecular diffusion coefficients for the medium to the VTK output files.	false
19	--vtk-write-tortuosities	A Boolean value that switches on (true) or off (false) the output of the tortuosity for each phase to the VTK output files.	false

Notes:

1) Cells colored in green in the "No." column indicate a new command line option or a change to an existing option for this release.
2) Cells colored in orange in the "No." column indicate the command line option is available but is now "hidden" from the main help listing, --help. These options can be listed using the --help-all command line option, that lists all the command line options included in the release, including experimental, obsolete, hidden and deprecated options.

[355] *Virtualization ToolKit ("VTK") files used to visualize the 3D results from the model using ParaView, an open-source, multi-platform general purpose data analysis and visualization application (https://www.paraview.org/overview/). For the commercial simulator's compatible output files OPM's ResInsight 3D visualization package can be used instead and is specifically tailored for OPM Flow and the commercial simulator.*

OPM Flow 2022-10 New And Deprecated Command Line Options			
No.	**Variable Name**	**Description**	**Default**
3) Cells colored in red in the "No." column indicate a deprecated command line option for this release.			

Table B.5: OPM Flow 2022-10 New and Deprecated Command Line Options

B.3.2 NEW FEATURES

In addition to the above the following new features have been added to the simulator:

1) Although the OPM Flow BC keyword in the SOLUTION section, that defines alternative boundary conditions to the model, has been available in previous releases, the keyword has not been documented in the manual. The keyword is now fully documented with the order of the parameters on the keyword changed from previous releases, in order to be more consistent with other BOX type keywords (*#3208* and *#3193*). Note that although the BC(TYPE) equal to DIRICHLET option did not make it into this release, it is expected to be in the next release.

2) Directional relative permeability assignment has now been implement using the KRNUM keyword (*#532*, *#4048*, *#714*, and *#3113*).

3) Added preliminary support for the Dune ALUGrid module, that provides an adaptive, load balancing, and unstructured implementation of the DUNE grid interface in two or three space dimensions supporting either simplices (triangle or tetrahedron) or cube elements (*#3972*).

4) Added support for Killough's hysteresis model, that is EHYSTR(HYSTMOD) equal to two and three. Together with hysteresis modeling for both relative permeability and capillary pressure, EHYSTR(HYSTOPT) equal to BOTH, PC and KR (*#3170*, *#535* and *#4148*).

5) Implemented OPM Flow specific keywords for the Joule-Thomson coefficients for gas, oil and water phases, for incorporating the Joule-Thomson effect in models (*#3029*), as well as incorporating Joule-Thomson effect in energy calculations (*#500*). See keywords GASJT, OIL JT and WATJT in the PROPS section.

6) Add supported for the LET family of relative permeability and capillary pressure functions (*#3018*, *#2917* and *#3756*).

7) Add support for MINPORV keyword, which is an alias for MINPV keyword. Also added additional checks, such that only one of MINPV and MINPORV can be active in the deck (*#3071* and *#3975*).

8) Implemented partial support for NETBALAN keyword in the SCHEDULE section, only the first three items are currently supported; however, work is ongoing to increase the functionality of the keyword (*#3954*).

9) Added a new preconditioner to the linear solver by extending the existing Constrained Pressure Residual ("CPR") preconditioner to include wells. The feature is activated via the command line parameter:

```
--linear-solver=cprw
```

and the feature should be considered experimental at this stage (*#3937* and *#3901*).

10) As per previous releases, the simulator has had an additional formulation to the standard polymer flooding model, known as the Polymer Molecular Weight Transport option, that uses the polymer molecular weight in calculating the polymer viscosity; however, the keywords associated with this model have not been documented. This has now been rectified in this release of the manual. The model is activated via the POLYMER and POLYMW keywords in the RUNSPEC section. The model does not account for non-Newtonian flow; the apparent viscosity is simply set equal to the zero-shear viscosity. Secondly, the standard polymer property data keywords: PLYROCK, PLYADS, PLYMAX, etc., are still required to fully describe the polymer fluid.

Although the Polymer Molecular Weight Transport model has been tested using metric units, using either field or laboratory units with the option should be considered experimental.

11) Added support for a simulation case to be run via Python and for dynamical changing the SCHEDULE section after the deck has been read from a file by a Python function (*#2833*, *#2821*, *#2781*, *#3910*, *#3031*, *#3918*, *#3037*, and *#3040*)

12) RESTART file additions and improvements:

    a) Improved RESTART file compatibility with the commercial simulator by adding a "small rate threshold" when outputting the lift gas max supply rate limit and total gas production limit. The threshold renders the output discontinuous, but is needed for compatibility with the commercial simulator (*#3039*).

    b) Added support for loading the NETBALAN keyword parameters from a RESTART file to enable using the parameters in a restarted run (*#3095*). Added restart support for the NETBALAN keyword (*#3095*, *#3103* and *#3110*).

    c) Improved RESTART file compatibility when Network Balancing is being performed, as requested via the NETBALAN keyword in the SCHEDULE section. The improvement takes into account the minimum time step size for network balancing, NETBALAN(NTSTEP), as well as preserving negative balancing intervals exactly instead of replacing these values by zero (*#3032*).

    d) Added the oil-water (PCOW) and oil-gas (PCOG) arrays to the restart file (*#3904*).

    e) Added support for UDA variables for the WELTARG keyword in reading and writing the RESTART file (*#3005*).

    f) Identified additional entries for WECON, WGRUPCON, and WVFPEXP keywords and fix an incorrect item attribution for the WTEST 'reason' parameter (*#3021* and *#3026*). Added restart support for known well economic limits; however the following well economic limits are not currently supported: maximum gas-liquid ratio WECON(GLR), maximum temperature WECON(TEMP), maximum reservoir fluid flow rate WECON(RESV), and follow-on well WECON(WELOPEN) (*#3095*, *#3103* and *#3110*).

    g) Fixed an issue with the well list positions for individual wells (i.e., IWLS) not strictly increasing in the restart files (*#3052*).

    h) The simulator can now restore the explicit THP control options (WVFPEXP keyword) from the RESTART file; thus, improving compatibility with the commercial simulator (*#3114* and *#3110*).

13) Implemented a new and faster linearization approach specifically for the Two-Point Flux Approximation ("TPFA") method used in the simulator. The new version of the linearizer is used for one, two and three-phase black-oil simulations; whereas, the original linearizer is still used for extended models such as solvent, polymer etc. Testing indicates a significant speedup for the linearization phase, yielding an improvement in total run times of 6-8% or more, depending on the case. (*#4017*).

14) Added water evaporation into gas in combination with oil vaporization (*#688*). See also *#3869* for water evaporation and salt precipitation.

15) Added water evaporation and salt precipitation for three-phase runs (oil, water and gas). Previously the simulator only supported gas-water models. Note that currently the simulator does not support:

    - Water evaporation with multi-segment wells, only standard wells are supported.
    - Brine with multi-segment wells, only standard wells are supported.

    See *#3869* for details.

16) Previously for the Vaporized Model, only Enumeration Initialization was supported via the RVW keyword. This has been enhanced by adding support for Equilibrium Initialization with the support of the OPM Flow specific keyword, RVWVD, that allows for the initial vaporized water-gas ratio to be

defined as a function of depth (#3107). In addition, the EQUIL keyword in the SOLUTION section was extended via EQUIL(EQLOPT6) parameter to allow for the RVWVD keyword to be used similarly, as for the RVVD keyword (#4034).

17) Added support for WCONINJE(RSRVINJ) parameter that defines the injected dissolved gas ratio in injected oil or the vaporized oil ratio in injected gas (#2999 and #3879).

18) Implemented support for the WVFPEXP keyword in the SCHEDULE section with the WVFPEXP(IMPEXP) and WVFPEXP(CONTROL) options (#4100 and #3837).

B.3.3 IMPROVEMENTS

Improvements include:

1) Improved the date reporting for the ACTIONX keyword to avoid ambiguous dates. The simulator now reports dates in DD-MMM-YYYY, for example:

```
Action ACT-01 triggered at 02-Nov-2018 00:00:00 (report interval 0 to 1).
```

See #4003 for more information.

2) Previously whether or not an ACTIONX block was triggered the simulator would write out a message to the *.PRT file at every time step, which in some cases produced excessive number of messages, especially for non-triggered blocks (issue #4042). Now the simulator writes this information to the *.DBG file, and only writes message to the *.PRT file for triggered ACTIONX blocks. Additional improvements to the print output included sending the list of keywords to be executed when an ACTIONX block was triggered to the *.DBG file, instead of the *.PRT file (#4047 and #3112).

3) Improved how the ACTNUM property was processed by the simulator for all known grid types, which also fixed a special case that was not handled previously (#3063). In addition, improved the numerical aquifer connection logic to ensure that only active cells are connected to the aquifer (#3064).

4) Adapt drift compensation code to account for isolated cells and various other improvements (#3968 and #699).

5) Re-factored the analytical and numerical aquifer code to have a common base, and implemented the code in the black-oil aquifer model (#3994).

6) Previously, if there were any invalid aquifer connections, the simulator world print a warning message for every invalid connection. This would lead to an excessive number of associated warning messages printed to the terminal console, in one case over 50,000 messages were printed. Now only the first ten warning messages are printed to the console, and in addition the total number of invalid connections found. All warning messages are still printed to the print file (#3098 and #4007).

7) The AQUCT(AQUTAB) parameter defines the user entered Carter-Tracy AQUTAB table to be used for the aquifer. If the parameter is defaulted, then the simulator uses the Radial Flow, Constant Pressure and Constant Terminal Rate Cases for Infinite Reservoirs (Table I) in Van Everdingen and Hurst's [356] paper, for AQUCT(AQUTAB) equal to one. Previously, the internal table had a maximum dimensionless time (tD) value of 100, which resulted in some models not matching the commercial simulator. The internal table has now been extend to a maximum dimensionless time value of 10,000, which should enable better consistency with the commercial simulator (#3099).

8) Improved the functionality of analytical aquifers by adding support for analytical aquifers to be used with OPM Flow's thermal modeling option. This includes adding support for AQUCT(TEMP) in the GRID section and AQUFETP(TEMP) in the SOLUTION section. Note that currently, one cannot use the restart facility when using the thermal model combined with analytical aquifers (#3109 and #4035).

[356] *Van Everdingen, A. & Hurst, W.,. The Application of the Laplace Transformation to Flow Problems in Reservoirs. Petroleum Transactions, AIME (December, 1949).*

9) Improved the BHP and IPR calculation so that the simulator obtains the highest valid production rate for a well (#3924). In addition, the current check for well operability using a well's IPR is different between standard and multi-segment wells. Now we check that all components have negative IPRs before making both well types inoperable. Secondly, fixed the hydrostatic correction for branched wells in the IPR calculation for multi-segment wells, as well as component and phase reference errors (#3926).

10) The COMPDATA(DIRECT) parameter, that sets the direction of the connection, previously only supported uppercase values of X,Y and Z; this has now been improved by supporting both upper and lower case values (#3171). Fixes issue #3164.

11) Previously, the *.DBG file contained hundreds of *computeBhpAtThpLimitProd()* messages per well, which made examining the output difficulty, these messages have now been switched off, but are still available to developers via a compile switch (#4097).

12) Improved the error logging for the FAULTS keyword when FAULTS (FLTNAME) is greater than eight characters. Now the simulator issues a warning message and truncates FAULTS(FLTNAME) to eight characters (#3065) and #3959).

13) Improved the error handling associated with being unable to find a fault when using MULTFLT keyword, due to either the fault name not being defined in the FAULTS keyword, or the fault name being greater than eight characters:

```
An error occurred while creating the reservoir properties
Internal error: Could not set fault transmissibility multiplier 0 for fault
FAULT01: Key FAULT01 not found.

FAULTS: invalid value 'FAULTLONG1' in record 1 for item 1
In file: MULTFLT-02.DATA, line 220
FAULTS(FLTNAME): Only names of faults up to 8 characters are supported. Will
ignore excess characters.
```

See #3062 and #3077 for details.

14) Improved the error logging for MULTZ keyword, since this keyword is handled differently than the other MULT keywords in order to allow the PINCH keyword PINCH(PINCHMUL) option to be processed. Previously the error message was:

```
Error:
An error occurred while creating the reservoir properties
Internal error: Region operations on 3D fields with global storage is not
implemented
```

Now the simulator issues the following message:

```
Error: Unrecoverable errors while loading input: Problem with keyword MULTIREG
In MULTIREG-MULTZ.DATA line 224
region operation on 3D field MULTZ with global storage is not implemented
```

See #3055 for further information.

15) Improved error logging for multi-column keywords, that is for SWOF etc., to more precisely inform the user the location of the error. Previously, if the number of columns was incorrect the simulator would issue the following message:

```
Error:
An error occurred while creating the reservoir properties
Internal error: Number of columns in the data file is inconsistent with the ones
specified
```

Now the simulator issues the following message:

```
Error: Problem with keyword SWOF
In MOD01-TEMP.DATA line 308.
Internal error: For table with ID 1: Number of input table elements (82) is not a
multiple of table's specified number of columns (4)
```

See #3045 and #3047 for further information.

16) Improved the handling of the ESMRY and ExtESmry files to add robustness to the concurrent usage of the files, so that one can safely load data from the files that are being created/updated by an active run (#3028).

17) Changed the terminal message for group switching control history. Previously, the full history was printed to the terminal, that is *RATE->FLD->RATE->FLD*. Now only the final control status is printed to the terminal and the previous format is now routed to the *DBG file (#4033).

18) Improved the convergence for problematic multi-segment wells by setting the default parameter for the command line parameter regularization_factor_ms_wells_ = 1.0 (#3864).

19) Improved numerical performance by applying heuristic scaling of trueimpes weights for the cpr and cprw methods (#4194 and #4230).

20) Changed multi-segment well "Regularization Factor" from one to 100 to increase multi-segment wells capability to flow at lower rates and to fix a bug introduced with the recently added LET commit (#3906). See also #3859 for a discussion on the topic covered by this change.

21) Various Open Computing Language ("OpenCL") improvements to the solver. See #3815, #3928, #3946, #3953 and #3973, which reverts #3953 due to compilation errors on certain C++ compilers.

    OpenCL is a framework for writing programs that execute across various platforms consisting of central processing units ("CPUs"), graphics processing units ("GPUs"), digital signal processors ("DSPs"), field-programmable gate arrays ("FPGAs") and other processors or hardware accelerators. OpenCL specifies programming languages (based on C99, C++14 and C++17) for programming these devices and application programming interfaces (APIs) to control the platform and execute programs on the compute devices. OpenCL provides a standard interface for parallel computing using task- and data-based parallelism. OpenCL is an open standard maintained by the *Khronos Group* a non-profit consortium.

22) Improved the handling of various PVT keywords, including: PVDO, PVDG, PVTO, PVTG, PVTW, DENSITY, and GRAVITY by incorporating the default behavior of using the previous table as the default for the next table, provided of cause the first table as been fully defined. For example, if there are two PVT regions, then one can default the second table by:

```
--
--        WATER PVT TABLE
--
PVTW
--        REF PRES  BW         CW         VISC     VISC
--        PSIA      RB/STB     1/PSIA     CPOISE   GRAD
--        --------  --------   -------    ------   ------
          4840.0    1.019      2.7E-6     0.370    1*         / TABLE NO. 01
                                                               / TABLE NO. 02
```

    This now works for all of the aforementioned keywords (#3058 and #3056).

23) Improved the *summary* utility, a standalone application that is used to list and extract data from SUMMARY files. The program will now list and query completion related summary vectors using keys of the form *Keyword:Wellname:CompletionNumber* e.g., WOPRL:P1:3. Previously, the keys were of the form *ComplKeyword:Wellname*, that is e.g., WOPRL__3:P1. See #3094.

24) In OPM Flow's Black-Oil Thermal model, temperature limits were applied during the Newton iterations that resulted in incorrect results. These have now been removed; thus, improving the overall results for these type of simulations (#712).

25) In OPM Flow's Black-Oil Thermal model energy calculations, the energy residual is now scaled so that it has the same relative order as the mass balance residuals. Previously, the simulator was using the same tolerances for the MB and CNV for energy and the various mass balance equations. This change makes the energy tolerance now more comparable with the ones for mass (#4037).

26) Improved the code base for identifying which wells are active and which wells are shut-in for both standard and multi-segment wells (#4101). See also #4102.

27) Previously when a well was changing control mode during a time step, the simulator printed each of control change. Now only the initial and final control modes are printed, and if the initial and final are the same then no message is printed. For example:

```
BHP->THP->ORAT->THP is reduced to BHP->THP
and
BHP->THP->BHP will not be reported.
```

Note that the previous printing behavior is now routed to the *.DBG file (#4143).

28) Improved the convergence of newly opened wells (#4149). Secondly, only apply drift compensation when wells are active (#4102).

29) Improved well and group convergence by only accepting solutions for when the controls have not been violated (#4094). Also, improved model performance by avoiding oscillation between ORAT and LRAT, when they are essentially equal (#4134).

30) Wells under group control may violate their minimum THP due to the well's group imposing a lower rate, but are restricted from changing to rate control (see the WVFPEXP keyword in the SCHEDULE section). Under these circumstances, these wells would not be allocated gas lift gas, even though previously they were under gas lift. This has now been changed, so that wells will continue to receive gas lift gas as required (#4054).

31) Improved the standard well code by applying a "Regularization Factor" to avoid problematic wells prematurely shutting-in (#3957).

32) The simulator now allocates gas lift gas, or other artificial lift quantity, prior to attempting to solve the equations when the well is being tested if it can flow to surface (#4155).

33) Improved well tolerance criteria by relaxing the tolerance after six Newton iterations so that unsolvable wells remain open (#3859).

34) Various improvements to OPMRUN, including corrections and additions to the keyword templates, together with the Production Schedule Utility being extended to generate production schedules based on monthly rate and monthly volume input data (#73 and #72).

In addition, the following new SUMMARY keywords are now recognized as described by the comments in Table B.6.

No.	Summary Keyword	Comment
1	BGPV, BOPV, BRPV and BWPV	Added block pore volume summary vectors BGPV, BOPV, BRPV and BWPV (#3016).
2	BOIP, BOIPG, BOIPL, BGIP, BGIPG, BGIPL, BWIP, BRS, and BRV	Added SUMMARY vectors BOIP (block-level oil-in-place) and BGIPL (block-level gas-in-place in liquid phase), etc.. In addition, also add support for outputting the block-level dissolved gas-oil-ratio (BRS) and vaporized oil-gas-ratio (BRV) - see #3002 and #3886.

No.	Summary Keyword	Comment
3	GEFF, WEFF, and WEFFG	Added support for the GEFF, WEFF, and WEFFG summary vectors that report the efficiency factors at the well (WEFAC) and group (GEFAC) levels. WEFFG additionally accounts for efficiency factors in a well's superior groups in the group tree (#3060).
4	Well Connection Vectors	Improved support for writing out well connection vectors, to the SUMMARY file (#3094 and #3091).

Table B.6: New SUMMARY Keywords for the 2022-10 Release

B.3.4 BUG FIXES

The following bug fixes and improvements have been incorporated into this release.

1) Previously the simulator was unable to correctly process cells that were made inactive via the ACTNUM keyword, and then later in the deck re-activated using the EQUALS and ACTNUM keywords. This has now been rectified (#3049).

2) Fixed an issue with the aquifer models and the dune-alugrid module. The module provides an adaptive, load balancing, and unstructured implementation of the DUNE grid interface in two or three space dimensions supporting either simiex or cube elements (#4128).

3) There was an equilibration problem with cells having zero thickness, when the offending cells were used as numerical aquifer cells, this has now been corrected (#4127).

4) Corrected several issues with analytical aquifers and their connections. The simulator now issues a warning message if there are connections to undefined aquifers and for the case when there are aquifers with no connections. For the later scenario, the aquifer is disabled (#3102 and #4015).

5) When using the THERMAL option combined with BC keyword, the free boundary, that is the boundary conditions not stipulated by the BC keyword, would potentially have erroneous temperature data on the edge blocks. This has now been corrected in this release (#753 and #746).

6) Fixed a coding bug in the Cartesian mapping code (#4009).

7) The block-jacobi-ILU preconditioner did not work with cusparseSolver, due to zeros on the diagonal, the bug prevented the zeros being removed. The fix should improve performance for this solver (#3935).

8) Fixed a bug associated with copying turtuosity and diffusion coefficients if diffusion was not enabled, that is when the DIFFUSE keyword in the RUNSPEC section is not in the input deck. Previously, the implicit assignment operator was used causing the copying to occur unconditionally. This fix also improved run time performance for when diffusion is not enabled (#692).

9) Fixed two issues with the diffusion model. The first was related to conversion from molar to surface rates, the second was related to convergence and derivatives of the gradient. The changes solved issues with oscillations and low diffusion flux (#695).

10) Previously, for the DRSDT keyword (Solution Gas (Rs) Maximum Rate of Increase Parameters), was not written out to the RESTART file (#3977), and the actual units were missing from the output files, this has now been rectified (#3080).

11) Previously, the simulator would silently ignore any JFUNC keyword data in the GRID section, unless the end-point scaling option was activated through the ENDSCALE keyword in the RUNSPEC section, resulting in the run continuing without informing the user of the issue. Now the simulator issues an error message if the JFUNC keyword is present and the ENDSCALE keyword is absent, and will also now terminate the run (#3038).

12) Fixed a bug with the loading of vectors from the ESmry and EXTESmry files that resulted in the vectors being loaded twice (#3033 and #3059). In addition, fixed an issue with the start date format that should have seven items, with the last element representing milliseconds (#3154).

13) Fixed a bug with the writing of the ESmry summary files, caused by the commercial simulator and OPM Flow creating time vectors that are dummy data and not used. This caused an issue with the creating the ESmry file (#3100).

14) Fixed a bug with the incorrect calculation of the FOE (oil recovery factor based on STOIIP) SUMMARY vector (#3048 and #3939).

15) Modified how gas lift assignment is performed. Now the maximum ALQ value is assigned to each gas lift producer at the start of the time step. This is done to allow the wells to attain a positive potential and thereby be considered as open. Then later in the time step, the full gas lift optimization procedure is used can adjust the ALQ to its "correct" value. Secondly, in some cases when gas lift is switched off by setting ALQ to zero, and later in the schedule is switched back on again, it might not be possible to determine BHP from THP and the iteration fails to converge due to small ALQ values. Now instead of aborting the gas lift optimization, we try increasing ALQ until we get convergence or until the maximum ALQ for the well is reached (#3868).

16) Fixed an issue with VFP calculations for two phase gas lift that did not work unless the rates are adapted to include a zero gas rate (#3876 and #3868).

17) There was an issue with the GASVISCT keyword in the PROPS section not being recognized by the simulator, which has now been rectified (#3192 and #3207). However, there are still issues using this keyword, see Known Issues for details.

18) Fixed an issue associated with how the grid transmissibilities are calculated in parallel runs to ensure consistency in load balancing (#3929 and #3936).

19) Fixed an error on how group constraints were applied. Now the simulator starts at FIELD level and moves downwards in the hierarchy checking the constraints. Thus, the simulator first checks if the group violates any constrains from above and then if it violates any individual constrains (#4135).

20) Refactored the code and fixed a bug associated with checking for violation of group constraints for multiple level of guide rates (#4029).

21) Fixed a bug introduced in #3864, that caused the regularization term to be used in all subsequent linearizations until the next report step when the regularize variable is reset to false. With this change the regularization terms are only applied after the strict number of inner iterations are completed (#3909).

22) Fixed several bugs with the simulator not recognizing the supported KRNUM series of keyword and the DIFF keyword, as well as the unsupported HAxxxxxx, HMxxxxxx, and HMMULTxx series of keywords (#3054).

23) Using the KRNUM keyword resulted in a warning message stating incorrectly that the keyword was not supported, when in fact it is. This has now been corrected (#4212 and #4253).

24) Previously when a keyword was missing a required terminating "/" the simulator would issue the following message:

```
Error: Unrecoverable errors were encountered while loading input
```

Which made finding the error challenging. Now the simulator will print the following message:

```
Error: Unrecoverable errors were encountered while loading input: Problem with
keyword DATES
In PROJ2_RESTART2.DATA line 408
Keyword is not properly terminated
```

Thus enabling the user to identify and correct the error (#3027 and #3912).

25) Previously, the simulator issued erroneous warning messages for the NNC keyword in GRID section when NNC(FACE1) and NNC(FACE2) were defaulted, this has now been corrected (*#3074*).

26) Fixed a bug with the equilibration of numerical aquifers when the numerical aquifer reference depth is significantly deeper than the OWC that caused the simulator to extrapolate the aquifer pressure to a NaN (Not a Number) - see *#3911*.

27) Fix an issue associated with premature shutting of wells for when a well is controlled by a group that has zero rate. Due to numerical instability the well still gets an epsilon rate and skips the improved initialization used for wells with trivial rate. Adding an epsilon in the check is sufficient to keep the well open and ready for production when the group gets a non-trivial target (*#3887* and *#3888*).

28) Fixed an issue with the PVTG keyword if the Rv values all had the same value (*#3850*, *#3881*, and *#505*).

29) Previously, attempting to restart a run when the TSTEP and DATES keywords caused a non-monotonic schedule section, caused the simulator to abort - this has now been addressed. Secondly, if the SKIPREST keyword was missing from a restart run input deck, then the simulator would abort, again this has also been fixed (*#3035*).

30) The saturation pressures arrays, PBUB and PDEW, were previously written to the restart file using SI units instead of the pressure units for the model. This has now been fixed (*#4133*).

31) The command line parameter *enable-opm-rst-file=true*, allows for additional solution arrays to be written to the restart file, including the fluid viscosity arrays. However, WAT_VISC array data contained the gas viscosity data instead of the water viscosity data (issue *#4245*). This has now been corrected (*#4249* and *#4256*).

    Thanks to Edmund Stephens for discovering the issue and providing an example model for analyzing the underlying issue.

32) If WELLDIMS(MXLIST) was set to zero in the RUNSPEC section, instead of one or 1*, this resulted in the simulator throwing an exception when attempting to write out the restart file. This has now been rectified (*#3195*).

33) There was a bug in the Salt Precipitation Model in how the solid salt accumulation term was calculated. This has now been corrected (*#716*).

34) Previously, the simulator would not run using the TEMP keyword in single phase water runs, this has now been fixed (*#3068*).

35) There was a bug associated with how the simulator calculated time on some operating systems, namely BSD, when the START date was 1 JAN 1900, or earlier. This has now be rectified (*#3101*).

36) There was a bug in the Two-Point Flux Approximation ("TPFA") version of local linearizer code that prevent it working for two-phase runs. This has now been corrected (*#713*).

37) Fixed a bug with the newly implemented TpfaLinearizer for when the MULTFLT keyword is used in the SCHEDULE section (*#750*, *#4232*, *#4207* and *#748*).

38) A bug with the TpfaLinearizer in parallel restarts caused the simulator to abort. This has now been fixed (*#4044*).

39) There was a bug associated with the *-use-gmres* command line option that requested that the Generalized Minimal Residual solver be used instead of the default Biconjugate Gradient Stabilized linear solver within the Newton iterations. The bug resulted in the command line option being ignored, this has now been fixed (*#4242* and *#4254*).

40) For models that only contain water, if the summary vector WBHP was requested, then the simulator would abort. This was because it was assumed that the oil phase was present when calculating WBP. This has now been fixed such that WBP is calculate based on the phases present (oil, gas and water). See *#3971* and issue *#3970* which is now closed.

41) For output of pore volume data, commit *6d3da3d* (PR *#3397*) introduced the notion of a "dynamic", pressure dependent pore volume and switched the PoreVolume aggregates to use reference conditions, for output to the PORV field in the *.PRT file. However, we failed to update all existing uses of the PoreVolume which introduced an inconsistency. In particular, for simulation models without hydrocarbons, for example in single-phase water runs, the numerator of the volume weighted average would include pressure effects through the rock compressibility, but the denominator would not. This has now been corrected and resolves issue *#3984*.

    Thanks to Edmund Stephens for discovering the issue and providing an example model for analyzing the underlying issue.

42) If the WELLDIMS(MXWSLIST) parameter was manually defaulted or set to zero, then the simulator would throw an exception when attempting to write to the restart file (see issue *#3194* for details). This has now been fixed (*#3210* and *#3195*).

43) In a gas-water system sometimes negative water saturations in the grid cells occurred, as per *OPM/opm-simulators#4002*. This has now been corrected (*#707*).

44) Fixed a convergence issue in OPM Flow's Water Vaporization and Salt Precipitation models when water disappears in a cell due to water evaporation (*#734*).

45) When a well event occurs, for example opening or closing or changing control mode, we need to update the well state to match the new control type. In addition, we have to update the primary variables before attempting to solve for the wells. Previously, the primary well variables were not updated, this is now done correctly (*#3907*).

46) Fixed an issue with the way the code handled well connections (*#3974*).

47) There was a couple of issues with the well connections in parallel runs that have now been corrected, together with some refactoring of the code base (*#3104*).

48) Wells can either can either be STOPPED, allowing for potential cross-flow down-hole if requested by the WELSPECS(XFLOW) parameter, or SHUT for being isolated from the reservoir cells. However, STOPPED wells without cross-flow is inconsistent and results in numerical problems. Thus, when wells are declared as STOPPED without cross-flow, they are now treated as SHUT, the same as the commercial simulator (*#3934*).

49) When computing the sum of rates etc. over all well perforations, we need to cater for the case of distributed wells; that is wells present on more than one domain in a parallel run, i.e. we need to also sum up over all processes involved after performing the local sum. However, one of these global sums was missing in the function computeWellConnectionDensitesPressures for producers when we computed the weights based on well transmissibilities. This has now been fixed (*#3976*). Note that distributed wells by default are not allowed, unless requested via setting the command line parameter --allow-distributed-wells equal to true.

50) In parallel runs, the number of local well perforations was not always correctly communicated between processes, especially if the distributed well had no perforations. This resulted in the simulator crashing. This has now been corrected (*#3983*).

51) Previously, the WECON keyword's economic limits were applied to both producers and injectors. This was incorrect, as the economic limits should only be applied to producers, this has now been rectified (*#4151*).

52) Fixed a bug in the multi-segment well model that resulted in non-finite values (*#534*).

53) For multi-segment wells using the WSEGVALV keyword to define a well segment to have a sub-critical valve ICD as part of a completion, if the value was shut then the rate and pressure drop for the value should be zero. However, this was not always the case. This has now been fixed by ensuring that rates and pressure drop for the valve are indeed zero for when the value is shut (*#3964*).

54) There was a potential sign error for the total flow rate for the top segment in multi-segment wells which has now been fixed (*#4110*).

55) Added the fluid flow velocity unit for multi-segment wells, required for exporting the data to the *.SUMMARY and *.RFT files (#3178).

56) Fixed a bug introduced with 06a8b0e in #3937 for standard wells that cause convergence issues on the GPU, for both cusparse and opencl implementations (#3958).

57) The error message for when VFPPROD tables had non-monotonic entries had the table number and the number of errors reversed. This output discrepancy has now been rectified (#3152). In addition, allowed the extrapolation of the tables to a greater range to improve the table lookup (#4098).

58) There was an issue with injection wells when the injection rate was set to zero for wells that were open to flow. This has now been fixed (#4116).

59) Fixed a bug with the WPIMULT keyword for not handling multiple WPIMULT keywords in the same time step correctly, as well as correctly defining the default values for the keyword as being any negative integer (#3067).

60) Fixed a regression bug with the WPIMULT keyword being used in an ACTIONX block (#3092).

61) Corrected an issue when using the WVFPEXP keyword in two-phase runs (#4200 and #4231).

B.3.5 KNOWN ISSUES

1) There is an issue with the GCONPROD keyword when GCONPROD(TARGET) is set to LIQ and both phases are not present in the input deck. This will cause the simulator to abort. The work around is to set GCONPROD(TARGET) equal to a phase that is present in the deck. See #3970 for a discussion.

2) There is an error with the RSM header for summary vectors whose NUMS entry in the SMSPEC file is derived from more than a single number source (e.g., single region or segment ID). This applies to all block vectors (BGPV, BOPV, BWPV, etc.), connection level quantities (COPT, etc.), and inter-region flows such as ROFT etc. The work around is to plot the data in OPM ResInsight and right-click on the plot to view and copy the data. See issue #3078.

3) OPM Flow does not support using LIQ as a well's preferred phase with the WELSPECS. keyword, that is WELSPECS(TYPE) equals LIQ. This is a long-standing bug/omission in the simulator stemming from a somewhat naive internal notion of phases so we don't have an entry for a liquid phase, only for the distinct oil and water phases. For producing wells this mostly matters if you plot the WPI summary vector (productivity index for well's preferred phase). In the current treatment WPI will not have contributions from the water phase if the declared preferred phase is LIQ. For injecting wells WELSPECS's preferred phase doesn't really matter at all since the preferred phase is (typically) reset to the injected phase in WCONINJE/WCONINJH anyway. See issue #3075.

4) If the simulator finds well connection being declared as connections via the COMPDATA keyword in the SCHEDULE section, then it writes out a warning message

```
Warning: Problem with COMPDAT keyword
In SIM_PEE_Basic_Simulator.DATA line 535
The cell (0,6,4) in well INJ1 is not active and the connection will be ignored

Warning: Problem with COMPDAT keyword
In SIM_PEE_Basic_Simulator.DATA line 535
The cell (16,0,4) in well INJ2 is not active and the connection will be ignored
```

However, the reported cell references are offset by minus one, meaning the correct warning messages should be:

```
Warning: Problem with COMPDAT keyword
In SIM_PEE_Basic_Simulator.DATA line 535
The cell (1,7,5) in well INJ1 is not active and the connection will be ignored

Warning: Problem with COMPDAT keyword
In SIM_PEE_Basic_Simulator.DATA line 535
The cell (17,1,5) in well INJ2 is not active and the connection will be ignored
```

See issue *#3167* for details.

5) Pull request *#3192* fixed an incorrect JSON definition of the GASVISCT keyword in the PROPS section, that prevented a thermal model from running. However, thermal models containing the GASVISCT keyword will still not run, and instead will write out the following error message:

```
Error:
An error occurred while creating the reservoir properties
Internal error: Tried to get back() from empty DeckView

Error: Unrecoverable errors while loading input: Tried to get back() from empty
DeckView.
```

Commenting out the keyword will enable the case to run. See issue *#3200* for details.

6) The tracer related output to summary file does currently not respect the command line variable --enable-tracer-mode, this means we may have the situation in which the input deck specifies tracer output, and the simulator is run without enabling the tracer model (enable-tracer-mode=false). In this scenario the trace SUMMARY vector output will consists of irrelevant tracer values.

7) The GDFILE keyword in the GRID section loads a grid file in various formats, with the FMTOPT parameter setting the format type of the file. If the variable FMTOPT is omitted then the default is for binary file input for the commercial simulator; whereas, OPM Flow derives FMTOPT from the file extension (*.EGRID or *.FEGRID), making FMTOPT superfluous. However, if the extension is lower case then OPM Flow may incorrectly determine the file type. The work around is change the extension to upper case.

8) As per previous releases of the radial model, the COORDSYS keyword item three must be set to COMP to complete the circle, this has not been implemented in this release. Also there appears to be a bug for full radial models when a well goes on BHP control that causes the well not to respect the BHP constraint, this eventually causes the well to die prematurely. See *#2640* for a discussion on the topic.

9) As in previous releases there are some issues with the OPERATE and OPERATER keywords associated with the input parsing; for various reasons a few of the fields require special case treatment in the grid processing, including (at least) MULTZ, PORV and ACTNUM, and for those keywords the OPERATE/OPERATER keyword doe not work. The work around is to use the MULTIPLY keyword instead.

10) For the UDQ ASSIGN operator after the terminating "/" normally any comments can be entered; however, if there is "/" within the comment field, as per:

```
ASSIGN FUNGLYLD 1.196   /    Condensate Yield (63.5/56.7)/(1.0 – 0.065)
```

then the simulator will abort. The work around is to manually place the comment characters "--" after the ASSIGN terminating "/", like so:

```
ASSIGN FUNGLYLD 1.196   / -- Condensate Yield (63.5/56.7)/(1.0 – 0.06)
```

11) At the moment, one cannot initialize tracers using the EQUALS keyword. Instead use the array format, that is the keyword followed by the required number of values, or the TVDP keyword in the SOLUTION section to set the initial tracer concentrations as a function of depth.

12) Currently, gas tracers cannot be used if the dissolved gas phase, as per the DISGAS keyword in the RUNSPEC section, is active in the model.

13) There is a potential issue with left-handed grids that may cause the simulator to stop due to how the check for left-handed versus right-handed coordinate systems is performed in the corner-point processing code, which implicitly assumes that all pillars have well-defined, unique top-point coordinates. However, in some cases the pillar top points may be at the same coordinate location if there are no active cells along a particular pillar. Secondly, although OPM ResInsight may load the input grid, the display will be incorrect. Currently there is no work around for this except for re-generating the grid from the static model (*#3896*).

14) In some cases when the program stops the error message is written to the terminal but not to the *.PRT or *DBG files. This is because the failing code is not aware of the C++ logging system. For example, this will occur if there is an error in parsing the grid data as the corner-point processing code is written in C and at the moment cannot call the C++ logging system. (*#3896*).

15) The summary vector RTIPTHEA, that defines the energy in-place between the initial and the current time for regions, is not supported unlike the FTIPTHEA and BTIPTHEA vectors. Secondly, the error message:

```
Warning: Problem with summary keyword RTIPTHEA
In RSM-THERMAL.data line 492
FIP region FIPHEA not defined in REGIONS section - RTIPTHEA ignored
```

is incorrect, as the message indicates that it is being treated like a named region, as per the FIP keyword, when it is actually a SUMMARY vector (*#3870*).

16) If there are cells that are very distorted, which can occur near fault planes, then the simulator may abort because it cannot calculate the pore volume of such cells. The work around is to re-generate the grid in the static model, taking care that the cells around the fault planes are more or less orthogonal (*#2992* and *#3770*).

17) Currently the OPERATOR keyword in the EDIT section does not work with the DEPTH, TRANX, TRANY and TRANZ property arrays (*#2994* and *#748*).

18) If a standard well is fully declared in an ACTIONX block which is then activated at a later date, and later the well is modified to be a multi-segment well using the WELSEGS and COMPSEGS keywords, then this will cause the simulator to abort with an assert failure. The solution to this issue is to not use this type of work flow in declaring wells (*#2891* and *#2895*).

19) Although the ACTIONX EXIT command works as expected, it does not write out the requested RSM file at the end of the run. However, the other SUMMARY and RESTART files are written out (*#2877*).

20) Although the GCONSUMP keyword in the SCHEDULE section is fully implemented as documented, it is not possible to verify the output as the associated SUMMARY vectors are not written out, that is the SUMMARY sales gas vectors FGSR, FGST, GGSR and GGST, and fuel vectors FGCR, FGCT, GGCR, and GGCT have not been implemented (*#2679*).

21) There are small differences in the behavior of the NEXTSTEP keyword in the RUNSPEC section between OPM Flow and the commercial simulator that remain unresolved (*#3745*).

Jostein Alvestad, Kai Bao, David Baxendale, Markus Blatt, Paul Egberts, Arne Morten Kvarving, David Landa-Marbán, Cintia Goncalves Machado, Halvor Møll Nilsen, Atgeirr Flø Rasmussen (Release Manager), Alf Birger Rustad, Tor Harald Sandve, Bård Skaflestad, Torbjørn Skille, and Pieter J. Verveer.

B.4 RELEASE 2022-04

The 2022-04 release consists of some new features and various improvements and bug fixes. Highlights for this release include: implementation of the Salt Precipitation Model, several ACTIONX implementation improvements – including support for the COMPSEGS, WELSEGS and WSGVALV keywords in an ACTIONX block, improvements to the CO2STORE model to work with numerical aquifers and to account for thermal effects, support for gas lift optimization for multi-lateral wells, the addition of various SUMMARY vectors, plus various enhancements to the RESTART file to improve compatibility and robustness. Significant work has also been conducted in fixing the number of reported bugs.

When building OPM Flow from source, the default is now be to compile a parallel binary if MPI is installed, this was already the default for the binary packages that were previously distributed. Secondly, support for building OPM Flow with the new current version of DUNE (2.8) has been added.

B.4.1 NEW AND DEPRECATED COMMAND LINE OPTIONS

The major command line changes made for this release are summarized in Table B.7

OPM Flow 2022-04 New And Deprecated Command Line Options			
No.	**Variable Name**	**Description**	**Default**
1	--sched-restart	A Boolean value that determines for a restart run if the case should initialize wells and groups from the historical SCHEDULE section (true), or from the well and group data on the restart file (false). Note that the commercial simulator always uses data from the restart file (false).	false
2	--enable-tracer-model	A Boolean value set to true or false that turns on (true) or off (false) transport tracer calculations for when tracers have been declared in the input deck. This option is deprecated (#3717).	
Notes: 1) Cells colored in green in the "No." column indicate a new command line option or a change to an existing option for this release. 2) Cells colored in red in the "No." column indicate a deprecated command line option for this release.			

Table B.7: OPM Flow 2022-04 New and Deprecated Command Line Options

B.4.2 NEW FEATURES

In addition to the above the following new features have been added to the simulator:

1) The Python version of the opm-parser is used in a number of scripts by some users for parsing specific parts of a data deck. The efficiency of the scripts are improved as the Python version is now aware which section a keyword belongs to and can ignore sections when getting the requested keyword(s). This feature was added in the 2021-10 release but was not documented for that release (#2527).
2) ACTIONX implementation improvements include:
   a) Support for the MULTX, MULTX-, MULTY, MULTY-, MULTZ and MULTZ- keywords in an ACTIONX block (#3686).
   b) Implemented support to access grid property data to enable the COMPDAT, COMPLUMP, etc., type of keywords to work within an ACTIONX bloc (#2828).

c) Added support for the COMPSEGS, WELSEGS and WSGVALV keywords within an ACTIONX block (#2849, #2852, and #2885).

d) Added support for the WELTARG keyword within an ACTIONX block (#2892).

e) Added support for the WTMULT keyword within an ACTIONX block (#2906).

f) Added a missing ACTIONX comparison (#2898).

g) Implemented support for the NEXTSTEP keyword in the SCHEDULE section (#3614, #3721 and #3736) as well as support in an ACTIONX block (#2904).

h) Added the NEXT keyword in the SCHEDULE section as an alias for NEXTSTEP (#2902) as well as support for NEXT in an ACTIONX block (#2905).

3) Added support for the CO2STORE model to use numerical aquifers (#3711).

4) Enabled the CO2STORE model to account for thermal effects (#3724).

5) Added support for the GRAVITY keyword in the PROPS section (#3670).

6) RESTART file improvements:

a) Added support for the Extended Network Model (#2857, #2825, #2809, #2808, #2779 and #2804).

b) Added support for reading and writing TRACER data to the RESTART file (#2869, #2861, #2864, #2859, #2855, #3702, #2854, #2879, #3708 and #3718).

c) Added additional entries for well connections (#2427) and improved error reporting when loading RESTART files (#2759).

d) Load and save GCONINJE guide rate injection settings for group water injection to and from the RESTART file (#2888).

e) Read and write GLIFTOPT options to the RESTART file (#2794 and #2777), load and store WLIFTOPT options if enabled for a well (#2791 and #2770).

f) Load and save group efficiency factor values to and from the RESTART file (#2773).

g) Added support for storing and loading group constraints at the field level set by the GCONPROD and GCONINJE keywords (#2814).

7) Added support for the WTMULT keyword in the SCHEDULE section (#2763).

8) Implemented the Salt Precipitation Model. This is an extension of the brine module to deal with salt precipitation/dissolution and allowing primary variable switching between salt concentration and solid(precipitated) see #3729, #3759, #681 and #490. In addition the following keywords have been added and/or updated:

a) The SALTSOL keyword in the PROPS section now supports setting the salt solubility by region (#2593). Note this is an OPM Flow specific keyword used with the simulator's Salt Precipitation model.

b) Added the SALTPVD keyword as part of OPM Flow's Salt Precipitation Model that defines the initial precipitated salt volume fraction versus depth tables for each equilibration region (#2920).

c) Implemented PRECSALT keyword in the RUNSPEC section that activates OPM Flow's Salt Precipitation Model (#2595).

d) Fixed a unit error with the units for salt concentration on the SALTVD keyword changing the units for the SALTCON parameter from lb/ft3 to the correct units of lb/stb (#2930).

e) Added the SALT keyword to define the initial equilibration salt concentration for all grid blocks and the SALTP keyword to define initial equilibration precipitated salt volume fraction (#2931 and #3774).

f) Added the RVW keyword in the SOLUTION section that defines the initial equilibration vaporized water in gas ratio for all grid blocks for use with the Vaporized Water Model (#2968, #3830, #684 and #491).

9) Added support for gas lift optimization for multi-segment wells (#3812).

10) Added support for gas-water initialization for two-phase runs using PVDG and PVTW keywords in the PROPS section. Two-phase gas-water models will now run; however, the RVCONST and the RVCONSTT keywords are currently unsupported and thus the liquid yields have to be calculated manually (#3838).

11) Added support for the WVFPEXP keyword in the SCHEDULE section (#2988 and #3866).

12) Added support for LIFTOPT(TSTEP) option that defines the frequency of the gas lift optimization calculations. This was previously documented as implemented but was not. The functionality has now been implemented in this release (#3777).

13) Previously, the water only and water only with thermal models were standalone binaries. With this release these models are now incorporated into the main OPM Flow binary (#3892 and #3891).

14) The VAPWAT keyword in the RUNSPEC section, that activates the vaporized water phase, is now active for this release for gas-water systems only. Work on the three phase case is currently under way and is expected to be implemented in the next release (#2989 and #2591).

In addition, the following new SUMMARY keywords are now recognized as described by the comments in Table B.8.

No.	Summary Keyword	Comment
1	BGPR and BWPR	Added support for block-level gas (BGPR) and water (BWPR) phase pressures(#3827 and #2967).
2	BNSAT	Implemented block level solvent saturation summary output keyword BNSAT (#3861 and #2986).
3	BTCNFHEA	Added support for the block temperature vector (#2962 and #3820).
4	FSIR, FSIT FSPR, FSPT, FSPC, and FSIP	Added support for salt production rates for the OPM Flow's Salt Precipitation model.
5	GSIR, GSIT, GSPR, and GSPC	Added support for salt production rates for the OPM Flow's Salt Precipitation model. Not GSPT is currently not supported.
6	GWIGR	Previously the simulator would tie reporting of group level Guide Rates to whether or not a group had any guide rates for production; however, this is insufficient for group level injection guide rates. OPM Flow has now been updated to handle group level water injection guide rate reporting on the SUMMARY file via the GWIGR variable (#3612 and #3598).
7	MSUMBUG MSUMCOMM MSUMERR MSUMMESS MSUMPROB MSUMWARN	In the 2021-04 release the message keywords were recognized by the parser; however, no data associated with these vectors is written to file (#2234). This is still the case.

No.	Summary Keyword	Comment
8	ELAPSED MLINEARS MSUMLINS MSUMNEWT NEWTON NLINEARS NLINSMAX NLINSMIN, STEPTYPE TCPU TCPUDAY TCPUTS TELAPLIN TIMESTEP	Various performance vectors are now supported either declared individually or via the PERFORMA keyword in the SUMMARY section. The ELASPSED, STEPTYPE, TCPUDAY, TCPUTS and TELAPLIN vectors are written out but all values are set to zero. See #2953 and #3801.
9	R*FR and R*FT	Added support for the following inter-region flow summary vectors: RGFR, RGFR+, RGFR-, RGFT, RGFT+, RGFT, RGFTG, RGFTL, ROFR, ROFR+, ROFR-, ROFT, ROFT+, ROFT, ROFTG, ROFTL, RWFR, RWFR+, RWFR-, RWFT, RWFT+, and RWFT- Note that variables ending in a + or – are OPM Flow's implementation of the commercial compositional simulator's variables (#3811, #3796, #2958, #2945, #2955, #2929, and #2928).
10	WSIR, WSIT WSPR, WSPT, and WSPC	Added support for salt production rates for the OPM Flow's Salt Precipitation model.
11	WSTAT	Added support for the well status summary variable (#2853).
12	WWIRT	In the 2021-04 release support for the WWIRT vector for writing out the water injection rate target was implemented (#2071); however, the field (FWIRT) and Group (GWIRT) vectors were not implemented. A warning message is printed stating the fact.

Table B.8: New SUMMARY Keywords for the 2022-04 Release

B.4.3 IMPROVEMENTS

Improvements include:

1) Improve ESmry file output by not writing out empty elements (#2915).
2) Improved how the summary keywords are handled for inter-region summary vectors in ESMRY file, to give a consistent naming convention (#2960).
3) If the simulator fails to find the a cell's saturation pressure (bubble and dew point pressure) then the following message was printed:

```
Finding the dew point pressure failed for 2 cells [1467066, 1467066]
Finding the dew point pressure failed for 8 cells [1467063, 1467063, 1467066,
1467066, 1467066, 1467066, 1467066, 1467066]
```

This has now be changed to be more readable to:

```
Finding the dew point pressure failed for 1 cell [(71,89,46)]
Finding the dew point pressure failed for 2 cells [(68,89,46), (71,89,46)]
```

See #3828.

4) Improvements on how the well potentials are calculated when the wells are under guide rate control (#3719).

5) Improvements on how the well rates are calculated for when the wells are or not under group control control (#3645).

B.4.4 BUG FIXES

The following bug fixes and improvements have been incorporated into this release.

1) In models with numerical aquifers, the numerical aquifer cells have large pore volumes which unduly influence the convergence criteria, indicating convergence has been achieved when in fact convergence has not been reached. This has now been corrected (#3638).

2) For numerical aquifers, the cell permeability used to compute aquifer to reservoir transmissibilities, did not take into account the MULTX, MULTY and MULTZ values applied to the reservoir cell; these are now taken into account when calculating the aquifer to reservoir transmissibilities (#2919).

3) ACTIONX bug fixes include:

   a) Fixed a bug associated with the ACTIONX keyword when a well is fully defined in an ACTIONX block and then referenced afterwards in the main deck (#2895).

   b) Fixed a bug associated with the ACTIONX keyword when a standard well is fully defined in the main deck and then converted to a multi-segment well in an ACTIONX block (#2895).

4) Corrected a bug in the C02STORE model associated with the reference oil density (#3727).

5) Gas lift and gas lift optimization issues corrected include:

   a) Fixed an incorrect gas lift rate calculation based on IGLR and ALQ (#2807).

   b) Corrected an issue with applying gas, oil and water reduction calculation in the gas lift optimization calculation procedure (#3678 and #3673).

   c) Fixed an issue when both well and group controls restrict gas lift availability (#3728).

   d) Previously, not all groups were considered when reducing oil rates to satisfy oil group limits. This has now been corrected (#3747).

   e) Defaulting GLIFOPTT(MAXLIFT) or GLIFTOPT(MXGAS) would cause the simulator to throw an error, this has now been rectified (#2884).

   f) The VFPPROD(ALQ) default value of 1* is " " or undefined, that covers the case when the ALQ variable is not entered, except for when gas lift is employed in the model. When gas lift is active then the default value for ALQ is set to GRAT. Secondly, it is possible to have only the OIL and WATER keywords in the RUNSPEC section and to use gas lift for the wells, without declaring the GAS phase in the RUNSPEC section. If the ALQ and VFPPROD(ALQ-DATA) parameters are absent then the VFPPROD(GFR-DATA) data will be used based on the flowing GOR plus the stipulated gas lift gas. This logic was previously missing from the simulator but is now implemented (#2971, #3833 and #3706).

   g) Ensure that group/well guide rates (i.e. the distribution between wells/groups) are updated when gas lift volumes vary during iterations, as defined by LIFTOPT(OPTLIFT) set to YES. This will also reduce the number of oscillations during the Newton iterations (#3807).

   h) If a well is under a group that is limited by a target, it should use as little gas lift gas as possible. The reduction algorithm will now reduce the gas lift of the well as long as the groups potential is above the groups target (#3795).

i) Fixed a bug associated with a group's gas lift artificial lift gas rate plus actual gas production being greater than the group's maximum gas rate as declared via the MXGAS parameter on the GLIFTOPT keyword (#3797).

6) Using the GRAVITY keyword in the PROPS section caused an exception to be raised and the simulator to abort. This has been fixed by implementing the GRAVITY keyword (#3670).

7) Fixed a bug associated with the default values on the NETBALAN keyword in the SCHEDULE section.

8) Added support for PVT extrapolation in the extended black-oil model. Previously, if the required values were outside the values supplied by the input keywords, then the simulator raised an exception and would abort (#680).

9) RESTART file bug fixes:

   a) Fixed an issue with the field, group, and well level cumulative production curves of solution/free oil and gas not being continuous in a restarted simulation run (#2871).

   b) If a well's BHP reference depth is defaulted and there are no connections for the well then the default missing value of -1.0E=20 should be written to the RESTART file, previously this was not the case (#2908). This also resolves issue #2442.

   c) Fixed a bug related to writing out tracer data to the restart file (#2878).

   d) For a restart run, previously the RPTRST keyword in the SOLUTION section was not retained for the run. This has now been rectified (#2876).

   e) The OPM Flow specific PORV_RC and TMULT_RC solution arrays were incorrectly marked as commercial simulator compatible arrays, this would cause errors when loading the RESTART file using the commercial simulator. This has now been fixed (#3683).

   f) Previously, the cumulative oil and water volumes were set to zero when restarting a simulation. This has now been rectified by using the correct volumes (#2805).

   g) Fixed an issue with loading LIFTOPT from the RESTART file (#2790).

   h) For the restart group, well and connection data the XWEL keyword items 36 to 39 need to be defined (see Error: Reference source not found). The current understanding is that the values are same as items one through three on the same keyword. Secondly, exponent letters need to uppercase (1.26E-04) and not lowercase (1.26e-04). Both issues have been address in #2975.

10) Previously, the RPTSCHED Well Production and Injection reports printed zeros for the field rates and totals, this has now been rectified (#2584 and #2880).

11) In parallel runs, applying transmissibility multipliers in the SCHEDULE section would cause an exception to be raised and the simulator to abort. This has now been fixed (#2923).

12) SUMMARY output bug fixes:

   a) Fixed an issue with the surface gas production volumes having incorrect field units for free and associated (dissolved) gas. This error was specific to runs using field units as opposed to metric units (#2918).

   b) Fixed a bug in parallel runs which caused the SUMMARY vectors associated with the free gas, dissolved gas, and the oil and vaporized oil vectors for all objects (field, well, etc.) to be incorrect due to the dissolved phase being under reported. This also effected the free phase volumes as well as Free (Oil, Gas) is calculated from Total (Oil, Gas) - Dissolved (Oil, Gas). Now both sequential and parallel runs give the same results (#3765).

13) Incorporated well efficiency, via the WEFAC keyword in the SCHEDULE section, in network calculations. Note that only the default value for WEFAC(WELNETWK) of YES is supported for the Extended Network Model (#3730).

14) Fixed WELTARG not supporting the LIFT option (#2881).

15) Corrected an issue with how wells are handled for the WECON keyword (#3720).

16) If the WRFT keyword was used to report RFT data to the WRFT file and the well had no connections then OPM Flow would write the records to the file. This would cause an issue with OPM ResInsight and other post-processing software attempting to read the resulting WRFT file. OPM Flow no longer writes the records if there are no connections to the well (#2909).

17) Using the WTEST keyword to test physically or economically shut-in wells, OPM Flow previously did not check if a well had been manually shut-in via the input deck. This has been corrected such that wells manually shut-in will now not longer be tested (#3713).

18) Fixed a bug associated wells prematurely closing due to negative flow potentials (#3834).

19) When converting from surface to reservoir rates the simulator needs to solve a 2x2 system of equations using Cramer's Rule, that is d = 1-rs*rv, where d is the determinant. If d is zero the system is singular and the system unsolvable. This should not happen with physical values of rs and rv, but for unconverged solutions this may happen due to extrapolation of rs/rv tables to unphysical pressures. The fix removes a hard throw and instead ignores the dissolved/vaporized part and tries to continue. Some times this is sufficient to get a more reasonable solution in the next Newton iteration, other times not, and the simulator needs to chop the time step, etc. (#3853 and #3855).

20) For the WELSPECS keyword the GRPNAME parameter should not be set to FIELD; however, this was not trapped and resulted in a program exception. This has been rectified by issuing an error message for when GRPNAME is set to FIELD (#2974).

21) When multi-segment wells were initially opened the oil and water rates were zero after being scaled because the perforation rates entering calculateSegmentRates routine only had a value for the gas rate and zero for the water and oil rates. This has now been corrected by ensuring the top segment rates will be approximately the same with the well rates after scaling from zero rates from opening the well (#3839).

22) Previously wells would per-maturely close due to the economic limits set via the WECON keyword because of inaccurate well potentials and inaccurate ratios (WOR, GOR, etc) caused by negative rates. This has now been corrected (#3851).

23) When multi-segment wells are initially opened the well rates and well potentials are zero causing a numerical issue due to creating a singular matrix. This fix ensures we have sensible initial rates etc. for the wells (#3847).

24) For multi-segment wells changed from multiplicative to addition when updating the pressure for each segment when the bottom-hole pressure is changed. This is a more robust treatment (#3835).

25) If there are no analytical aquifer connections, for example, if the AQUANCON keyword has not been entered or that all the defined connections are invalid, then the simulator would stop. This is still the case but now a more descriptive error message is printed (#2973).

26) If there are no cells with valid corner-point geometry, typically caused by using GDFILE to read non-finite data such as all ZCORN = -1.0E+20, then the simulator will throw an exception. Now instead an error message is issued and the simulator terminates gracefully (#3845 and #2979).

27) Fixed an issue with being unable to calculate a well's BHP from its THP limit (#3760).

28) Under some circumstances a well's BHP constraint may be violated during an iteration cycle, for example, when an injector is opened to give pressure support to the producers it may take a few Newton iterations for the pressure support to be propagated and the solution to converge. Before convergence is achieved the simulator may calculate negative potentials resulting in the well being shut-in. This has now been rectified (#3834).

29) Check if group and well guide rates have been violated and also ensure that a group's efficiency is accounted for when summing guide rates (#3814).

30) When a restarted model uses SKIPREST, there must be either a DATES or TSTEP keywords that correspond exactly to the requested restart time. The previous diagnostic message for:

    missing record in DATES

    ```
    Error: Problem with keyword DATES
    In CASE.DATA line 223
    At date: 2019-04-18 - scanned past restart data: 2019-04-12
    ```

    and missing report step in TSTEP

    ```
    Error: Problem with keyword TSTEP
    In PRED_FLOW.DATA line 219
    At date: 2019-04-16 - scanned past restart data: 2019-04-12
    ```

    did not offer sufficient information to the user as to what the underlying issue might be. The new message are now for missing record in DATES:

    ```
    Error: Problem with keyword DATES
    In CASE.DATA line 223
    In a restarted simulation using SKIPREST, the DATES keyword must have
    a record corresponding to the RESTART time 12-Apr-2019 00:00:00.
    Reached time 18-Apr-2019 00:00:00 without an intervening record.
    ```

    and missing report step in TSTEP:

    ```
    Error: Problem with keyword TSTEP
    In CASE.DATA line 219
    In a restarted simulation using SKIPREST, the TSTEP keyword must have
    a report step corresponding to the RESTART time 12-Apr-2019 00:00:00.
    Reached time 16-Apr-2019 00:00:00 without an intervening report step.
    ```

31) If a well has been declared as SHUT via the WCONINJE or WCONPROD keywords but is later activated using the WELOPEN, keyword we must record this status change in the well's internal has_produced or has_injected flags as appropriate. Otherwise, the counts of "abandoned" wells (summary vectors [FG]MW[IP]A) will not be correct (*#2961*).

32) If a well under group control is constrained such that the well's THP is below the well's THP limit, then avoid switching to THP control if this results in production or injection rates increasing. For example, if the well's group has a zero rate then we do not want to switch the well control to THP control as this will increase the well's production or injection rate (*#3824*).

33) If an injection well had an insignificant injection rate then the commercial simulator returns zero for the THP regardless of the BHP. OPM Flow now does the same (*#3773*).

34) Fixed an issue with group and well guide rates not being updated after new wells are opened up (*#3771*).

35) In some rare circumstances the WOR/WCT/GOR/GLR limits may be exceeded at the well level but not at any of the individual connections. This can occur due to numerical noise caused by very small rates. If this occurs the simulator will now not shut the well (*#3758*).

36) Previously, the WTEST code only checked the BHP constraints and potentially opened wells limited by a THP constraint. This has now been addressed by checking for both BHP and THP constraints before opening wells using the WTEST facility. (*#3803* and *#3816*).

37) If the well residuals contained NaNs (Not a Number) this was counted as an error, now the event is more correctly recorded as unconverged. The simulator behavior remains unchanged (*#3867*).

38) Fix a bug associated with the command line -shut-unsolvable-wells equals true option. The bug did not effect the default behavior (*#3863*).

39) Fixed an error associated with the saturated density in the DRSDTCON keyword (#3884 and #3878).

40) Added the DR, DTHETA and OUTRAD keywords as unsupported keywords with messages to enable the user to debug the input deck (#3875 and #3883).

41) Fixed an issue with checking well potentials, now OPM Flow only checks if wells have negative potentials if the command line enable-well-operabilty-check=true, which is the default behavior (#3874 and #3885).

42) Fixed an issue with initialization when there is no dissolved gas in the model (#3871 and #3877).

43) Stopped wells are wells that are shut-in at the surface and are operable in order to allow cross-flow in the wellbore, as opposed to wells that are shut-in at the sand face. Now stopped wells are changed to shut -in if they cannot converge after repeated time-step chopping. The change makes the well modeling code more robust (#3890 and #3872).

44) Fixed an error associated with Brine model accessing an uninitialized variable if the enableSaltPrecipitation Boolean variable is set to false, that is salt precipitation is not enabled (#691 and #690).

45) Fixed an issue with the COMPLUMP keyword with the indices I, J, K1 and K2 not having the default value of zero (#3006 and #3011).

B.4.5 KNOWN ISSUES

1) The tracer related output to summary file does currently not respect the command line variable -enable-tracer-mode, this means we may have the situation in which the input deck specifies tracer output, and the simulator is run without enabling the tracer model (enable-tracer-mode=false). In this scenario the trace SUMMARY vector output will consists of irrelevant tracer values.

2) The GDFILE keyword in the GRID section loads a grid file in various formats, with the FMTOPT parameter setting the format type of the file. If the variable FMTOPT is omitted then the default is for binary file input for the commercial simulator; whereas, OPM Flow derives FMTOPT from the file extension (*.EGRID or *.FEGRID), making FMTOPT superfluous. However, if the extension is lower case then OPM Flow may incorrectly determine the file type. The work around is change the extension to upper case.

3) As per previous releases of the radial model, the COORDSYS keyword item three must be set to COMP to complete the circle, this has not been implemented in this release. Also there appears to be a bug for full radial models when a well goes on BHP control that causes the well not to respect the BHP constraint, this eventually causes the well to die prematurely. See #2640 for a discussion on the topic.

4) As in previous releases there are some issues with the OPERATE and OPERATER keywords associated with the input parsing; for various reasons a few of the fields require special case treatment in the grid processing, including (at least) MULTZ, PORV and ACTNUM, and for those keywords the OPERATE/OPERATER keyword doe not work.

5) For the UDQ ASSIGN operator after the terminating "/" normally any comments can be entered; however, if there is "/" within the comment field, as per:

```
ASSIGN FUNGLYLD 1.196   /    Condensate Yield (63.5/56.7)/(1.0 – 0.065)
```

then the simulator will abort. The work around is to manually place the comment characters "--" after the ASSIGN terminating "/", like so:

```
ASSIGN FUNGLYLD 1.196   / -- Condensate Yield (63.5/56.7)/(1.0 – 0.06)
```

6) At the moment, one cannot initialize tracers using the EQUALS keyword. Instead use the array format, that is the keyword followed by the required number of values, or the TVDP keyword in the SOLUTION section to set the initial tracer concentrations as a function of depth.

7) Currently, gas tracers cannot be used if the dissolved gas phase, as per the DISGAS keyword in the RUNSPEC section, is active in the model.

8) When using the AQUANCON keyword to declare declare analytical aquifer connections, using for example:

```
--
--                      ANALYTIC AQUIFER CONNECTION
--
--       ID     ---------- BOX ---------   CONNECT  AQF     AQF     ADJOIN
--       NUMBER I1  I2   J1  J2   K1  K2   FACE     INFLX  MULTI   CELLS
AQUANCON
         1      3  54   117  142   1  30   J+       1*     1*       1*   /
         2     83 131    20   48   1  30   J-       1*     1*       1*   /
         2     53  84     2   19   1  30   J-       1*     1*       1*   /
/
```

Then if there are invalid aquifer connections OPM Flow will issue a warning message for each invalid connection. If there are numerous invalid connections because of the irregular shape of the reservoir edges then the simulator will take a long time time to write out all the invalid connections. The work around is to more precisely define the aquifer connection data (*#2993*).

9) There is a potential issue with left-handed grids that may cause the simulator to stop due to how the check for left-handed versus right-handed coordinate systems is performed in the corner-point processing code, which implicitly assumes that all pillars have well-defined, unique top-point coordinates. However, in some cases the pillar top points may be at the same coordinate location if there are no active cells along a particular pillar. Secondly, although OPM ResInsight may load the input grid, the display will be incorrect. Currently there is no work around for this except for re-generating the grid from the static model (*#3896*).

10) In some cases when the program stops the error message is written to the terminal but not to the *.PRT or *DBG files. This is because the failing code is not aware of the C++ logging system. For example, this will occur if there is an error in parsing the grid data as the corner-point processing code is written in C and at the moment cannot call the C++ logging system. (*#3896*).

11) The summary vector RTIPTHEA, that defines the energy in-place between the initial and the current time for regions, is not supported unlike the FTIPTHEA and BTIPTHEA vectors. Secondly, the error message:

```
Warning: Problem with summary keyword RTIPTHEA
In RSM-THERMAL.data line 492
FIP region FIPHEA not defined in REGIONS section - RTIPTHEA ignored
```

is incorrect, as the message indicates that it is being treated like a named region, as per the FIP keyword, when it is actually a SUMMARY vector (*#3870*).

12) If there are cells that are very distorted, which can occur near fault planes, then the simulator may abort because it cannot calculate the pore volume of such cells. The work around is to re-generate the grid in the static model, taking care that the cells around the fault planes are more or less orthogonal (*#2992* and *#3770*).

13) Currently the OPERATOR keyword in the EDIT section does not work with the DEPTH, TRANX, TRANY and TRANZ property arrays (*#2994* and *#748*).

14) If a standard well is fully declared in an ACTIONX block which is then activated at a later date, and later the well is modified to be a multi-segment well using the WELSEGS and COMPSEGS keywords, then this will cause the simulator to abort with an assert failure. The solution to this issue is to not use this type work flow in declaring wells (*#2891* and *#2895*).

15) Although the ACTIONX EXIT command works as expected, it does not write out the requested RSM file at the end of the run. However, the other SUMMARY and RESTART files are written out (*#2877*).

16) Although the GCONSUMP keyword in the SCHEDULE section is fully implemented as documented, it is not possible to verify the output as the associated SUMMARY vectors are not written out, that is the SUMMARY sales gas vectors FGSR, FGST, GGSR and GGST, and fuel vectors FGCR, FGCT, GGCR, and GGCT have not been implemented (*#2679*).

17) There are small differences in the behavior of the NEXTSTEP keyword in the RUNSPEC section between OPM Flow and the commercial simulator that remain unresolved (*#3745*).

Jostein Alvestad, Kai Bao, David Baxendale, Markus Blatt (Release Manager), Paul Egberts, Joakim Hove, Arne Morten Kvarving, David Landa-Marbán, Cintia Goncalves Machado, Atgeirr Flø Rasmussen, Alf Birger Rustad, Tor Harald Sandve, Bård Skaflestad, Torbjørn Skille, and Pieter J. Verveer.

B.5 RELEASE 2021-10

The 2021-10 release consists of some new features and various improvements and bug fixes. Highlights for this release includes support for a new output SUMMARY file type Enhanced SMRY (*.ESMRY), passive tracer support, two phase gas-water support and single phase water support (only for Enumeration Initialization), Carter-Tracy and numerical aquifers support with the CO2 Storage Model, and the implementation of the Microbially Induced Calcite Precipitation ("MICP") model used to investigate CO2 leakage remediation.

B.5.1 NEW AND DEPRECATED COMMAND LINE OPTIONS

The major command line changes made for this release are summarized in Table B.9

OPM Flow 2021-10 New And Deprecated Command Line Options

No.	Variable Name	Description	Default
1	--accelerator-mode	A defined character string that defines the usage of the GPU (cusparseSolver or openclSolver) or FPGA (fpgaSolver) as the linear solver, usage '--accelerator-mode=[none|cusparse|opencl|fpga|amgcl]'.	"none"
2	--enable-esmry	A Boolean value that switches on (true) or off (false) the output SUMMARY vectors to the ESMRY file for fast loading of summary data (*#2590*, *#2606*, *#2609*, *#2621*, and *#3453*).	false
3	--enable-well-operability-check-iter	A Boolean value set to true or false that enables (true) checking of a well's operating status during iterations, or disables (false) the checking during iterations.	false
4	--force-disable-resv-fluid-in-place-output	A Boolean value set to true or false that instructs OPM Flow to not to print the Reservoir Volume Fluid In-Place report after each report time step (true) or not (false).	false
5	--fpga-bitstream	A character string that specifies the bit stream file for the fpgaSolver (including path), usage: '--fpga-bitstream=<filename>'.	""
6	--linsolver	Command line parameter changed from --linear-solver to --lin-solver Functionality is the same as before.	"ilu0"
7	--max-newton-iterations-with-inner-well-iterations	A positive integer that specifies the maximum newton iterations with inner well iterations.	8.0
8	--max-temperature-change	A real positive value that stipulates the maximum absolute change of temperature in a single iteration.	5

OPM Flow 2021-10 New And Deprecated Command Line Options			
No.	**Variable Name**	**Description**	**Default**
9	--maximum-number-of-well-switches	A positive integer values that stipulates the maximum number of times a well can switch to the same control.	3
10	--relaxed-pressure-tol-msw	A real positive value that sets the relaxation tolerance for the multi-segment well pressure solution.	50000
11	--relaxed-well-flow-tol	A real positive value that sets the relaxation tolerance for the well flow residual.	1.0
	-sched-restart	A Boolean value that determines for a restart run if the case should initialize wells and groups from the historical SCHEDULE section (true), or from the well and group data on the restart file (false). Note that the commercial simulator always uses data from the restart file (false). (#3601). This option was also in the previous release.	true
12	--shut-unsolvable-wells	A Boolean value that determines if the simulator should shut unsolvable wells (true), or not (false).	false
13	--solver-continue-on-convergence-failure	A Boolean value that stipulates if the simulator should continue (true) instead of stopping (false) when the minimum solver time step is reached.	false
14	--strict-inner-iter-wells	A positive integer that specifies the number of inner well iterations with strict tolerance.	40
15	--strict-outer-iter-wells	A positive integer that specifies the number of newton iterations for which wells are checked with strict tolerance.	99
16	--temperature-max	A real positive value that sets the maximum absolute temperature.	400
17	--temperature-min	A real positive value that sets the minimum absolute temperature.	280
18	--gpu-mode	Use GPU cusparseSolver or openclSolver as the linear solver, usage: '--gpu-mode=[none|cusparse|opencl]'	
19	--relaxed-flow-tol-inner-iter-msw	Relaxed tolerance for the inner iteration for the MSW flow solution.	
20	--relaxed-pressure-tol-inner-iter-msw	Relaxed tolerance for the inner iteration for the MSW pressure solution.	
21	--strict-inner-iter-ms-wells	Number of inner iterations for multi-segment wells with strict tolerance.	

OPM Flow 2021-10 New And Deprecated Command Line Options			
No.	**Variable Name**	**Description**	**Default**
22	--use-inner-iterations-ms-wells	Use nested iterations for multi-segment wells.	
23	--use-inner-iterations-wells	Use nested iterations for standard wells.	

Notes:

1) Cells colored in green in the "No." column indicate a new command line option for this release.
2) Cells colored in red in the "No." column indicate a deprecated command line option for this release.

Table B.9: OPM Flow 2021-10 New and Deprecated Command Line Options

B.5.2 NEW FEATURES

In addition to the above the following new features have been added to the simulator:

1) Added a new output SUMMARY file type Enhanced SMRY (*.ESMRY) that is optimized for fast loading of selected vectors by post-processing applications. The *.ESMRY file is re-written for every time step and does not have a performance impact on the simulation. The file type is compatible with OPM ResInsight and the option is activated via the enable-esmry=true command line option (*#2590*, *#2606*, *#2609*, *#2621*, and *#3453*).

2) Passive tracer support has now been implemented in this release via the TRACER, TRACERS and TVDP keywords. The feature is activated by the command line variable -enable-tracer-mode and is by default set to "true". Testing indicates the results are comparable to the commercial simulator. Tracer saturations can be visualized in OPM ResInsight together with the field and well SUMMARY vectors. Work on partitioning tracer support, tracers that transfer between phases, for example a gas phase tracer dissolving into the oil phase, is ongoing (*#2553* and *#3433*).

3) Added two phase gas-water support and single phase water support. Note this only works with Enumeration Initialization, that is the standard Equilibrium Initialization will result in OPM Flow throwing an exception since currently Equilibrium Initialization only works when oil is present in the model (*#3454*).

4) Carter-Tracy aquifers (AQUCT keyword in the GRID section) can now be used with the CO2 Storage Model (CO2STORE keyword in the RUNSPEC section). In addition, the TEMP parameter on the AQUCT keyword may be used to define the initial temperature of the aquifer at the aquifer's datum depth (*#2694*).

5) Numerical aquifers (AQUNUM keyword in the GRID section) can also now be used with the CO2 Storage Model, CO2STORE keyword in the RUNSPEC section (*#3532*).

6) Implemented the Microbially Induced Calcite Precipitation ("MICP") model used to investigate CO2 leakage remediation. The module requires the WATER and MICP keywords in the RUNSPEC section, in addition to the MICPPARA keyword in the PROPS section to define the model parameters. Enumeration Initialization is via the SBIOF, SCALC, SMICR, SOXYG, and SUREA keywords in the SOLUTION section, to set the initial values for all cells in the model. The WMICP keyword in the SCHEDULE section sets the injection concentrations for the wells (*#2743*, *#671*, *#3617*, and *#3590*).

7) Added the calculation of the SUMMARY variable FPR for when there is no oil in the system (*#3491*).

8) COMPORD added the DEPTH ordering option to the keyword, all options on the keyword are now supported (*#2585*).

9) GCONINJE and GCONPROD keywords in the SCHEDULE section now support the RESV option for the TARGET parameter on both keywords (*#3363* and *#2476*).

10) The GCONINJE keyword with the addition of the RESV option, now supports all guide rate options via the GUIPHASE parameter (*#3363* and *#2476*).

11) The GPMAINT keyword is now supported which defines the groups under pressure maintenance control, the associated flow rate and pressure targets, and fluid in-place regions associated with pressure maintenance, as well as various pressure maintenance controls (#3541).

12) GRIDUNIT keyword in the GRID section is now supported (#2588).

13) Keywords MAPAXES and MAPUNITS are now supported in *.EGRID files (#2617).

14) Added the RPTSCHED reservoir volume report with the FIPRESV option on the keyword (#3398).

15) Various improvements to the restart file output to enable restarting from a restart file, including: sequential and parallel aquifer support (#3474), connection level cumulative volumes. Support for loading the ACTIONX and UDQ data from the restart file has also be implemented, although testing of this feature is limited as of this release (#3543). Well test data, as per the WTEST keyword, is now also written to the restart file (#2718).

B.5.3 IMPROVEMENTS

Improvements include:

1) Performance improvements in the both the standard and multi-segment well models by optimizing the potential calculation using BHP and THP (#3524).

2) Changed how inoperable wells are handled during well model iterations (#3539).

3) Performance improvements in reading and writing out binary files, that reduces load times of around 60% (#2634).

4) Improvements and corrections to the RESTART file for better compatibility with the commercial simulator (#2714), support for Group Guide rate values (#2739, #2751 and #3581), and loading WTEST keyword associated data (#3594, #2746 and #2747).

5) The code for handling the WTEST keyword in the well model model was improved by only opening wells that satisfy the economic conditions and that can physically flow (#3547, #2737 and #2744).

6) Improved how the Guide Rate calculations are calculated for both groups and wells under guide rate control (#2537 and #2552).

7) Various improvements to the parser in checking valid input to various keywords and the features supported by the keywords (#2586, #2585, and #3440).

In addition, the following new SUMMARY keywords are now recognized as described by the comments in Table B.10.

No.	Summary Keyword	Comment
1	CVPR, CVPT, CVIR and CVIT	Added support for the connection reservoir voidage volume summary vectors CVPR (production rate), CVPT (cumulative production total), CVIR (injection rate), and CVIT (cumulative injection total).
2	FOIPR, FWIPR, FGIPR, and FRPV	Added support for the Field Reservoir (conditions) Pore Volume (FRPV) vector, as well as the compositional oil, water and gas fluid in-place vectors FOIPR, FWIPR, and FGIPR. that are reported at reservoir conditions, as oppose to FOIP, FGIP and FWIP that are reported at surface conditions (#2701).
3	FGLIR, GGLIR, and WGLIR	Implemented support for gas lift variables. Note that the WALQ is also written out but all the values are zero (#2495).
4	FSIP	Added support for the Field Salt In-Place vector (#3493).

No.	Summary Keyword	Comment
5	FTIC, FTIR FTIT, FTPC, FTPR, and FTP	Implemented support for field tracer variables (#3430).
6	MSUMBUG MSUMCOMM MSUMERR MSUMMESS MSUMPROB MSUMWARN NEWTON	In the 2021-04 release the message keywords were recognized by the parser; however, no data associated with these vectors is written to file (#2234). This is still the case.
7	RPTONLY RPTONLYO	Added support for SUMMARY keywords that activates (RPTONLY) and deactivates (RPTONLYO) summary output at report time steps only. The keyword works in both the SUMMARY and SCHEDULE sections (#2597 and #2761).
8	SUMTHIN	Implemented support for the SUMTHIN keyword that defines a time interval for writing out the SUMMARY data to the SUMMARY file and the RSM file. The keyword works in both the SUMMARY and SCHEDULE sections (#2489, #2596, #2489 and #2761).
9	SWCT	The water cut vector for Multi-Segment Well segments has now been implemented (#2636).
10	WTIC, WTIR, WTIT, WTPC, WTPR, and WTPT	Implemented support for well tracer variables (#3430).
11	WWIRT	In the 2021-04 release support for the WWIRT vector for writing out the water injection rate target was implemented (#2071); however, the field (FWIRT) and Group (GWIRT) are still not available. This is still the case.

Table B.10: New SUMMARY Keywords for the 2021-10 Release

B.5.4 BUG FIXES

The following bug fixes and improvements have been incorporated into this release.

1) Previously, if the ACTNUM keyword was not supplied in the input deck when using the GDFILE keyword to define the grid, then the simulator would abort with a segmentation fault, even though the ACTNUM keyword is optional. This has now been corrected in this release (#2675).

2) Fixed a bug in RESTART output when using the FREQ=x option on the RPTRST keyword. which caused the output to the restart file to be based on calendar time instead of simulation time. For example if x was set to six, then the RESTART records were written January and July, instead of every six months of simulated time. This has now been fixed in this release (#2551).

3) OPM Flow does not support the SAVE file format but instead writes out a RESTART file record instead at the requested time. However, the RESTART record was written at the TSTEPn+1 instead

of TSTEPn, this has now been changed so that the RESTART record is written at TSTEPn, the same as the commercial simulator's equivalent SAVE file output (#2661).

4) Corrected an issue for when a region has zero active cells, when calculating the average region properties (#3514).

5) Fixed an issue with wells shut-in for the norne 4_MSW test case which may impact user's cases (#3481).

6) Previously the IMPORT keyword could not utilize the directory path declared via the PATHS keyword, this has now been rectified (#2600).

7) Fixed and issue when there are unknown keywords and the parser throws an exception in sequential runs and a segmentation fault in parallel runs (#3503).

8) Production and injection cumulative volumes were not necessary correct if wells changed from being a producer to an injector or vice versa. This has now been corrected (#2581).

9) Fixed an issue in parallel runs when initially all connections are close for a well (#3495).

10) Fixed a bug on the FIP report so that the PORV and TOTAL PORE VOLUMES values are the same as the commercial simulator, previously there were small discrepancies (#3398).

11) Reverted some changes on how the FACTOR (efficiency factor) parameter is applied on the GEFAC keyword in the SCHEDULE section. Now FACTOR is only applied while accumulating rates from subordinate groups (#3464).

12) Fixed a bug associated with the writing out of SUMMARY variables to the SUMMARY and RSM files referencing the first simulation date as oppose to the start of simulation date as per the START keyword (#2685).

13) Various ACTIONX keyword improvements and fixes, including:

    a) Verifying that the ACTIONX parameters and dimensions are consistent with the values entered on the ACTDIMS keyword in the RUNSPEC section. Previously, if the ACTDIMS parameters were inconsistent with the ACTIONX parameters OPM Flow would abort (#2695).

    b) For the ACTIONX date parameter MNTH, an error would occur if the month was enclosed in quotes, now both quoted ('JAN') and unquoted (JAN) values for MNTH are accepted (#2696).

    c) Numerical values for the ACTIONX date parameter MNTH are now also accepted. The values will be rounded to the nearest integer. So a MNTH value of 4.4 would result in the month being APR for April (#2699).

    d) Ensuring that matched wells are passed to WCONPROD and WCONINJE keywords using the '?" string as the well name (#2753).

14) If the WTEST keyword was activated and opened a well to flow, then the well status remained unchanged, instead of being updated to open. This has now been rectified for this release (#3537).

15) Previously, the numerical aquifer keywords AQUCON and AQUNUM were incorrectly reported as unsupported. This has now been corrected in this release (#3230).

16) In the previous release, if the aquifer summary variables ANQR, ANQP and ANQT used the default version of the keywords, that is:

```
ANQR
/
```

    then the vectors were not written out. This has now been rectified in this release (#2474).

    Note that the FNQR and FNQT keywords are unsupported and a corresponding warning message is issued.

17) Hitherto the COPYREG keyword was reported as unsupported but was in fact supported and functioning. The incorrect unsupported message no longer occurs in this release (#3234).

18) In the SOLUTION section the RPTSOL keyword was up to now marked as active and was treated as such by the simulator; however, no output was generated in the print file. The keyword is now labeled as unsupported (#3248). Note this is the print keyword for the SOLUTION section, and should not be confused with the SCHEDULE section print keyword RPTSCHED.

19) Rectified several issues with the GCONINJE re-injection options and fixed several well modeling bugs that caused the simulator to crash (#2708, #3546 and #2666).

20) When running the carbon dioxide (CO2) storage model, activated by the CO2STORE keyword in the RUNSPEC section, in parallel, the simulator would deadlock due to passed parameters being unreasonable. The deadlock has now be fixed for this release and error trapping is now the same as per a serial run. (#3545 and #3556).

21) In CO2 sequestration runs, activated by the CO2STORE in the RUNSPEC section, that typically model large time steps, the reported dates were inaccurate, this has now been rectified (#3560).

22) Corrected an error in restart runs for history matching wells. Previously, the wells used the default value of one atmosphere for the BHP instead of the actual BHP value in the restarted run, which resulted in numerical issues (#2715).

23) For the WECON keyword in the SCHEDULE section, the units for GOR and WGR parameters caused the well to prematurely shut-in, this has now been corrected. (#2726). Also fixed an issue with completions not being opened after a well event (#3563 and #2723) and how the WTEST keyword in the SCHEDULE section was handled by the simulator (#2720).

24) Fixed an issue with aquifer connections in parallel runs (#3572).

25) Corrected how aquifer connections and MINPV interact so that only active cells are connected to the aquifer (#2730 and #2734).

26) Fix an issue with writing out tracer data to the restart file for time steps that are not report time steps (#3606).

27) Fix several issues with well operability that caused wells to be shut-in (#3569) and for history match wells that cannot produce terminating the run prematurely (#3607).

28) In the radial model there was a bug that caused all the radial cells in the grid having a zero thickness, this has been addressed in this release (#2640)

29) Previously, both the GLIFTOPT and NETWORK keywords were reported as unsupported when in fact they were supported and fully functional. This has now been fixed for this release (#3280).

B.5.5 KNOWN ISSUES

1) The tracer related output to summary file does currently not respect the command line variable -enable-tracer-mode, this means we may have the situation in which the input deck specifies tracer output, and the simulator is run without enabling the tracer model (enable-tracer-mode=false). In this scenario the trace SUMMARY vector output will consists of irrelevant tracer values.

2) Although the SUMMARY vector WALQ is written out all the values are zero.

3) The GDFILE keyword in the GRID section loads a grid file in various formats, with the FMTOPT parameter setting the format type of the file. If the variable FMTOPT is omitted then the default is for binary file input for the commercial simulator; whereas, OPM Flow derives FMTOPT from the file extension (*.EGRID or *.FEGRID), making FMTOPT superfluous. However, if the extension is lower case then OPM Flow may incorrectly determine the file type. The work around is change the extension to upper case.

4) Currently two phase gas-water and single phase water models only work with Enumeration Initialization.

5) For the saturation tables (SWOF, SWFN etc.) if the capillary pressure values were defaulted with 1* then the simulator should use linear interpolation to calculate the defaulted values. This is no longer working.

6) As per previous releases of the radial model, the COORDSYS keyword item three must be set to COMP to complete the circle, this has not been implemented in this release. Also there appears to be a bug for full radial models when a well goes on BHP control that causes the well not to respect the BHP constraint, this eventually causes the well to die prematurely. See *#2640* for a discussion on the topic.

7) Using the GRAVITY keyword in the PROPS section will cause an exception to be raised and the simulator to abort. Use the DENSITY keyword instead to avoid this error.

8) As per the previous releases, the RPTSCHED WELLS sub-report does not print the FIELD group production data.

9) As in previous releases there are some issues with the OPERATE and OPERATER keywords associated with the input parsing; for various reasons a few of the fields require special case treatment in the grid processing, including (at least) MULTZ, PORV and ACTNUM, and for those keywords the OPERATE/OPERATER keyword doe not work.

10) OPM Flow does not support the SAVE file format for fast restarts like the commercial simulator, but instead writes a standard RESTART record at the requested time step in the SCHEDULE section. If the keyword is encountered in the RUNSPEC section it is ignored. However, the simulator issues a false warning saying the keyword is not supported, when in fact RESTART records are indeed written out in the SCHEDULE section if requested by the SAVE keyword.

11) Gas lift optimization currently only works for cases that have the gas phase present, that is runs with dead oil and no gas phase will cause the simulator to throw an exception (*#3658*).

12) For the UDQ ASSIGN operator after the terminating "/" normally any comments can be entered; however, if there is "/" within the comment field, as per:

```
ASSIGN FUNGLYLD 1.196  /  Condensate Yield (63.5/56.7)/(1.0 – 0.065)
```

then the simulator will abort. The work around is to manually place the comment characters "--" after the ASSIGN terminating "/", like so:

```
ASSIGN FUNGLYLD 1.196  / -- Condensate Yield (63.5/56.7)/(1.0 – 0.06)
```

13) At the moment, one cannot initialize tracers using the EQUALS keyword. Instead use the array format, that is the keyword followed by the required number of values, or the TVDP keyword in the SOLUTION section to set the initial tracer concentrations as a function of depth.

14) Currently, gas tracers cannot be used if the dissolve gas phase, as per the DISGAS keyword in the RUNSPEC section, is active in the model.

Bård Skaflestad, Jostein Alvestad, Kai Bao, Markus Blatt, Joakim Hove, Arne Morten Kvarving, Cintia Goncalves Machado, David Landa-Marbán, Atgeirr Flø Rasmussen (Release Manager), Alf Birger Rustad, Tor Harald Sandve, Torbjørn Skille, Pieter J. Verveer, and David Baxendale.

B.6 RELEASE 2021-04

The 2021-04 release consists of some new features and various improvements and bug fixes. Highlights for this release includes support for numerical aquifers, gas lift optimization, network models, spider grids and support for diffusive flow for carbon dioxide storage, as well as convective dissolution of carbon dioxide into in situ brine.

B.6.1 NEW COMMAND LINE OPTIONS

The major command line made to this release are summarized in Table B.11

OPM Flow 2021-04 Major Command Line Option Changes			
No.	**Variable Name**	**Description**	**Default**
1	-h or --help	A character string that causes OPM Flow to print a help message that gives a brief description of the available command line parameters. **This now only prints the active user facing command line options.**	N/A
2	--help-all	Prints all the command line options included in the release, including obsolete, hidden and deprecated options.	N/A
3	--allow-distributed-wells	Allow the perforations of a well to be distributed across multiple processes in parallel runs.	false
4	--opencl-ilu-reorder	A defined character string that selects the reordering strategy for ILU for openclSolver and should be set to: 1) "level_scheduling", or 2) "graph_coloring" Note that "level_scheduling" behaves like Dune and cusparse, whereas "graph_coloring" is more aggressive and is likely to be faster, but is random-based and generally increases the number of linear solves and linear iterations significantly.	"graph_coloring"
5	--vtk-write-diffusion-coefficients	A Boolean value that switches on (true) or off (false) the output of the molecular diffusion coefficients to the VTK[357] output files.	false
6	--vtk-write-effective-diffusion-coefficients	A Boolean value that switches on (true) or off (false) the output of the effective molecular diffusion coefficients for the medium to the VTK output files.	false
7	--vtk-write-tortuosities	A Boolean value that switches on (true) or off (false) the output of the tortuosity for each phase to the VTK output files.	false

[357] *Virtualization ToolKit ("VTK") files used to visualize the 3D results from the model using ParaView, an open-source, multi-platform general purpose data analysis and visualization application (https://www.paraview.org/overview/). For the commercial simulator's compatible output files OPM's ResInsight 3D visualization package can be used instead and is specifically tailored for OPM Flow and the commercial simulator.*

OPM Flow 2021-04 Major Command Line Option Changes			
No.	**Variable Name**	**Description**	**Default**
8	--zoltan-imbalance-tol	A real positive that defines the tolerable imbalance of the load balancing provided by Zoltan package.	1.1

Table B.11: OPM Flow 2021-04 Major Command Line Option Changes.

B.6.2 NEW SIMULATOR FEATURES AND IMPROVEMENTS

In addition to the above the following new features have been added to the simulator:

15) Gas Lift Optimization: Added support for gas lift optimization, including in parallel, that enables the simulator to optimize allocate gas lift to wells and groups.

16) Numerical Aquifers. Added support for numerical aquifers that allow for flexible and efficient inclusion of pressure support from adjacent aquifers. Analytical aquifers have been supported in previous releases. See the AQUNUM and AQUCON keywords in the GRID section

17) Support for aquifers in MPI runs, that is in parallel runs.

18) Support for wells split between several process domains. This potentially gives better parallel grid partitioning. The well split is only implemented for standard wells, not for multi-segment wells; although the latter may be implemented in a future release.

19) Added guide rate support for injection groups via the GCONINJE keyword in the SCHEDULE section using the GRPGUIDE and GUIPHASE parameters on the keyword. Only the RATE, NETV and VOID guide rate options are supported.

20) Network Option. Implemented the Network option that allows for well/group coupling through common tubing head pressures when required.

21) Previously in an ACTIONX block only a subset of the well keywords could be used in the SCHEDULE section, this has now been extended to include the GCONPROD, GCONINJE and GLIFTOPT group keywords.

22) ACTIONX block can now handle well lists declared via the WLIST keyword for well keywords that are available within an ACTIONX block.

23) ACTIONX User Defined Quantity ("UDQ") can now be used for the artificial lift quantity on the WCONHIST and WCONPROD keywords in the SCHEDULE section, the ALQ-WELL parameter on the aforementioned keywords.

24) Expanded possible actions in ACTIONX block, including the use of the WELPI keyword parameter and UDQ updates.

25) Implemented convective DRSDT for CO2 storage as described by Sandve at al.[358]. The convective DRSDT is activated with the DRSDTCON keyword in the SCHEDULE section.

26) Added experimental support for diffuse flow via the DIFFC and DIFFUSE keywords, this facility is not available via OPM Flow but can be accessed via re-compiling the source code with the option enabled. Due to the experimental nature of this feature and the unavailability in the released binary packages, the keywords and the functionality are not documented in this release of the manual.

27) Added support for the WELPI keyword for well productivity based scaling of a well's connection transmissibility factors.

28) Three-point vertical scaling of relative permeability functions (keywords KRGR, KRORG, KRORW, and KRWR).

[358] *Tor Harald Sandve1, Sarah E. Gasda, Atgeirr Rasmussen, and Alf Birger Rustad. Convective dissolution in field scale CO_2 storage simulation using the OPM Flow simulator. Submitted to TCCS 11 – Trondheim Conference on CO_2 Capture, Transport and Storage Trondheim, Norway – June 21-23, 2021.*

29) Support for well and group names with more than eight (8) characters in summary output.

30) Added support on the WECON keyword in the SCHEDULE to apply the economic limit test using a well's potential, previously only the default of using a well production rate was implemented.

31) Support for IMPORT keyword in GRID section.

32) Added support for Python to access the PORO array.

33) Added support for SPIDER grids. The SPIDER keyword activates the radial grid geometry option for the model using Cartesian coordinates, if this keyword and the RADIAL keyword are omitted then Cartesian geometry is assumed by OPM Flow. This keyword will create a spiderweb-shaped grid based on a corner-point grid using the standard radial grid keywords: INRAD, DRV, DTHETAV, DZ/DZV etc. in the GRID the section. In addition the PERMR and PERMTHT keywords for radial and spider grids are now supported.

34) Added support for the multi-segment well keyword WSEGAICD that incorporates an autonomous inflow control device in a multi-segment well.

35) Continued improving OPM Flow's restart capability inline with the commercial simulator's functionality, including obtaining the restart point based on TSTEP in addition to the DATES keyword time steps.

In addition, the following new SUMMARY keywords are now recognized as described by the comments in Table B.12.

No.	Summary Keyword	Comment
1	AAQTD AAQPD	Added support for the AAQTD and AAQPD vectors in the SUMMARY section, for calculating and reporting the dimensionless time (simulator time divided by aquifer's time constant) and pressure (influence function evaluated at dimensionless time) values as part of the Carter-Tracy aquifers.
2	BKROG BKROW	Added support for the BKROG and BKROW vectors that output the block-level values of the relative permeability of oil in the two-phase oil/gas and oil/water subsystems of a three-phase simulation run.
3	BSOIL BSGAS BSWAT BPRESSUR	Added support for SUMMARY variables BSOIL, BSGAS, BSWAT, and BPRESSUR as aliases for BOSAT, BGSAT, BWSAT, and BPR respectively.
4	CORPL COFRL	Added CORPL and CORFL as as aliases for WOFRL for a well's completion oil rate.
5	RHPV	Added RHPV vector, a regions hydrocarbon pore volume, to the SUMMARY section variables.
6	WEPR WEPT	Added support for the WEPR and WEPT energy vectors in the SUMMARY section.
7	WWIRT	Added support for the WWIRT vector in the SUMMARY section for writing out the water injection rate target. Note that the field (FWIRT) and Group (GWIRT) are still not available.

No.	Summary Keyword	Comment
8	MSUMBUG MSUMCOMM MSUMERR MSUMMESS MSUMPROB MSUMWARN NEWTON	The message keywords are now recognized by the parser; however, no data associated with these vectors is written to file. Previously these keywords would cause an error and the simulator would stop due to the error.

Table B.12: New SUMMARY Keywords for the 2021-04 Release

B.6.3 BUG FIXES

The following bug fixes and improvements have been incorporated into this release.

1) Various fixes for thermal simulations, including how the thermal conductivity was calculated in thermal runs, now the thermal conductivity is calculated in the same manner as permeability and diffusivity.

2) Fixes in the GCONINJE implementation.

3) On both the GCONPROD and GCONINJE keywords in the SCHEDULE section, the GRPCNTL property sets if a the group is subject to higher level group control or not. If the group being defined is the FIELD then GRPCNTL is supposed to be ignored, this was previously not the case. This has now been corrected.

4) Fixes in the boundary setup when using the OPM specific BC keyword.

5) Correct inclusion of the group efficiency for the top group level

6) Improved convergence of the multi-segmented well equations.

7) Various MPI related fixes, avoiding dead-locks etc.

8) Corrected various issues with the UDQ facility, including when the UDQ keyword contains both ASSIGN and DEFINE statements such that only the ASSIGN should be only be evaluated for the first time step, and then in consecutive time steps the DEFINE statement should be executed. Also when a UDQ expression refers to other UDQ's the previous value for the other UDQ's should be used when evaluating the expression.

9) Fixed an error in the processing of the COPYREG keyword due to picking up the wrong item; however, COPYREG keyword is reported as unsupported but is in fact supported and functioning.

10) Corrected various issues with well reference depths used to report BHP, so that the reference depth is not updated when new connections are added via the COMPDAT keyword, but is updated when the WELSPECS keyword is introduced.

11) Fixed an error with the CO2STORE keyword being reported as unsupported. The keyword is now recognized and functioning.

12) Fix a bug that causes the simulator to throw an exception if the SUMMARY temperature keywords were used (WBHPT, FTIRHEA, etc.) and were not recognized by the parser.

13) Fixed a bug with the WTICHEA and WTPCHEA variables in the SUMMARY section not reporting the correct temperatures. Now the temperature as set by WTEMP is reported for the injector, which is consistent with the commercial simulator. For non-thermal decks both injectors and producers now report the standard condition temperature of 60o F or 15.56o C, depending on the units being used. For thermal decks the producers report the temperature at same reference depth as the BHP and the bottom-hole temperature is now calculated.

14) Fix an error associated with the number of FIPNUM regions and the SUMMARY file output associated with using the general form of the region summary vectors, that is:

```
RPR
/
```

as opposed to declaring the region format.

```
RPR
1  2 /
```

15) The issue with the ALL keyword in the SUMMARY section that reports "Warning: Unhandled summary keyword ALL" multiple times has been fixed. The message now states the actual summary vector associated with the ALL keyword that is not available, instead of stating the actual ALL keyword.

16) If gas lift optimization was not used the simulator did not use the ALQ value on the WCONPROD keyword as the default value, this has now been fixed.

17) A bug that caused the simulator to throw exception when the depth on the SALTVD table did not cover the depth range of the model has been fixed.

18) Previously the SUMMARY vector ROEW, that is a regions oil recovery factor, was based on a well's production total (WOPT), this has now be changed to be based on a well's completion total (COPT) instead, as it was not obvious how to assign wells to regions.

19) Fixed an error in the ALQ calculation for injected gas lift that used rate instead of volume in the calculation.

20) Added the unused commercial compositional simulator's threshold saturation of Killough's hysteresis model to the EHYSTR. The parameter is not used by OPM Flow.

21) As per the previous releases, two phase gas-water models (GAS and WATER only keywords in the RUNSPEC section) will not initialize in OPM Flow. Previously, the simulator would throw an exception, this is now trapped and a message is issued stating the gas-water systems are not currently supported. The work around is to convert the model to a three phase model (OIL, GAS and WATER keywords in the RUNSPEC section) and modify the PROPS section PVT keywords accordingly.

22) Previously the OPERATE keyword EQUATION options of MAXLIM and MINLIM were not supported, this has been rectified for this release and both options are now fully supported.

B.6.4 KNOWN ISSUES

1) The numerical aquifer keywords AQUCON and AQUNUM are incorrectly reported as unsupported. This has already been corrected in the current MASTER, so will be fixed in the next release.

2) The aquifer summary variables ANQR, ANQP and ANQT variables are supposed to be supported and simulator does not issue a warning message if these keywords are in the input deck. However, the output from these keywords is not written to the SUMMARY or RSM file. if the default version of the keywords are used, that is:

```
ANQR
/
```

will not work, but specifying the aquifer numbers will work, so for aquifer number one we would use:

```
ANQR
1 /
```

Note that the FNQR and FNQT keywords are unsupported and a corresponding warning message is issued.

3) The COPYREG keyword is reported as unsupported but is in fact supported and functioning. The issue has been corrected for the next release.

4) In the SOLUTION section he RPTSOL keyword is marked as active and is treated as such by the simulator; however, no output is generated in the print file. The keyword should be labeled as

inactive. Note this is the print keyword for the SOLUTION section, and should not be confused with the SCHEDULE section print keyword RPTSCHED. The issue has been corrected for the next release.

5) Using radial grids via the RADIAL keyword in the RUNSPEC section will cause the simulator to throw an exception as radial grids are not included in the release. Instead one can use the SPIDER grid keyword for radial flow simulation that converts the radial entered keywords to Cartesian coordinates using Irregular Corner-Point Grid – see the example in the section on SPIDER grids (*Spider Grids*).

6) For radial grids the COORDSYS keyword item three must be set to COMP to complete the circle, this has not been implemented in this release.

7) Both the GLIFTOPT and NETWORK keywords are reported as unsupported when in fact they are supported and fully functional. This has already been fixed for the next release.

8) Using the GRAVITY keyword in the PROPS section will cause an exception to be raised and the simulator to abort. Use the DENSITY keyword instead to avoid this error.

9) As per the previous releases, the Tracer model is not working. In order to activate the Tracer model the command line variable -enable-tracer-mode should be set to "true"; the simulator will still issue a message saying *Warning: Keyword 'TRACERS' is not supported by flow* but will continue to run. However, the results from the tracer tracking appear to be incorrect and should not be relied upon. The standard results, rates, pressures, saturations, etc., are identical to the comparable no tracer run.

10) As per the previous releases, the RPTSCHED WELLS sub-report does not print the FIELD group production data.

Bård Skaflestad, Jostein Alvestad, Kai Bao, Markus Blatt, Joakim Hove, Arne Morten Kvarving, Cintia Goncalves Machado, Atgeirr Flø Rasmussen, Alf Birger Rustad, Tor Harald Sandve (Release Manager), Torbjørn Skille, and David Baxendale.

B.7 RELEASE 2020-10

Since the OPM 2020.04 release in May 2020 the developers have worked on many aspects of the simulator, with a particular focus on prediction mode abilities and expanding support for user-defined quantities and dynamic actions (ACTIONX keyword). The project team have also added specialized physics models including Brine and CO2 storage capabilities as well as two experimental accelerators for the linear solvers based on OpenCL and CUDA for GPU calculations.

B.7.1 NEW OUTPUT FILES AND COMMAND LINE OPTIONS

The following new out files and command load options have been implemented in this release:

1) Add a new output file, *.INFOSTEP, that contains per time step performance numbers, that is the number of linear solves, number of linear iterations, setup time, etc.

2) Improve efficiency in loading summary files, especially to support manipulating summary data from Python wrappers. This also includes a new special purpose utility, called "make_lodsmry", that creates files named *.LODSMRY which contains the summary vectors collected in the time direction for all values for one vector, before all values of another vector.

3) Exclusively use the "flexible" framework for selecting linear solvers, through the new command line option "--linsolver". The simulator supports the following pre-configured options:

    1) "ilu0",
    2) "cpr_trueimpes",
    3) "cpr_quasiimpes",
    4) "cpr" (alias for "cpr_trueimpes"), or
    5) "amg".

    Advanced users may alternatively select a runtime configuration represented in a JSON[359] file by passing a filename with the extension ".json". The default setting is "ilu0".

4) Added an option to use non-strict tolerances only if a small fraction of the total pore volume violates the strict tolerances. By default the simulator uses a non-strict pore volume threshold of 0.03 (3%), but this is configurable at run time with the new option '--relaxed-max-pv-fraction'.

5) Add new option ProjectSaturations (--project-saturations) which ensures all saturation values are in the interval (0, 1), including runs that use solvents.

6) Replaced the various –vtk-write command line options with one global option,--enable-vtk-output, that enables or disables the writing of VTK files.

B.7.2 NEW SIMULATOR FEATURES

In addition to the above the following new features have been added to the simulator:

1) Initial support for calling simulator time step functions from Python language wrappers have been implemented.

2) A new PVT model for CO_2 and brine has been implemented; see the C02STORE keyword in the RUNSPEC section and SALINITY keyword in the PROPS section.

3) Added support salt dependent water PVT properties, see the PVTWSALT keyword in the PROPS section and SALTVD keyword in the SOLUTION section.

4) Added support for using user defined arguments (UDAs) in the WELTARG and GCONPROD keywords.

[359] *JavaScript Object Notation is an open standard file format, and data interchange format, that uses human-readable text to store and transmit data objects consisting of attribute–value pairs and array data types.*

5) Implemented support for using SOF2 (Family II) with Family I (SGOF/SWOF) keywords in the case of three-phase solvent models.

6) Added support for well-level temperature summary output in thermal simulations (SUMMARY keywords WTICHEA and WTPCHEA). However, subsequent testing indicates that data written out is incorrect.

7) Implemented support gas lift optimization for standard (non-segmented) wells, see the LIFTOPT, GLIFTOPT and WLIFTOPT keywords in the RUNSPEC section.

8) Implemented support for field, group, and well level summary output of gas lift injection rates (keywords FGLIR, GGLIR, WGLIR in the SUMMARY section). However, although the keywords are recognized and the data written to the SUMMARY file, subsequent testing indicates all the values are zero.

9) Added summary output for analytical aquifers: AAQR, AAQT, and AAQP. Note that the following analytical aquifer summary keywords will be ignored: FAQR, FAQT, AAQTD and AAQPD.

10) Implemented the SUMMARY vectors FMWIA, FMWPA, GMWIA, and GMWPA for counting abandoned producers and injectors. The vectors can also written out via the FMWSET and GMWSET keywords in the SUMMARY section.

11) Incorporated the FILLEPS keyword in the PROPS section. Saturation function scaling end-points now also honor the TOLCRIT value defined by the TOLCRIT keyword when written to the *.INIT file.

12) Generalized the implementation of the FIP keyword and to handle the additional summary variables that include fluid in-place region name defined by a five character string following the FIP characters on the FIP keyword, FIP-BLK1 for example. This enables the regional properties for FIP-BLK1to be written to the SUMMARY file. For example to write the regional pressure for "-BLK1" one would use the SUMMARY keyword RPR-BLK1, where "-BLK1" can be any character string.

13) Improved the error messages from the initial input parsing of the input deck for great clarity and to better enable input deck validation.

14) Added support for group control targets/prediction mode with guide rates in the simulator (see the GUIDERAT keyword in the SCHEDULE section) that also incorporates voidage replacement and re-injection scenarios. Note that the simulator will erroneously report that the GUIDERAT keyword is not recognized, when in fact it is and will be used by the simulator.

15) Added support for outputting guide rate values at well and group levels to the SUMMARY files (summary keywords W*PGR and G*PGR).

16) Added the support for the GCONSALE keyword that defines group sales gas production targets and constraints for when the gas production from an oil field group is exported under a Gas Sales Agreement ("GSA") and the oil field group also has oil production targets and constraints. In addition the GCONSUMP keyword has been implemented that defines the group gas consumption (fuel) rate. Note again the simulator will erroneously report that the two keywords are not recognized, when in fact they are and will be used by the simulator. Secondly, the SUMMARY vectors associated with these keywords (FGSR, FGST, FGCR, FGCT, GGSR, GGST, GGCR, and GGCT) have not been implemented).

17) Added additional logging information to the *.PRT file for the setup phase of the simulator when processing the *.DATA file, including which keywords and their file locations are being processed at any one time. This is to assist the user in narrowing down problems in the input file.

18) Add support for editing transmissibility values in the EDIT section.

19) Add support for handling MULTZ in a pinched-out column of cells when the PINCH multiplier processing mode is 'ALL'.

20) Implemented a clean shutdown in a parallel simulation run if any process throws an exception.

21) Add support for running Zoltan[360] based partitioning on a single process only.

B.7.3 BUG FIXES

The following bug fixes and improvements have been incorporated into this release.

1) Fix a bug in parsing the UDQ keyword that caused the UDQ tokenizer to split on the '-' character when parsing numerical literals on scientific form: 1E-5.
2) Added the missing GLR_LIMIT to the WCUTBACK keyword, solving bug #1877.
3) The simulator now correctly handles the RPTRST keyword in SOLUTION section.
4) Removed OPERATER and OPERNUM keywords from the keywords not supported since they are supported.
5) Fix the default pressure initialization for Carter-Tracey analytical aquifers.
6) Fix the way the simulator incorporates the gravity terms in boundary conditions fluxes.
7) Fix bugs associated with the TRAN* series of keywords in EDIT section.
8) Ensure that VTK data set filenames use relative paths.
9) Various bug fixes for the Message Passing Interface ("MPI"). MPI is a standardized and portable message-passing system developed for distributed and parallel computing.
10) Fixes to the source building work flow, especially for the "opm-upscaling" module.
11) Fix a bug in the interaction of TUNING and WSEGITER. This was needed for compatible with the commercial simulator's restart feature.
12) The simulator issued a message "All completions in well XXX is shut at X.XXXXX days. The well is therefore also shut" multiple times, this has been fixed for this release.
13) Fixed missing tubing length data in the WELSPECS Multi-Segment Well Connection sub-report.
14) Previously OPM Flow incorrectly reported that the MAXVALUE keyword was not supported. The warning message was incorrect as the values entered on the keyword were applied to the named array. The warning message has been removed for this release.
15) Previously for the Brine model, the values entered via the SALTVD keyword were ignored and reset to zero. This has been fixed for this release.
16) Fixed an issue with the simulator failing to correctly read the RTEMP and RTEMPA keywords.
17) Added the following polymer model summary vectors to the SUMMARY file: FCIR, FCIT, FCPC, FCPR, FCPT, GCIR, GCIT, GCPC, GCPR, GCPT, WCIR, WCIT, WCPC, WCPR, and WCPT.
18) Fixed a bug with the standard usage of ROCKTAB and throw an exception on the more exotic usages that are unsupported.

B.7.4 KNOWN ISSUES

1) The simulator will throw an exception if the depth entries on the SALTVD table do not cover the range of the model depths, as this keyword currently not set up to extrapolate outside its domain of definition. The work around is to ensure the depths on the SALTVD keyword cover the depth range in the model.

[360] *The Zoltan library is a toolkit of parallel combinatorial algorithms for unstructured and/or adaptive computations, for dynamic partitioning using graph coloring and ordering. In addition to native implementations of many algorithms, Zoltan interfaces to the graph and hypergraph partitioning libraries of PT-Scotch, PaToH and ParMETIS. See* http://www.cs.sandia.gov/zoltan/ *for further information.*

2) The simulator may throw an exception if a SUMMARY keyword (WBHPT, FTIRHEA, etc.) is not recognized by the parser. The work around is to delete the offending keyword(s) from the input deck.

3) There is an issue with the ALL keyword in the SUMMARY section that reports " *Warning: Unhandled summary keyword ALL*" multiple times. The ALL keyword is handled by the simulator and the currently available supported summary vectors will be written out to the SUMMARY and RSM files (if requested). The message should state the actual summary vector associated with the ALL keyword that is not available, instead of stating the actual ALL keyword.

4) As per the previous release, two phase gas-water models (GAS and WATER only keywords in the RUNSPEC section) will not initialize in OPM Flow, this has been the status for past versions of the simulator as well. The work around is to convert the model to a three phase model (OIL, GAS and WATER keywords in the RUNSPEC section) and modify the PROPS section PVT keywords accordingly.

5) As per the previous release, the Tracer model is not working. In order to activate the Tracer model the command line variable -enable-tracer-mode should be set to "true"; the simulator will still issue a message saying *Warning: Keyword 'TRACERS' is not supported by flow* but will continue to run. However, the results from the tracer tracking appear to be incorrect and should not be relied upon. The standard results, rates, pressures, saturations, etc., are identical to the comparable no tracer run.

6) The OPERATE keyword EQUATION options of MAXLIM and MINLIM are currently not supported.

7) As per the previous release, the RPTSCHED WELLS sub-report does not print the FIELD group production data.

Bård Skaflestad (Release Manager), Jostein Alvestad, Kai Bao, Markus Blatt, Joakim Hove, Arne Morten Kvarving, Cintia Goncalves Machado, Atgeirr Flø Rasmussen, Alf Birger Rustad, Tor Harald Sandve, Torbjørn Skille, and David Baxendale.

B.8 RELEASE 2020-04

B.8.1 NEW FEATURES

Writing out of the SUMMARY file data in a columnar format to the RSM file has been implemented. Normally the SEPARATE keyword in the SUMMARY section is invoked to direct the data stream to a separate RSM file for easy loading into other programs, for example, Microsoft's EXCEL or LibreOffice's CALC spreadsheet program. However, this is the default behavior for OPM Flow.

Additional SCHEDULE section reports has been added to this release including the WELLS and WELSPECS options on the RPTSCHED keyword. The reports are similar to those from the commercial simulator.

The Brine Tracking option has now been activated in this release and should be considered experimental and used with caution. The keywords associated with this feature are BRINE, BDENSITY, PVTWALT and WSALT.

A Python scripting facility has been implemented in OPM Flow via the:

1) PYINPUT/PYEND keywords that processes standard Python commands that can be used to manipulate and define the simulators input parameters during processing of the input deck. The main purpose of the facility is to script the construction of the various keywords.
2) PYACTION keyword that provides similar functionality as the ACTIONX scripting facility, but instead uses the standard Python interpreter. PYACTION provides an interface between the simulator and Python via a Python function that enables obtaining various run times parameters from the simulator.

Experimental support for using the Graphics Processing Unit ("GPU") *--cusparseSolver* as the linear solver has been implemented, see the command line parameter *–use-gpu* in section *2.2 Running OPM Flow 2023-10 From The Command Line*.

A command line parameter --sched-restart has been added to allow a restart run to initialize wells and groups from the historical SCHEDULE section (current and past default behavior), or from the well and group data on the RESTART file. Note that the commercial simulator always uses data from the restart file. The intention is to duplicate the behavior of the commercial simulator after suitable testing.

The initialization of parallel cases has been completely reworked, the simulator now uses considerably less memory when initializing a parallel run.

Group control of wells in prediction mode has been made more consistent and improved in several ways, this may impact the results compared to previous versions of the simulator.

All black-oil keywords are now documented to varying level of detail depending on the functionality implemented in OPM Flow, and all are now recognized by the OPM Flow input deck parser.

In addition the following new keywords are now recognized with varying functionality as described by the comments in Table B.13.

No.	Keyword	Comment
1	BDENSITY	Define the Surface Brine Density for the Fluid.
2	BRINE	Activate Brine Tracking Option.
3	DATE	Activate the DATE Option for the SUMMARY File.
4	EXIT	Exit Simulation from within an Action Section, this is an OPM Flow specific keyword.
5	PERMFAC	Permeability Factor Reduction Due to Salt Precipitation for OPM Flow's Salt Precipitation model.

No.	Keyword	Comment
6	PVTGW	Gas PVT Properties for Dry Gas with Vaporized Water for OPM Flow's Salt Precipitation model.
7	PVTGWO	Gas PVT Properties for Wet Gas with Vaporized Water and Oil for OPM Flow's Salt Precipitation model.
8	PVTWSALT	Define Brine Water Fluid Properties for Various Regions for the Brine Tracking and Salt Precipitation models.
9	PYACTION	Define Python Based Action Conditions and Command Processing
10	PYEND	End the Definition of a PYINPUT Section
11	PYINPUT	Define the Start of a PYINPUT Section
12	RUNSUM	Activate RSM File Output of the SUMMARY Data.
13	RWGSALT	Water Vaporization versus Pressure and Salt Concentration or OPM Flow's Salt Precipitation model.
14	SALTSOL	Define the Salt Solubility Limit for All Cells for OPM Flow's Salt Precipitation model.
15	SALTVD	Define the Equilibration Salt Concentration versus Depth Tables for the Brine model. Note: Although the data is read by the simulator it is currently ignored.
16	SEPARATE	Activate the Separate RSM File Output Option.
17	SKIPREST	Activate Skipping of Restart Schedule Data (this keyword is now functional, previously it was only recognized by the input deck).
18	WSALT	Define Water Injection Well Salt Concentrations.
19	WSEGVALV	Define Multi-Segment Well Sub-Critical Valve for Inflow Control Device

Notes:

1) Cells colored in green in the "No." column indicate that the keyword is recognized by the input deck parser and fully or partially supported by OPM Flow.
2) Cells colored in orange in the "No." column indicate that the keyword is recognized by the input deck parser but not supported by OPM Flow.
3) Cells colored in red in the "No." column indicate that the keyword is not recognized by the input deck parser and OPM Flow.

Table B.13: New Keywords for the 2020-04 Release

B.8.2 BUG FIXES

The following bug fixes and improvements have been incorporated into this release.

1) Fixed a bug with the PVTSALT keyword that caused the keyword to be set to unrecognized.

B.8.3 KNOWN ISSUES

1) Two phase gas-water models (GAS and WATER only keywords in the RUNSPEC section) will not initialize in OPM Flow, this has been the status for past versions of the simulator as well. The work around is to convert the model to a three phase model (OIL, GAS and WATER keywords in the RUNSPEC section) and modify the PROPS section keywords data accordingly.

2) OPM Flow incorrectly reports that the MAXVALUE keyword is not supported. The warning message is incorrect and the values entered on the keyword will be applied to the named array.

3) For the Brine model the SALTVD is active in this release; however, the values entered via the keyword are ignored and reset to zero. A fix is ongoing but did not make it into the 2020-04 release.

4) The Tracer model is not working. In order to activate the Tracer model the command line variable -enable-tracer-mode should be set to "true"; the simulator will still issue a message saying *Warning: Keyword 'TRACERS' is not supported by flow* but will continue to run. However, the results from the tracer tracking appear to be incorrect and should not be relied upon. The standard results, rates, pressures, saturations, etc., are identical to the comparable no tracer run.

5) The OPERATE keyword EQUATION options of MAXLIM and MINLIM are currently not supported.

6) The RPTSCHED WELLS report does not print the FIELD group production data.

B.8.4 DEVELOPER CHANGES

1) For restart runs the commercial simulator does not consider the historical part of the SCHEDULE section in the input deck, as all necessary well and group information is obtained from the restart file, and the interpretation of the SCHEDULE keywords starts at the restart date in the input deck. Historically, OPM Flow has treated restart runs differently. OPM Flow uses the SCHEDULE section data to initialize the wells and groups, combined with the solution arrays (PRESSURE, SWAT, etc.) from the restart file. There is an ongoing effort to duplicate the same behavior as the commercial simulator to further increase the compatibility between the commercial simulator and OPM Flow. This work is being done in stages to ensure existing models continue to work. See the --sched-restart command line parameter in section *2.2 Running OPM Flow 2023-10 From The Command Line*.

Joakim Hove (Release Manager), Cintia Goncalves Machado, Kai Bao, Tor Harald Sandve, and David Baxendale.

B.9 RELEASE 2019-10

B.9.1 NEW FEATURES

An experimental foam module has been added to OPM Flow 2019-10 release. With this it is possible to simulate certain types of surfactant injection. Such injection stimulates formation of foam to change mobility ratios, and give better reservoir sweep. The implemented foam model treats surfactant transported in the gas phase, and reduces the mobility of that phase depending on the surfactant concentration. In addition to mobility reduction, adsorption to the reservoir rock is included in the model. To test the foam module use the keywords, FOAM, FOAMADS, FOAMMOB, FOAMOPTS, FOAMROCK and WFOAM. The model has not been tested on anything but artificial test cases so far, it is therefore likely to have omissions and bugs. If you try it out, please send feedback by the mailinglist (*opm-request@opm-project.org*), or by raising an issue on GitHub (*https://github.com/OPM*). A simple test case based on SPE1 has been added to the opm-tests repository, in the directory spe1_foam (*https://github.com/OPM/opm-tests/tree/master/spe1_foam*).

First implementation of the ACTIONX facility and associated keywords. The ACTIONX keyword defines a series of conditions that invoke run time processing of ACTION functions and is similar to executing a run time script. This is the general purpose version of the ACTION series of keywords that can apply Boolean conditional tests to variables at the field, group, region, well segment and well levels. The ACTION, ACTIONG, ACTIONR, ACTIONS and ACTIONW keywords are not implemented in OPM Flow and are unlikely to be so, as the ACTIONX keyword implements their functionality with greater flexibility. As this is the first release with this functionality, users should exercise caution using this functionality.

Rock compaction has been implemented via the OVERBURD, ROCK2D, ROCK2DTR, ROCKCOMP, and the ROCKNUM keywords. The facility is activated by the ROCKCOMP keyword in the RUNSPEC section.

OPM Flow now supports output of a common subset of end-point arrays to OPM Flow's *.INIT file. Specifically, OPM Flow outputs the drainage and imbibition curve end-points but does not currently support directionally dependent end-points (e.g., SGCRX-). This version also supports the FILLEPS keyword to output the actual end-points, whether taken from direct assignment in the deck or derived from the corresponding saturation function table. Note that OPM Flow does not currently support the TOLCRIT keyword which will affect *CR end-points derived from the tables. This version also activates the pertinent flags in the *.INIT file's LOGIHEAD vector which means that the ResInsight relative permeability plot is now able to distinguish the scaled from the unscaled curves in a cell even for result sets generated by OPM Flow.

Initial implementation of the well list facility via the WLIST keyword for static well lists has been incorporated into this release. In addition, improve support for the WTEST keyword has been added and the WECON keyword now supports GOR checking.

The issue with the Ubuntu Linux 18.04 LTS (64-bit version only) release that prevented mpirun working with OPM Flow under this version of the operating system has been resolved in this release of OPM Flow.

There is an on going effort to recognize all known black-oil keywords by the OPM Flow input deck parser, and to document all these keywords with varying level of detail depending on the functionality implemented in OPM Flow.

In addition the following new keywords are now recognized with varying functionality as described by the comments in Table B.14.

No.	Keyword	Comment
1	ACTION	Define Action Conditions and Command Processing (Field).
2	ACTIONG	Define Action Conditions and Command Processing (Groups).
3	ACTIONR	Define Action Conditions and Command Processing (Regions).
4	ACTIONS	Define Action Conditions and Command Processing (Well Segments).
5	ACTIONW	Define Action Conditions and Command Processing (Wells).

No.	Keyword	Comment
6	ACTIONX	Define Action Conditions and Command Processing.
7	ACTPARAM	Define Action Facility Target and Tolerance Parameters.
8	ENDACTIO	End the Definition of ACTION Commands.
9	UDADIMS	Define the Dimensions of the User Defined Arguments.
10	UDQ	Declare User Define Quantities (“UDQ”).
11	UDQDIMS	Define the Dimensions of the User Defined UDQ Feature.
12	UDQPARAM	Define Parameters for the User Defined Quantity Feature.
13	UDT	Declare User Define Tables (“UDT”).
14	UDTDIMS	Define the Dimensions of the User Defined Tables.
15	FILLEPS	Activate Saturation End Point Export to INIT File.
16	FOAM	Activate the Foam Phase and Model.
17	FOAMADS	Define Foam Rock Adsorption Tables.
18	FOAMDCYO	Define Foam Decay versus Oil Saturation Tables.
19	FOAMDCYW	Define Foam Decay versus Water Saturation Tables.
20	FOAMFCN	Define Foam Gas Mobility Reduction versus Capillary Number.
21	FOAMFRM	Define Foam Gas Mobility Reduction versus Reference Mobility.
22	FOAMFSC	Define Foam Gas Mobility versus Surfactant Concentration Functions.
23	FOAMFSO	Define Foam Gas Mobility Reduction versus Oil Saturation.
24	FOAMFST	Define Foam Gas-Water Surface Tension versus Surfactant Concentration.
25	FOAMFST	Define Foam Gas-Water Surface Tension versus Surfactant Concentration.
26	FOAMMOB	Define Foam Gas Mobility versus Foam Concentration Tables.
27	FOAMMOBP	Define Foam Mobility Reduction versus Oil Pressure.
28	FOAMMOBS	Define Foam Mobility Reduction versus Shear.
29	FOAMOPTS	Define Foam Model Options.
30	FOAMROCK	Define Foam Rock Properties.
31	OVERBURD	Define Rock Overburden Pressure versus Depth Tables.
32	ROCK2D	Pore Volume Compaction versus Pressure and Sw Tables.
33	ROCK2DTR	Transmissibility Compaction versus Pressure and Sw Tables
34	ROCKCOMP	Activate Rock Compaction.
35	ROCKFRAC	Define the Rock Volume to Bulk Volume Fraction for All the Cells.
36	ROCKNUM	Define Rock Compaction Table Region Numbers.

No.	Keyword	Comment
37	ROCKOPTS	Define Rock Compaction and Compressibility Options.
38	ROCKPAMA	Define Coal Palmer-Mansorri Rock Model Parameters.
39	ROCKTAB	Rock Compaction Tables.
40	ROCKTABH	Rock Compaction Hysteresis Tables.
41	ROCKTABW	Rock Compaction Tables (Water Induced).
42	ROCKWNOD	Water Saturation Values for Compaction Pressure-Sw Tables.
43	WFOAM	Define Well Foam Injection Concentrations.
44	WLIST	Define Well Lists (Static).
45	WLISTARG	Modify Well List Target and Constraint Values (Static).
46	WLISTNAM	Define Well Lists (WLISTARG).

Notes:

1) Cells colored in green in the "No." column indicate that the keyword is recognized by the input deck parser and fully or partially supported by OPM Flow.
2) Cells colored in orange in the "No." column indicate that the keyword is recognized by the input deck parser but not supported by OPM Flow.
3) Cells colored in red in the "No." column indicate that the keyword is not recognized by the input deck parser and OPM Flow.

Table B.14: New Keywords for the 2019-10 Release

B.9.2 BUG FIXES

The following bug fixes and improvements have been incorporated into this release.

1) Restart values on the RESTART file are now only read once, previously this was done twice.
2) Fixed several bugs concerning the input and output of RESTART files.
3) EBOS now logs output to both the *.PRT and *.DBG files.
4) OPM Flow now abort a run without reading the deck if the command line parameters are incorrect.
5) Use grid region mapping from opm-grid.
6) Fixed a bug related to negative THP values when extrapolating values from VFP tables.
7) Printing of logging information from Well Testing is now written to both the *.PRT and *.LOG files.
8) Several bug fixes to multi-segement well model have been implemented.
9) Both the *.INIT and *.GRID files are now written out on a restart run.
10) OPM Flow now does not update RESV variable for producers in prediction mode.
11) Fixed an issue with the simulator over writing the FPR summary vector instead of writing out the FPRP summary vector instead..
12) The simulator now always writes out the transmissibilities between vertical neighbors to TRANZ (even for non-neighbor connections).

B.9.3 DEVELOPER CHANGES

For the 2019-10 release, the module "ewoms" has been renamed "opm-models". The repository on github has been renamed, but the old name will continue to work for some time. The figure below shows the current module structure for the 2019.10 release.

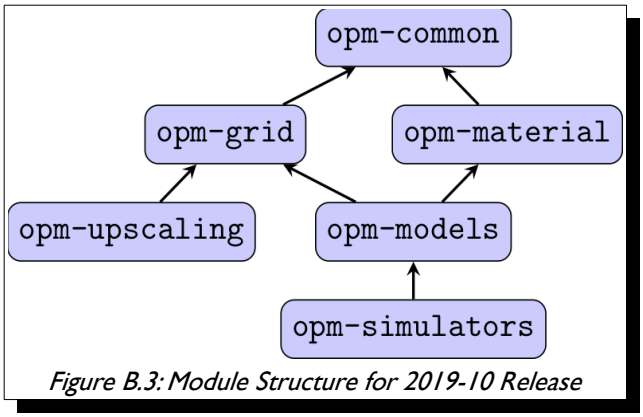


Figure B.3: Module Structure for 2019-10 Release

Most files/header that were located in directory opm/autodiff in opm-simulators have been moved to opm/simulators/aquifers, opm/simulators/linalg, opm/simulators/utils, or opm/simulators/wells depending on their content.

Markus Blatt, Atgeirr Flø Rasmussen, Bård Skaflestad, Tor Harald Sandve, Arne Morten Kvarving and David Baxendale.

B.10 RELEASE 2019-04

Error message reporting has been significantly improved for this release. Previously, when OPM Flow found an error in the input deck, an exception was thrown immediately and the program terminated after writing out an error message. In many cases there are multiple errors in an input deck, but only the first will be reported. In this release all errors are collected and OPM Flow continues until the input deck has been completely assembled. If there have been errors during processing all error messages are now written to the standard output files, after which the program will terminate. This should greatly improve debugging of OPM Flow input deck. The feature is activated by the command line option:

```
flow --strict-mode=true CASE.DATA
```

See section *2.2Running OPM Flow 2023-10 From The Command Line* for additional information.

In addition the following new keywords are now recognized with varying functionality as described by the comments in Table B.15.

No.	Keyword	Comment
1	AQUFETP	The AQUFETP keyword defines Fetkovich Analytical aquifers and the aquifer properties.
2	DRSDTR	DRSDTR defines the maximum rate at which the solution gas-oil ratio (Rs) can be increased in a grid cell for various regions in the model.
3	DRVDTR	DRVDTR defines the maximum rate at which the solution oil-gas ratio or condensate-gas ratio (Rv) can be increased in a grid cell for various regions in the model.
4	FILEUNIT	The FILEUNIT keyword defines the units of the of the data set, and is used to verify that the units in the input deck and any associated include files are consistent grid data.
	GDFILE	The GDFILE keyword loads a GRID file that contains the structural data for the grid as a set of topological cuboidal cells, and EGRID files that contain structural and property data. Added in 2018-10 but didn't work, this has been fixed in 2019-04 and is fully functional.
5	FLUXTYPE	Recognized by the input deck parser only.
6	ISGLPC	ISGLPC defines the imbibition connate gas saturation for all the cells in the model via an array when the end-point scaling option has been invoked. Recognized by the input deck parser, and internalised into OPM Flow's 3D property representations. Missing Some Functionality - Use with Caution.
7	ISWLPC	ISWLPC defines the imbibition connate water saturation for all the cells in the model via an array when the end-point scaling option has been invoked. Recognized by the input deck parser, and internalised into OPM Flow's 3D property representations. Missing some functionality. Use with caution.
8	MULTIN	This keyword switches on the Multiple Input Files option for all input files.
8	MULTOUT	This keyword switches on the Multiple Output Files option for all output files.
9	OVERBURD	Recognized by the input deck parser and simulator support is available in the experimental "ebos" simulator.
10	ROCK2D	Recognized by the input deck parser and simulator support is available in the experimental "ebos" simulator.

No.	Keyword	Comment
11	ROCK2DTR	Recognized by the input deck parser and simulator support is available in the experimental "ebos" simulator
12	ROCKWNOD	Recognized by the input deck parser and simulator support is available in the experimental "ebos" simulator.
13	SGLPC	SGLPC defines the connate gas saturation for all the cells in the model via an array when the end-point scaling option has been invoked. Recognized by the input deck parser, and internalised into OPM Flow's 3D property representations. Missing some functionality. Use with caution.
14	STONE	This keyword is an alias for STONE2 keyword that activates Stone's second three phase oil relative permeability model as modified by Aziz and Settai. Keyword was previously supported by OPM Flow but not recognized by the input deck parser. This has now been fixed and the keyword is fully functional as per the documentation in the manual.
15	STONE2	This keyword activates Stone's second three phase oil relative permeability model as modified by Aziz and Settar. Keyword was previously supported by OPM Flow but not recognized by the input deck parser. This has now been fixed and the keyword is fully functional as per the documentation in the manual.
16	SWLPC	SWLPC defines the connate water saturation for all the cells in the model via an array when the end-point scaling option has been invoked. Recognized by the input deck parser, and internalised into OPM Flow's 3D property representations. Missing some functionality. Use with caution.
17	TBLK	Recognized by the input deck parser and simulator support is available in the experimental "ebos" simulator
18	THPRESFT	Recognized by the input deck parser and simulator support is available in the experimental "ebos" simulator.
19	WLIST	WLIST declares a group of wells to belong to a named well list.

Notes:

1) Cells colored in green in the "No." column indicate that the keyword is recognized by the input deck parser and fully or partially supported by OPM Flow.
2) Cells colored in orange in the "No." column indicate that the keyword is recognized by the input deck parser but not supported by OPM Flow.
3) Cells colored in red in the "No." column indicate that the keyword is not recognized by the input deck parser and OPM Flow.

Table B.15: New Keywords for the 2019-04 Release

Bård Skaflestad, Tor Harald Sandve, and David Baxendale.

B.11 RELEASE 2018-10

The Open Porous Media project is please to announce that version 2018.10 of the OPM suite of simulation software has been released. Binary packages for Red-Hat Enterprise Linux 6 and 7 and Ubuntu 16.04 and 18.04 are available for download at the OPM website download page *https://opm-project.org/?page_id=36.* Installation instructions can also be found at the download page as well as in section *CHAPTER 2: INSTALLING AND RUNNING FLOW* of this manual.

In addition to the usual list of fixes and improvements two significant improvements include:

1) OPM Flow can now write restart files which can be used to restart runs using the commercial simulator.

2) Performance has been improved significantly, on selected field models, where OPM Flow is significantly faster than the commercial simulator.

Due to problems with the Zoltan package we have unfortunately been forced to disable MPI for the Ubuntu 18.04 package. We are looking into this, and if possible we will make updated packages at a later stage.

In terms of new features OPM Flow now includes an option for thermal modeling. The energy black-oil implementation in OPM Flow is a mixture of the commercial simulators black-oil and the commercial simulators "compositional thermal" keywords, as well as some OPM Flow specific keywords. The temperature option (TEMP keyword) and the thermal option (THERMAL keyword) are two separate modeling facilities in the commercial simulator. OPM Flow's thermal implementation is based on solving the energy equation fully coupled with the black-oil equations so the results are not directly equivalent to commercial simulator's black-oil TEMP or compositional THERMAL formulations. See the *5.3.141 THERMAL– Activate the Thermal Modeling Option* keyword in the RUNSPEC section outlining the available keywords.

Apart form the new thermal keywords summarized in section *5.3.141 THERMAL– Activate the Thermal Modeling Option*, the following new keywords have been incorporated in this release and are active:

No.	Keyword	Comment
1	GDFILE	The GDFILE keyword loads a GRID file that contains the structural data for the grid as a set of topological cuboidal cells, and EGRID files that contain structural and property data. Fully functional – Not Working in this Release
2	PLMIXNUM	The PLMIXNUM keyword defines the polymer region number for each grid block that is used to assign the mixing tables as well as the maximum polymer and salt concentrations, for when the polymer option has been activated.
3	TOLCRIT	This keyword defines the Critical Saturation Tolerance.
4	ISGLPC	The WSEGSICD keyword defines a multi-segment well segment to be a spiral Inflow Control Device. ("ICD") as part of a completion for a multi-segment well.

Notes:

1) Cells colored in green in the "No." column indicate that the keyword is recognized by the input deck parser and fully or partially supported by OPM Flow.
2) Cells colored in orange in the "No." column indicate that the keyword is recognized by the input deck parser but not supported by OPM Flow.
3) Cells colored in red in the "No." column indicate that the keyword is not recognized by the input deck parser and OPM Flow.

Table B.16: New Keywords for the 2018-10 Release

Joakim Hove

B.12 RELEASE 2018-04

The Open Porous Media project is please to announce that version 2018.04 of the OPM suite of simulation software has been release. Installation instructions can found on OPM website download page *https://opm-project.org/?page_id=36* and in section *CHAPTER 2: INSTALLING AND RUNNING FLOW* of this manual. Packages for Ubuntu 16.04 and Red-Hat Enterprise Linux 6 and 7 is prepared. New in this release is that also the Red-Hat packages are shipped with mpi support.

In addition to various minor bug-fixes and a reorganization of the code, the release contains new functionality for the OPM Flow simulator like DRSDT support, experimental CPR preconditioning and enhanced output capability. Note that number of modules are reduced to simplify the build process and the code maintenance. The new module organization is shown opposite.

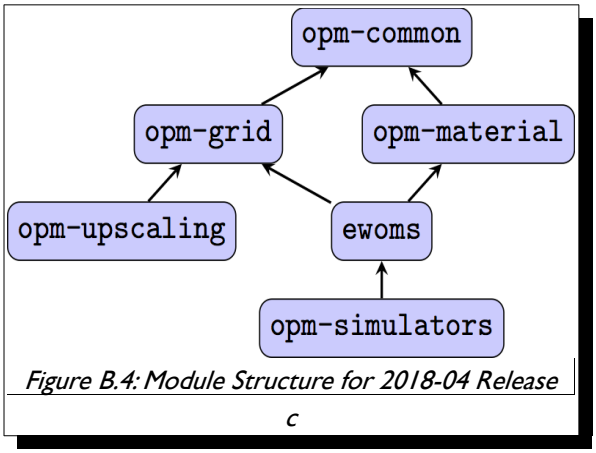


Figure B.4: Module Structure for 2018-04 Release

Since the 2017.10 release the opm-core module has been removed, and the modules opm-parser and opm-output have been folded into opm-common.

In terms of new features OPM Flow now includes analytical aquifers using the Carter-Tracy analytical aquifer and the ability model multi-segment wells. The following new keywords have been incorporated in this release and are active:

No.	Keyword	Comment
1	AQUANCON	AQUANCON keyword defines how analytical aquifers are connected to the simulation grid., this includes Carter-Tracy and Fetkovich analytical aquifers.
2	AQUCT	The AQUCT keyword defines Carter-Tracy aquifers, the properties of the aquifer, including the Carter-Tracy aquifer influence function associated with the aquifer.
3	AQUDIMS	The AQUDIMS keyword defines the dimensions of the various aquifer property data.
4	AQUTAB	AQUTAB keyword defines additional Carter-Tracy aquifer functions to be used in the model.
5	COMSEGS	The COMSEGS keyword defines how a multi-segment well is connected to the reservoir by defining or modifying existing well connections..
6	DRSDT	DRSDT defines the maximum rate at which the solution gas-oil ratio (Rs) can be increased in a grid cell.
7	DRVDT	DRVDT defines the maximum rate at which the solution oil-gas ratio or condensate-gas ratio (Rv) can be increased in a grid cell.
8	WELSEGS	The WELSEGS keyword defines a well to be a multi-segment well and defines the well's segment structure.
9	WSEGSDIMS	The WSEGSDIMS keyword defines the multi-segment well dimensions for the multi-segment well model.

No.	Keyword	Comment
Notes: 1) Cells colored in green in the "No." column indicate that the keyword is recognized by the input deck parser and fully or partially supported by OPM Flow. 2) Cells colored in orange in the "No." column indicate that the keyword is recognized by the input deck parser but not supported by OPM Flow. 3) Cells colored in red in the "No." column indicate that the keyword is not recognized by the input deck parser and OPM Flow.		

Table B.17: New Keywords for the 2018-04 Release

Tor Harald Sandve

B.13 RELEASE 2017-10 UPDATE 1

We have created an update for the 2017.10 release. It fixes a few bugs in Flow that could affect simulation results slightly and also lead to very bad performance when running Flow in parallel using MPI. The effect was most pronounced when running with more than 4 MPI processes.

Binary packages for Ubuntu 16.04 and Red Hat Enterprise Linux 6 and 7 have been updated, and for most users your systems will ask to install the updated version or do it automatically.

For those who compile OPM from source, the release branches on GitHub have been updated and tagged with

```
release/2017.10/update1
```

The master branch of course includes the same fixes.

Atgeirr Flø Rasmussen

B.14 RELEASE 2017-10

On behalf of the OPM project, I'm happy to announce that version 2017.10 has been released. Packages for Ubuntu 16.04 and Red-Hat Enterprise Linux 6 and 7 have been prepared or should be available soon.

As usual, this release contains a multitude of new features and improvements. Most notable are probably that the Flow simulator is now considerably faster than in the 2017.04 release, Flow now supports the solvent and polymer black-oil extensions and that there now is freely available documentation for the file format that is used to specify the input.

Finally, as the release manager, I'd take the opportunity and thank everyone involved in making the release process for 2017.10 go as smoothly as it did.

Andreas Lauser

B.15 RELEASE 2017-04

The Open Porous Media project is glad to announce that version 2017.04 of the OPM suite of simulation software has been released! Installation instructions can found on our download page.

Certainly the most significant change of this release is the introduction of the "flow_ebos" simulator. Compared to the previous "flow" simulators, "flow_ebos" uses a different approach to linearize the non-linear system of partial differential equations and as a result exhibits significantly better performance. The new simulator is intended to eventually fully replace the current family of "flow" simulators (i.e., "flow", "flow_mpi", "flow_solvent", "flow_polymer", etc.) and should already provide a proper superset of the capabilities of the "flow" simulator of previous OPM releases. For this reason, the name "flow" has been made an alias for "flow_ebos" in OPM 2017.04. If, for some reason, the previous "flow" simulator must be used, it is still shipped under the name "flow_legacy", but we strongly encourage you to send us bug reports if you encounter any case that can be simulated using "flow_legacy" but not using "flow_ebos".

Besides the introduction of "flow_ebos", plenty unit tests have been added, a plethora of bugs has been fixed, well handling has been considerably improved and now supports e.g. top-hole pressure controls and vertical flow performance tables, ECL output and restart capabilities have been made much more comprehensive, and all grid related functionality of opm-core has been moved to the opm-grid module in preparation of the former module's eventual retirement. In addition, a Docker container has been uploaded to Docker Hub to ease deployment for people who are into container technologies.

Last but not least, I would like to thank everyone who contributed to making the many changes of this release happen so smoothly.

Modules involved in the release (maintainers are given in parenthesis):

- opm-data (Alf Birger Rustad)
- opm-common (Atgeirr Rasmussen, Bård Skaflestad, Arne Morten Kvarving, Joakim Hove, Robert Klöfkorn, Tor Harald Sandve, and Andreas Lauser)
- opm-parser (Joakim Hove)
- opm-output (Joakim Hove)
- opm-grid (Atgeirr Rasmussen, Robert Klöfkorn, and Bård Skaflestad)
- opm-material (Andreas Lauser, Robert Klöfkorn, and Tor Harald Sandve)
- opm-core (Atgeirr Rasmussen, Robert Klöfkorn, and Bård Skaflestad)
- ewoms (Andreas Lauser, Robert Klöfkorn, and Tor Harald Sandve)
- opm-simulators (Atgeirr Rasmussen, Robert Klöfkorn, Tor Harald Sandve, and Andreas Lauser)
- opm-upscaling (Arne Morten Kvarving, Atgeirr Rasmussen, and Bård Skaflestad)

Andreas Lauser

APPENDIX C: OPMRUN – FLOW JOB SCHEDULER

APPENDIX TABLE OF CONTENTS

APPENDIX INDEX OF FIGURES

C.1 INTRODUCTION

OPMRUN is graphical user interface to Flow that has similar functionality to the commercial simulator's ECLRUN program. Target audience are Reservoir Engineers in a production environment. Developers and experienced Linux users will already have compatible work flows. The application performs the following:

- Allows editing and management of OPM Flow's run time parameters. Default parameters are automatically loaded from OPM Flow, and the user can reset the default parameter set either from a parameter or PRT file. Editing of a job's parameter file is also available.
- Runs under Linux and Windows 10. For Windows 10 OPM Flow is run via the Windows Subsystem for Linux ("WSL").
- Allows simulation jobs to be queued and run in either foreground (under OPMRUN), or in background in a xterm terminal session in Lunux or WSL under Windows 10. Jobs in the queue can be set to run in NOSIM mode or RUN mode.
- Foreground jobs can be killed from OPMRUN, with the option of killing all the jobs in the queue.
- Queues can be edited, saved and loaded.

Various additional simulation input generation and conversion utilities are available including:

- Compressing a job to save space (DATA, and all OPM Flow output files) and uncompressing previously compressed jobs,
- Keywords, a keyword generator based on the Apache Velocity Template Language ("VTL"). The templates can therefore also be used with any editor that supports VTL, jEdit for example. There is one template per keyword, with the formatting the same as the OPM Flow manual. Over 450 templates are currently implemented. One can also customize the existing templates as well as creating User defined templates. The keywords are examples, one still has to edit the resulting deck with the actual required data, but the format with comments should make this a straight forward process.
- A Production Schedule application that takes a comma delimited CSV file containing historical production and injection data and converts the data to an OPM Flow SCHEDULE file using the WCONHIST series of keywords. Currently only production data is supported.
- Sensitivities application that generates sensitivity cases based on a "Base" case file. The Base file contains "Factors" (variable names), $X01, $X02, etc., that are substituted with user defined values using the data entered and the type of Sensitivity Scenario selected.
- A Well Specification application that uses the standard well export files from OPM ResInsight to reformat the data in a more user-friendly manner for the WELSPECS and COMPDAT keywords. Optionally, the application can generate the COMPLUMP keyword based on the OPM ResInsight layers file, with one completion per defined reservoir layer.
- Calling OPM ResInsight and loading the currently selected job into OPM ResInsight for viewing.
- A Well Trajectory Conversion application that converts a Schlumberger Petrel exported well trajectory file into an OPM ResInsight file, containing all the wells.

The software can be downloaded from the following link:

https://github.com/OPM/opm-utilities/tree/master/opmrun

OPMRUN is written in Python 3 and tested under various Ubuntu distributions. Note that only Python 3 is supported and tested, Python2 support has been deprecated.

The program requires the following standard module Python libraries:

- datetime, getpass, importlib, os, numpy, pkg_resources, pandas, pathlib, platform, psutil, sys, re, subprocess, and tkinter as tk.

In addition, the following non-standard Python modules are required:

- airspeed, notify-py, pyDOE2, and PySimpleGUI.

For some Linux systems the relevant package manager may have to be used to install *tkinter as tk;* whereas for Windows 10 users the tkinter package is pre-installed with Python.

C.2 OPMRUN FUNCTIONALITY

OPMRUN enables the editing and management of OPM Flow's run time parameters, setting up job queues to run a series of simulation jobs sequentially, as well as the management of the job queues. Figure C.1shows the initial display.

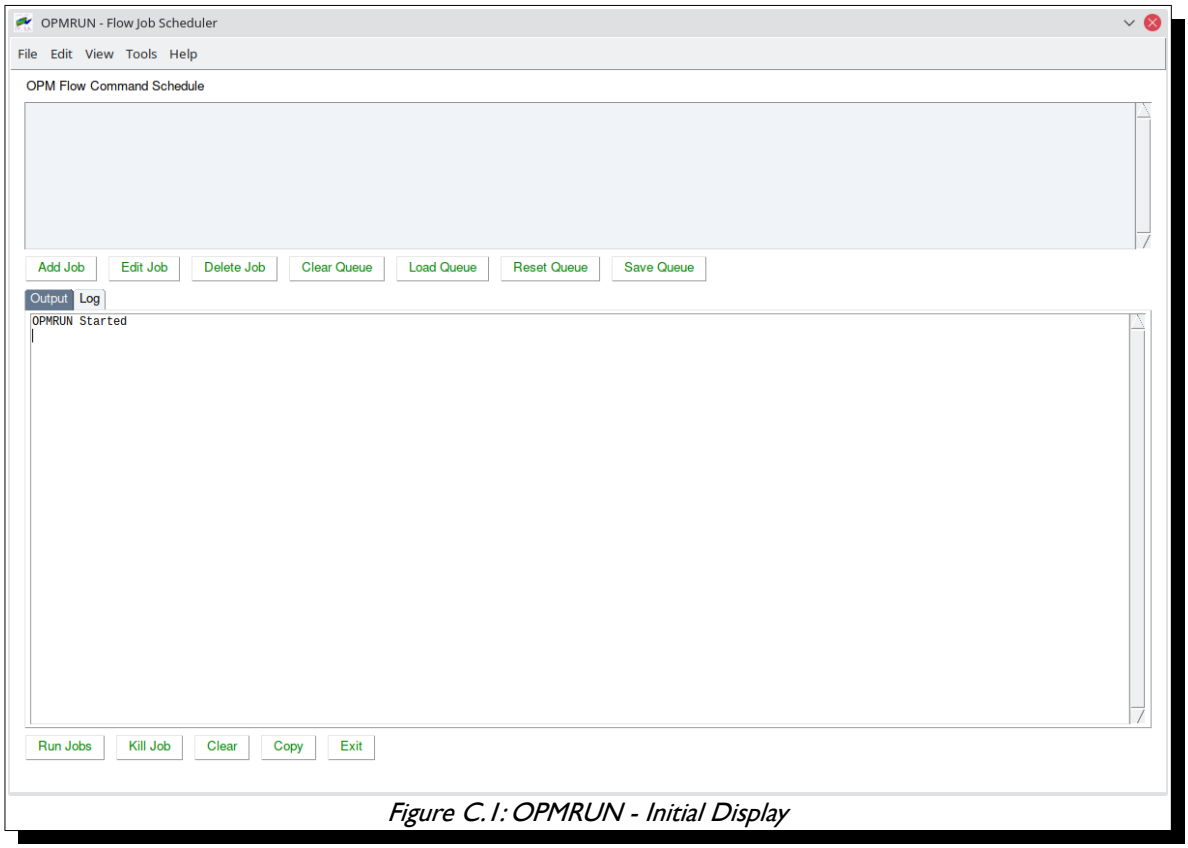


Figure C.1: OPMRUN - Initial Display

Upon launch the program runs OPM Flow to get a list of command line parameters from the current version of OPM Flow. These default parameters can be edited for each case, or alternative default parameter sets can be loaded from an existing parameter file from another job, or a *.PRT file from a completed simulation.

As can be seen in Figure C.2 the program has upper and lower display elements. The upper element shows a list of simulation jobs that are in the job queue and the lower element consists of two elements, one for the OPM Flow Output (the terminal output from OPM Flow) and a second element (OPM Run Log) that is a session log of the jobs run by OPMRUN. Clicking the OPM Flow Output and OPM Run Log tabs switches the display on the lowered element between two display types.

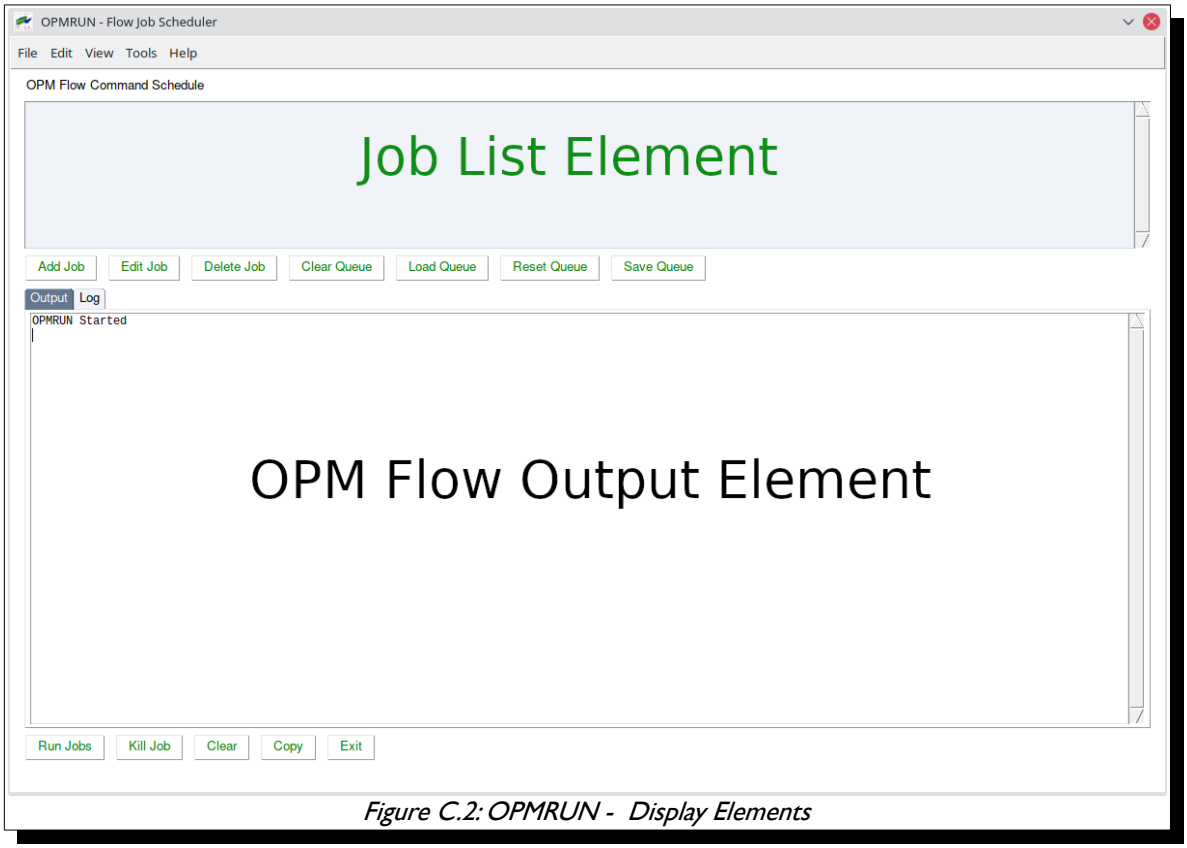


Figure C.2: OPMRUN - Display Elements

C.2.1 ADD JOB AND SELECT RUN TYPE

To add jobs to the queue use the Add Job button or load an existing job queue using the Load Queue button. Jobs can be edited or deleted from the queue using the Edit Job and Delete Job buttons, and a series of jobs can be saved as a job queue by using the Save Queue button. The Clear Queue button deletes all jobs from the queue.

Pressing the *Add Job* button will display the following dialogue box:

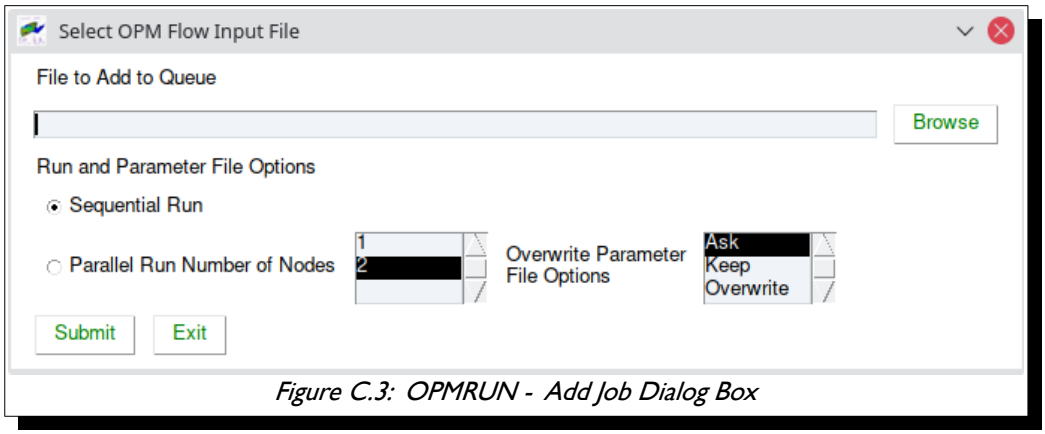


Figure C.3: OPMRUN - Add Job Dialog Box

Use the *Browse* button to select the input file to add to the queue, then select the Run Parameters for this input file, then press the *Submit* button to add the input file to the job queue (Figure C.4).

The Overwrite Parameter File Options allow for different default treatments of existing *.PARAM files, which is particularly useful when adding multiple jobs at the same time.

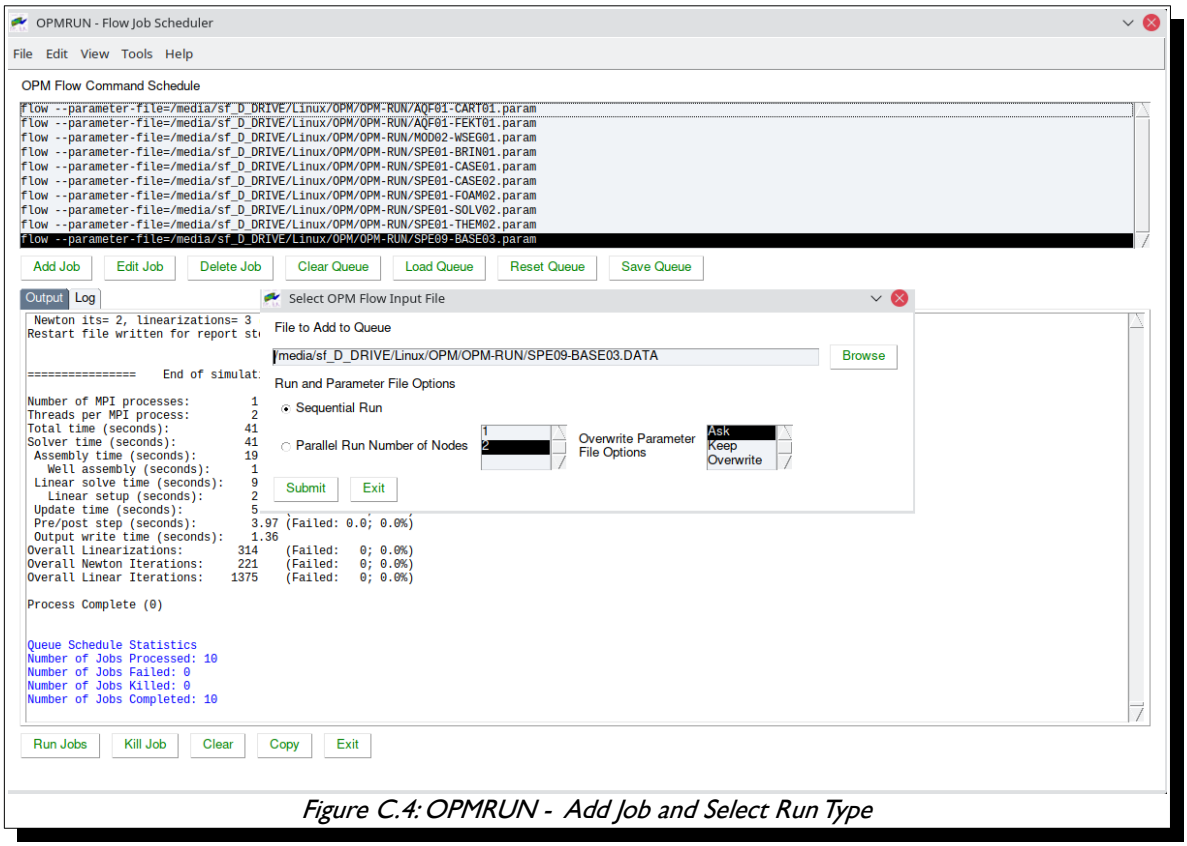


Figure C.4: OPMRUN - Add Job and Select Run Type

The reason for this is because different versions of OPM Flow have different parameter sets and if a newer version of OPM Flow runs with a previous version's *.PARAM file then the simulator will stop with an error message for the various invalid parameters for the current version of the simulator.

C.2.2 EDIT JOB DATA AND PARAMETER FILE

Jobs in the queue can be edited by selecting the *Edit Job* button that will display two options (Figure C.5): one to edit the input file using the defined editor and the second to edit the OPM Flow Parameter File.

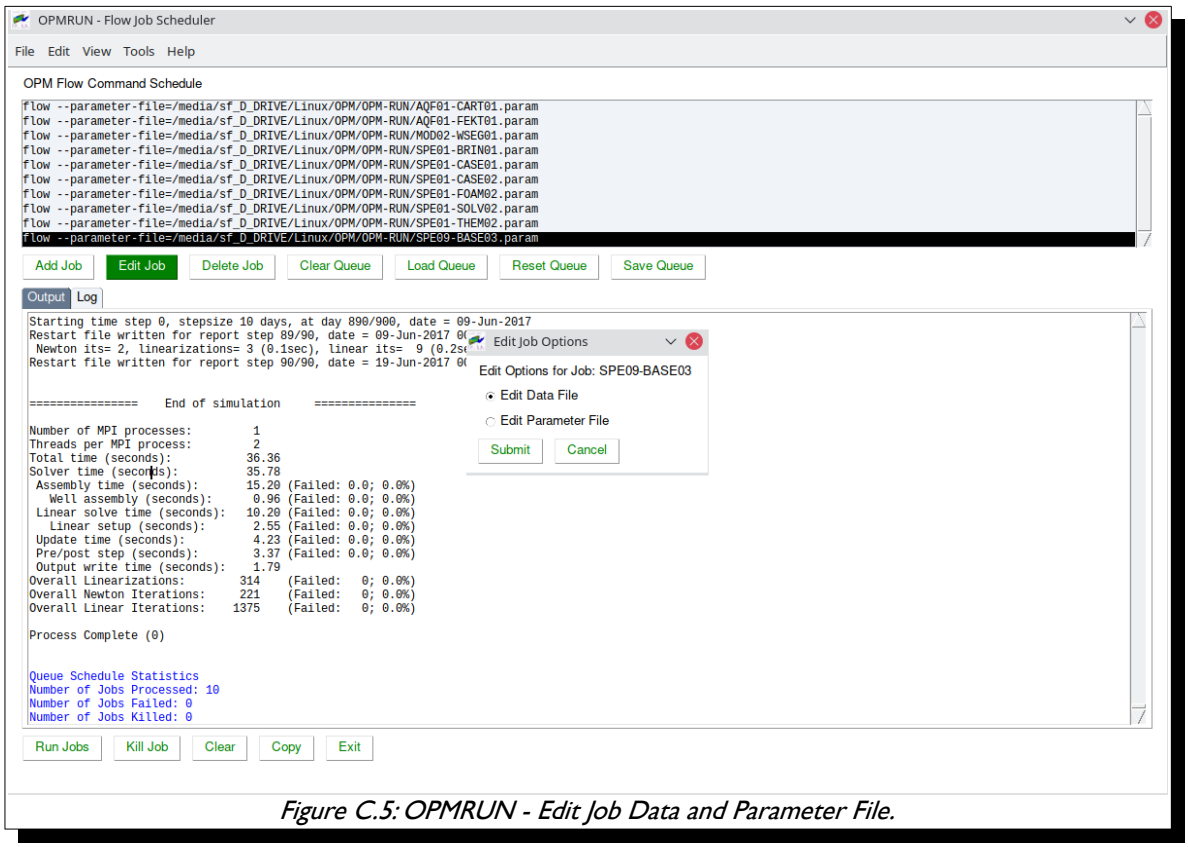


Figure C.5: OPMRUN - Edit Job Data and Parameter File.

If the second option is selected OPMRUN will display a dialog box that shows a list of the OPM Flow command line parameters together with the parameter help information Figure C.7). Selecting a parameter from the list and selecting the Edit button will display the setting for the selected parameter (alternatively one can double click the required entry). One can then edit the parameter as required. Use the Save button to save the change and use the Exit button to save all the changes to the parameter file. The Cancel button will cancel all changes to the parameter file.

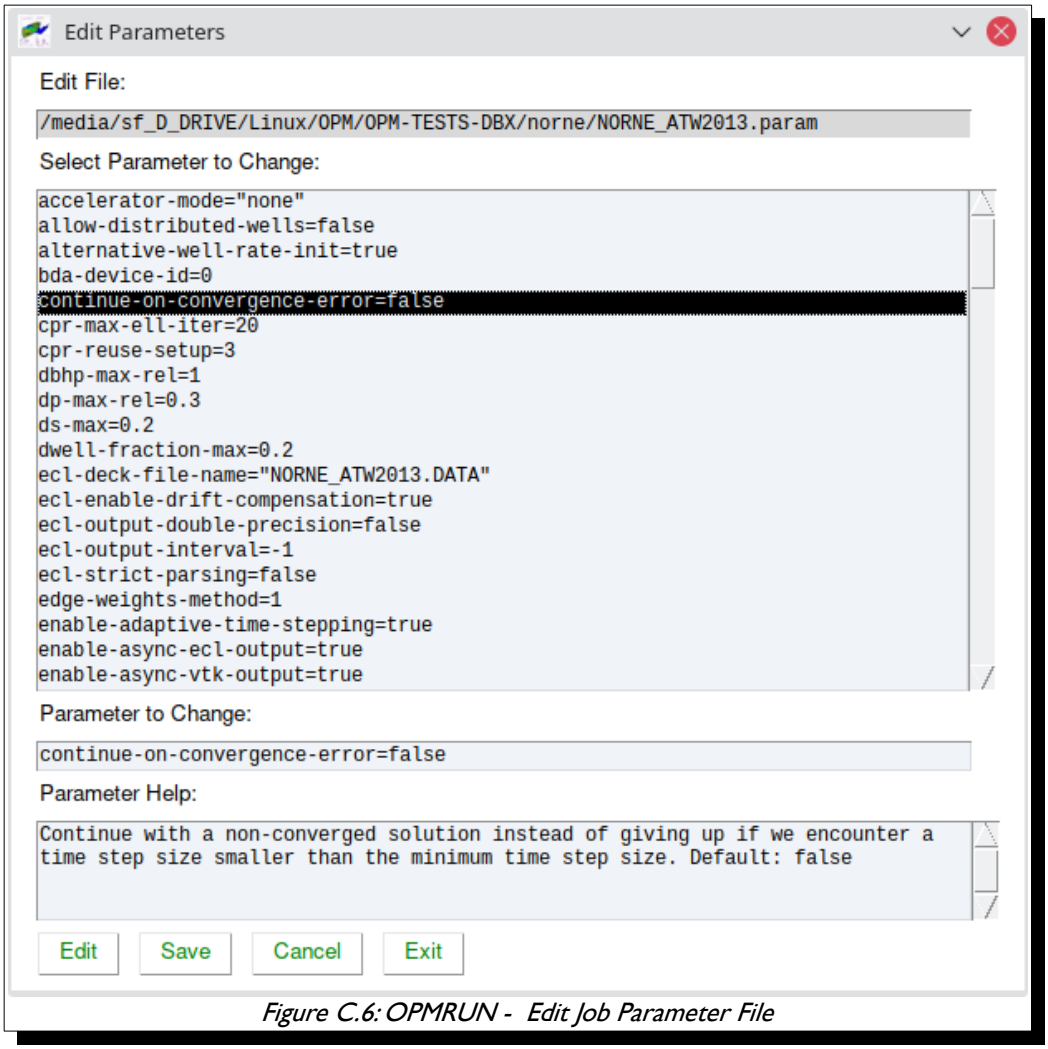


Figure C.6: OPMRUN - Edit Job Parameter File

Alternatively one can use the:

1) Edit OPM Flow Parameter menu option to edit the parameter file for a job.
2) List OPM Flow Parameters menu option to list the commands in the parameter file for a job.
3) Set OPM Flow Default Parameters to set the default parameters for all subsequent jobs added to the queue. This option allows the user to load a default set of parameters from (1) OPM Flow, (2) an OPM Flow Parameter File, or (3) an OPM Flow print file (*.PRT).

One can also right-click on a job and select one of the available options.

C.2.3 LOAD PREVIOUSLY SAVED QUEUE

To load a previously saved job queue, press the *Load Queue* button this will display a dialog box allowing the user to select a queue file (*.que), after pressing the *OK* button the jobs will be displayed in the Job List E*lement* as illustrated in Figure C.7.

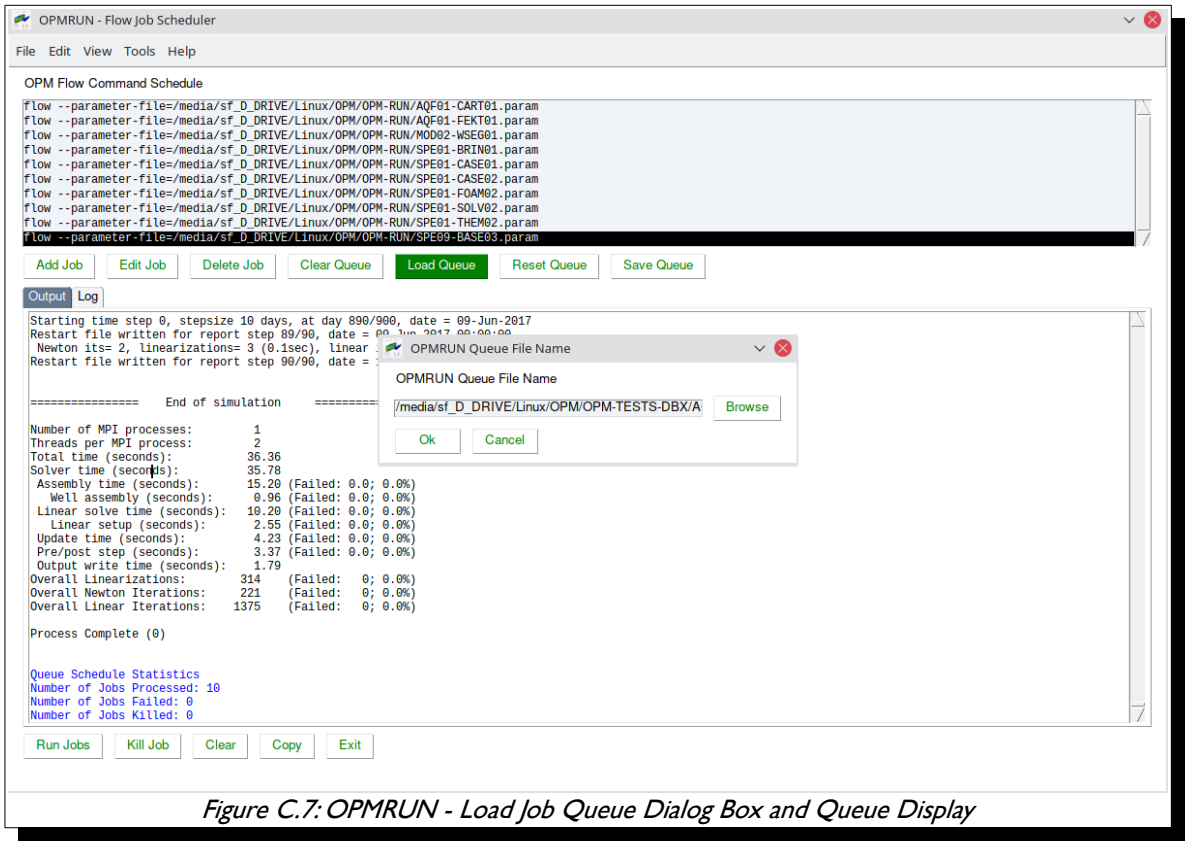


Figure C.7: OPMRUN - Load Job Queue Dialog Box and Queue Display

Queue files allow for various jobs to be load efficiently, especially for ensemble and sensitivity cases and may contain a large number of cases.

Note

When running under Windows 10 the Job names will follow the Windows 10 file naming convention and OPMRUN automatically handles the file names for running the jobs under the Window Subsystem for Linux. However, care is needed for any "included" file in the input deck. In this case the *PATHS – Define Filename Directory Path Aliases* keyword in the RUNSPEC section may be of use.

C.2.4 RESET JOB QUEUE PARAMETERS

Reset Job Queue Parameters allows jobs run under Windows 10 WSL to be renamed for running under Linux, and changing jobs from serial to parallel and vice versa.

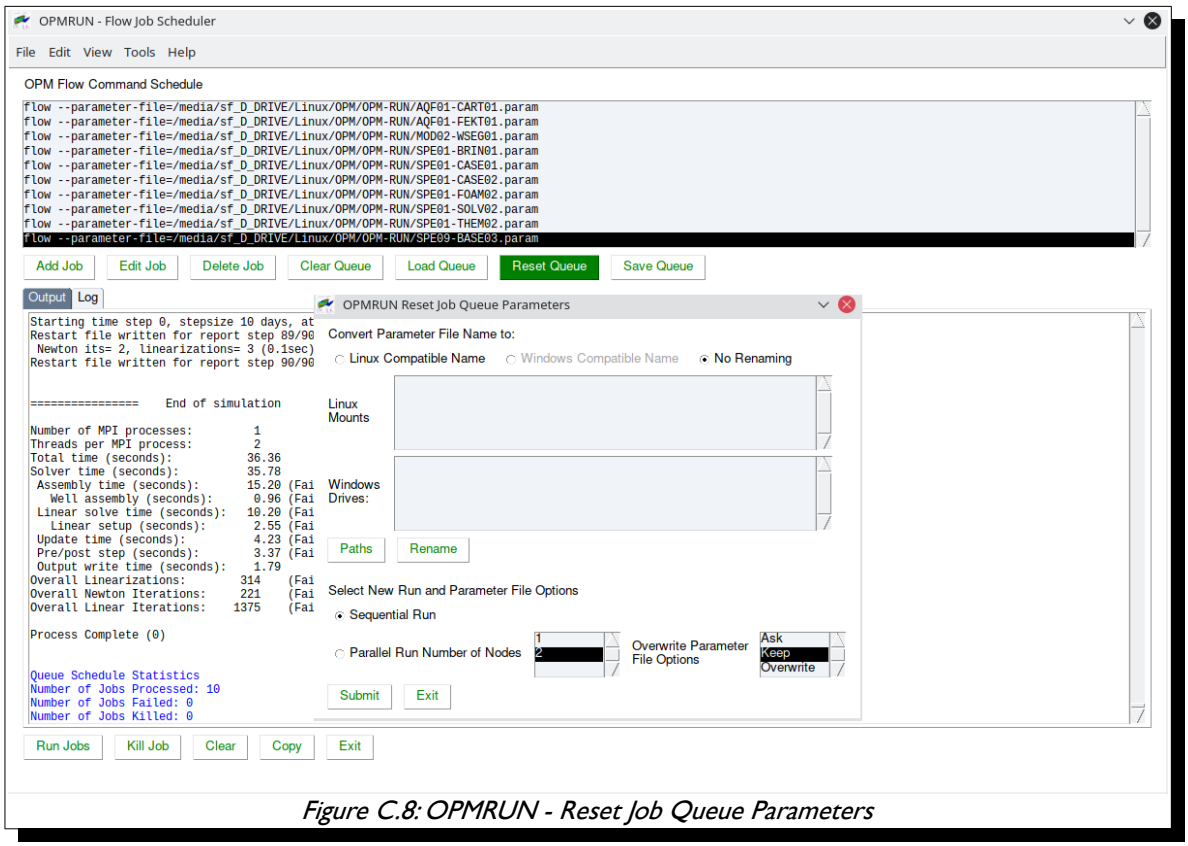


Figure C.8: OPMRUN - Reset Job Queue Parameters

The application will attempt to list or Linux mount points and Windows drives depending on the host operating system, once the two systems mount and drive points have been selected then the files in the queue will be renamed from the previous host system to the current host system. This is only performed for the *.DATA files, included files in the input deck are currently not converted.

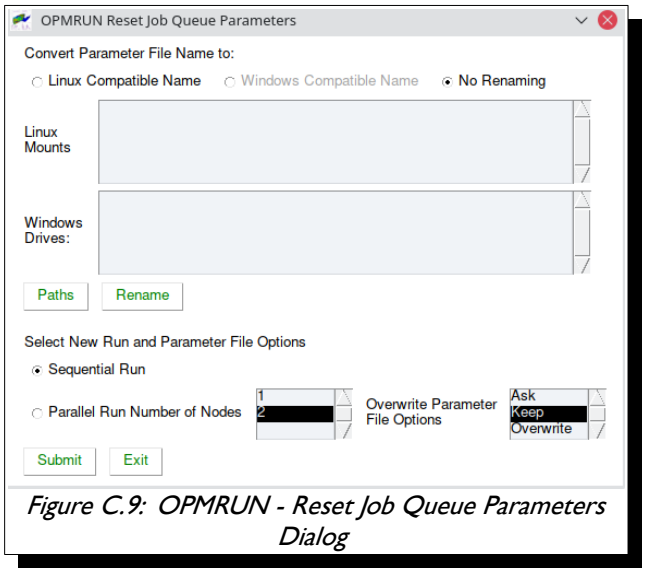


Figure C.9: OPMRUN - Reset Job Queue Parameters Dialog

As previously mentioned, one can also change the queue run time parameters from serial to parallel or vice versa.

C.2.5 RUN JOBS IN QUEUE WITH VARIOUS OPTIONS

Selecting the Run Jobs button displays the Select Run Option dialog box shown in Figure C.10 and Figure C.11.

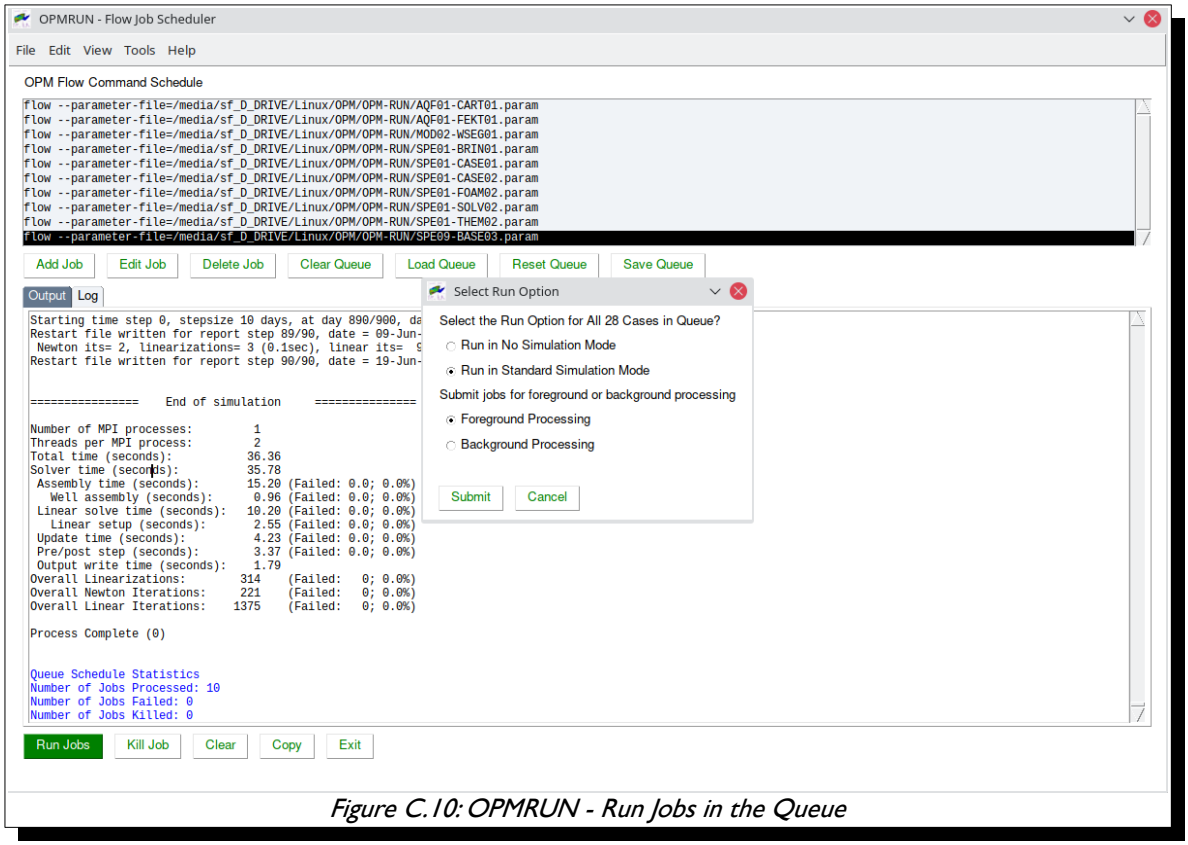


Figure C.10: OPMRUN - Run Jobs in the Queue

On the Select Run Dialog, the Run in No Simulation Mode option is equivalent to setting the NOSIM option in the input deck for all jobs in the queue (see see section *5.3.97 NOSIM – Activate the No Simulation Mode for Data File Checking* and the –enable-dry-run command line parameter in Error: Reference source not found in section *2.2 Running OPM Flow 2019-04 From The Command Line*.This allows for checking all the jobs at once.

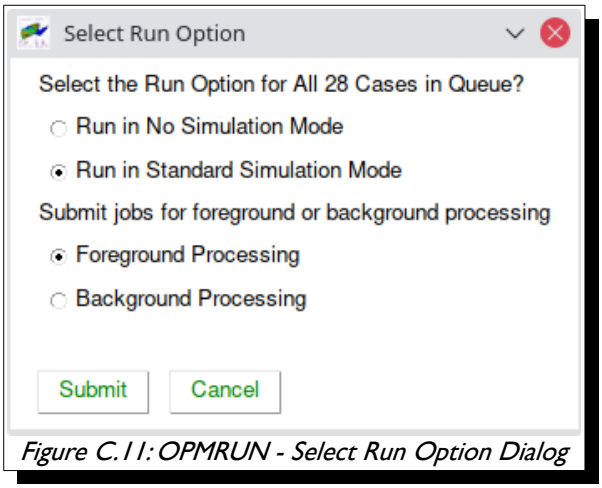


Figure C.11: OPMRUN - Select Run Option Dialog

Selecting Run in Standard Simulation Mode will run all the jobs in the queue sequentially, with the OPM Flow terminal output directed to OPM Flow Output Element, as shown in Figure C.12. The terminal output is also directed to a *.LOG file as well, similar to what the commercial simulator does.

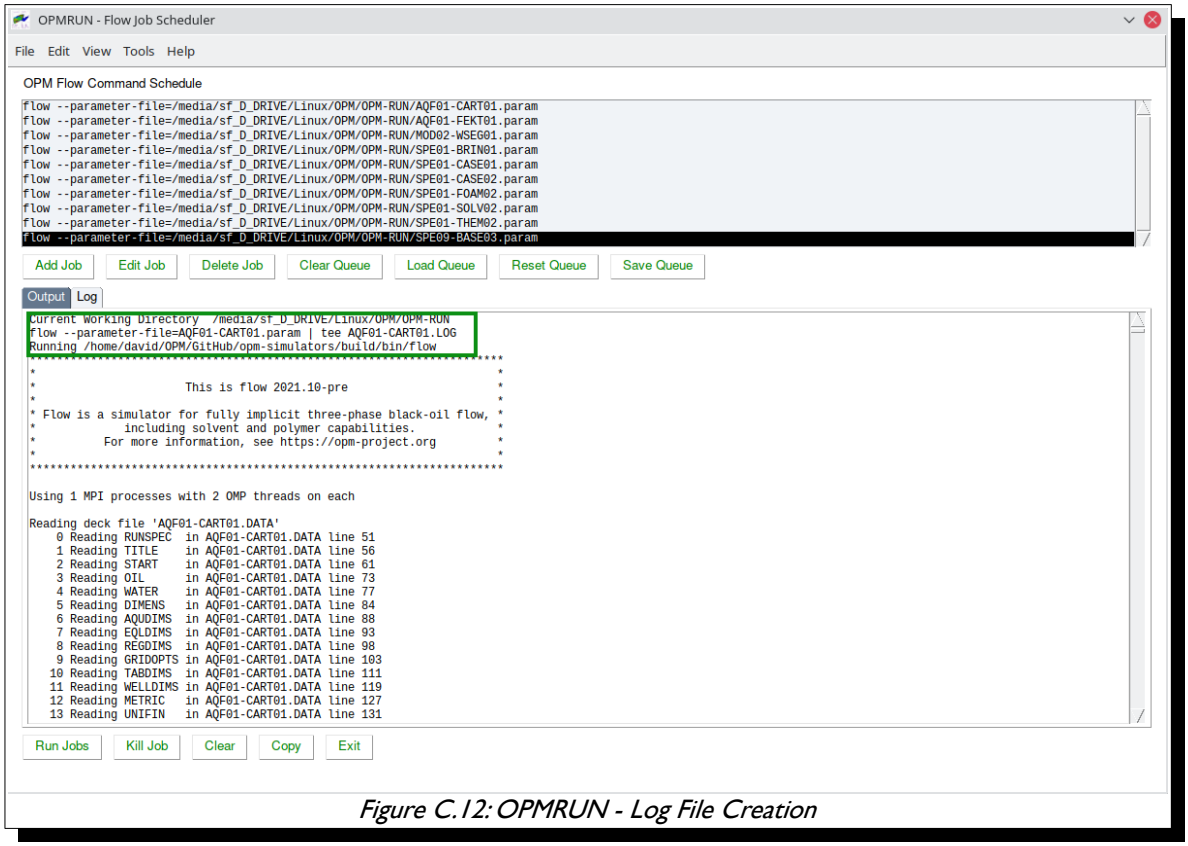


Figure C.12: OPMRUN - Log File Creation

Clicking the OPM Run Log tab displays the OPMRUN's session log file that records the time and date of the major events that have occurred, including the start and end times of each run. Notice also how OPMRUN deletes all the existing output files for a given job, if they exists, before running OPM Flow. as well as creating a Schedule Log for tracking progress (Figure C.13).

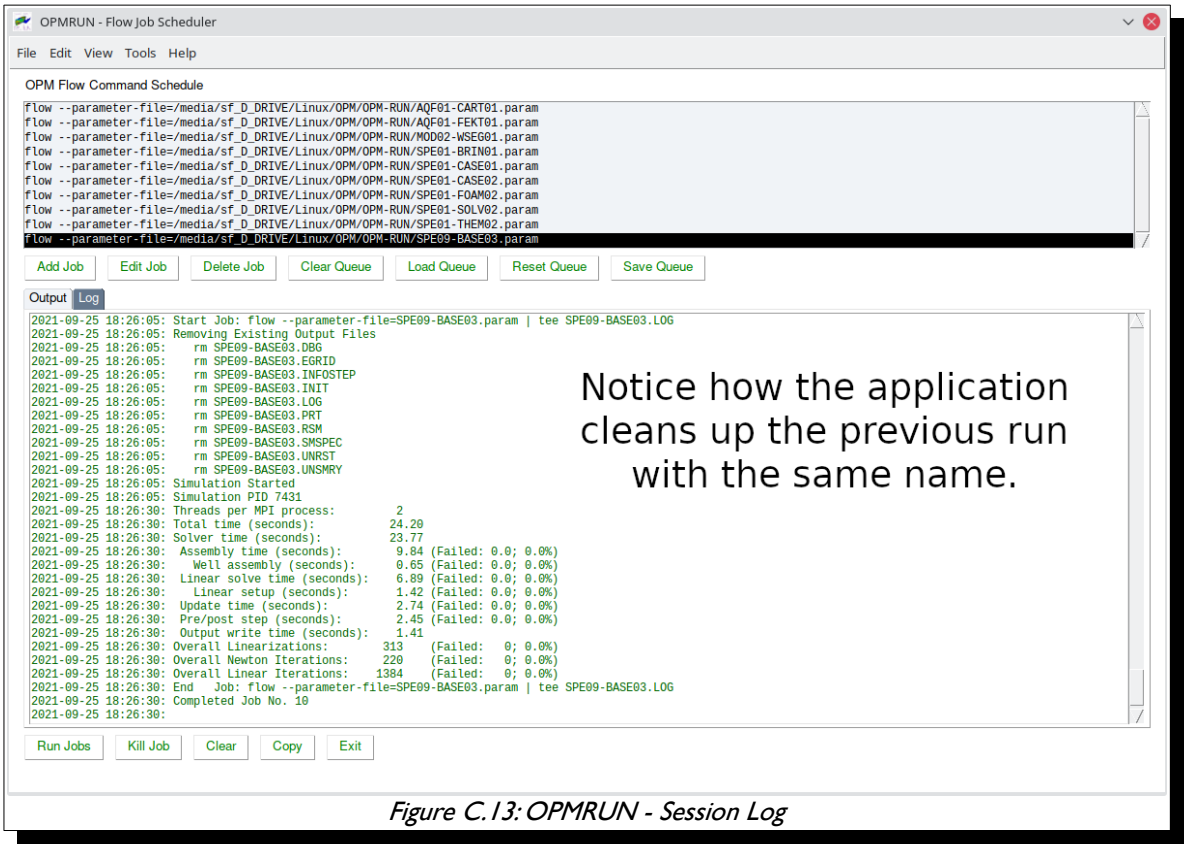


Figure C.13: OPMRUN - Session Log

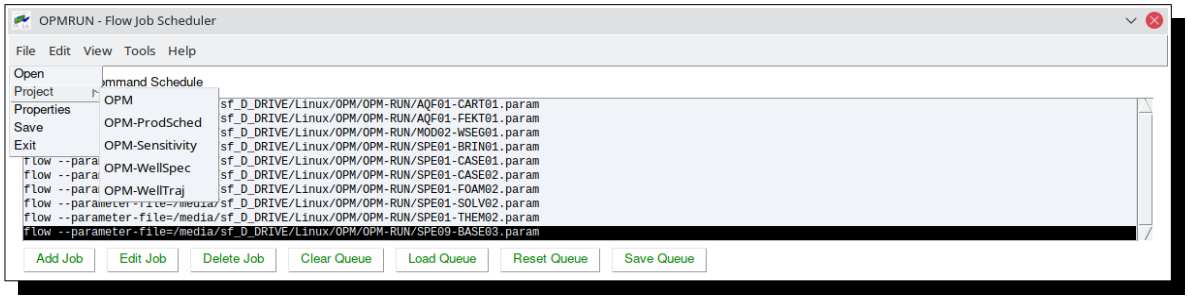
The Kill button will ask the user if the current running job should be killed, and if the job is to killed, the application will prompt as whether or not all the jobs in the queue should be killed.

The *Clear* clears the OPM Flow Output Element from the currently displayed tab (*Output* or *Log*) and the *Copy* button copies the data to the clipboard.

C.2.6 MENU OPTIONS

File Menu Options

Enables open and saving the job queue, switching projects and listing OPMRUN's user properties.



Edit Menu Options

Lets one add jobs, add jobs recursively (all jobs in the selected directory and below), edit the data file (Figure C.14) and the parameter file for the selected job,(see section C.2.2 Edit Job Data and Parameter File) edit, list and set the default parameters for running jobs that will be added to the queue, set OPMRUN options, and set the project's project directories.

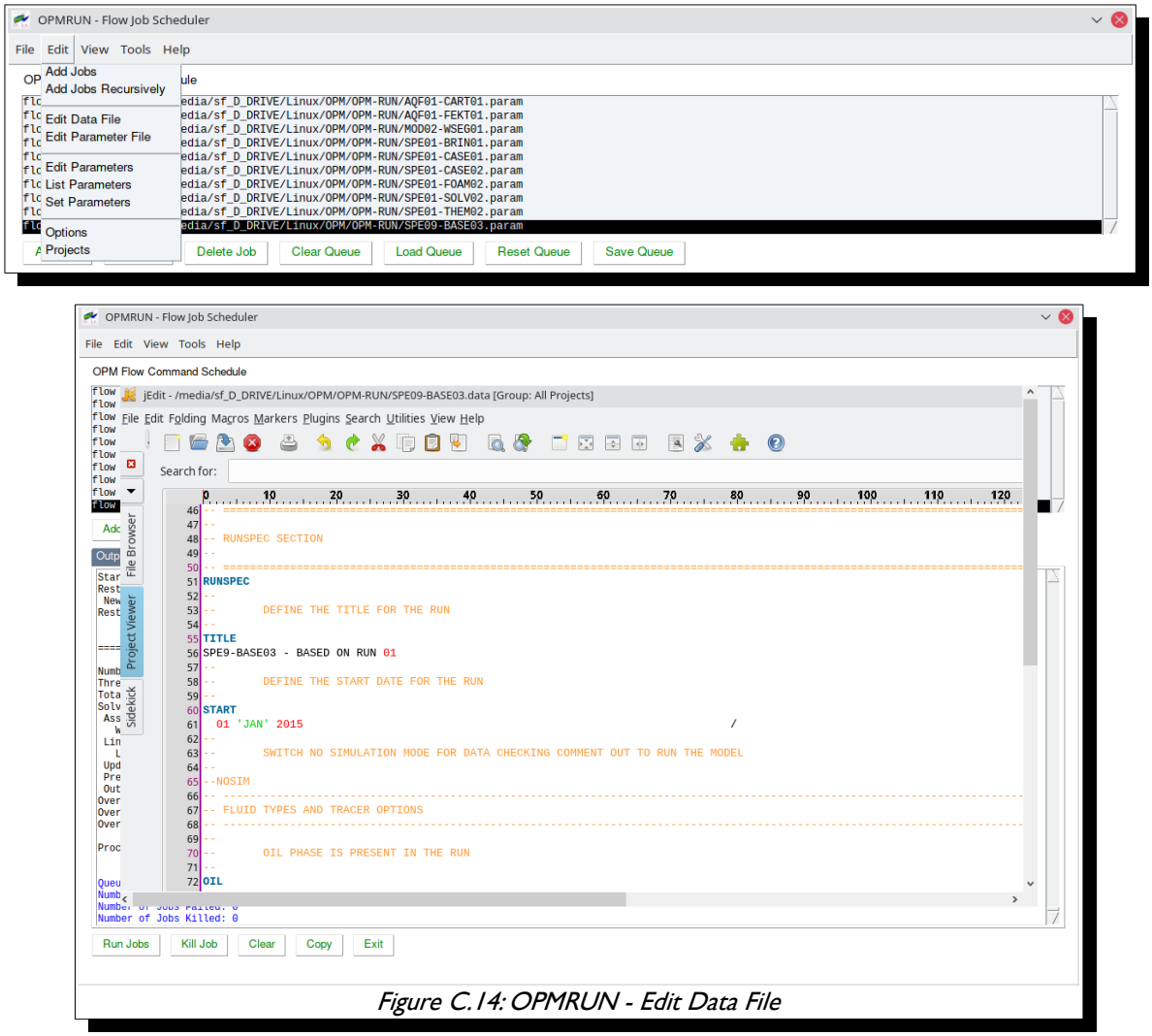


Figure C.14: OPMRUN - Edit Data File

The options are also available by right-clicking a job in the **Job List Element**

The Edit Parameters, List Parameters and Set Parameters relate to the default parameter set, not the parameter set for a particular job. When a new job is added to the queue the application checks if a *.PARAM file exists for the job, if not then the default parameter set is used for the job. Editing the default parameter set is the same as editing a job parameter set (see section C.2.2 Edit Job Data and Parameter File for more information).

One can also define the default parameter set by using the OPM Flow default values, which is the set created when OPMRUN is first initialized. In addition, one can load an existing parameter set from an existing *.PARAM file or from an OPM Flow *.PRT file (Figure C.15).

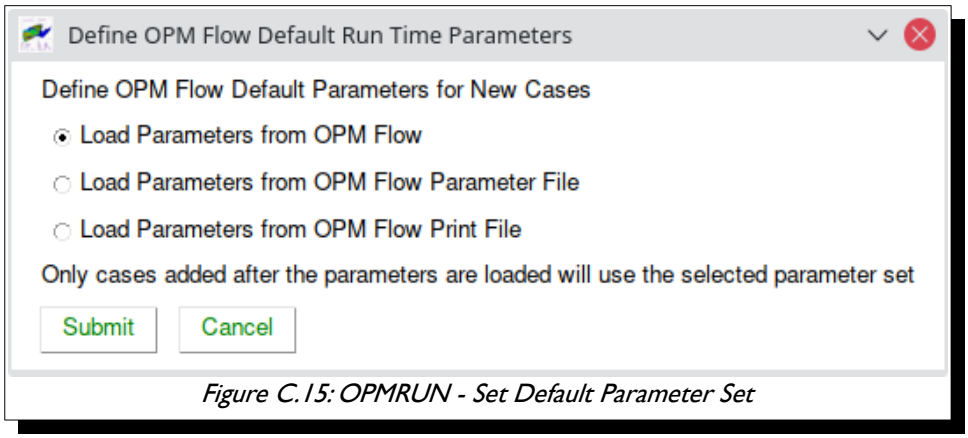


Figure C.15: OPMRUN - Set Default Parameter Set

OPMRUN has several configuration options that can be set via the Edit/Options menu option as illustrated in Figure C.16. These include setting:

- The location of the OPM Flow manual.
- The Keyword Generator Template Directory, one of tools supplied with OPMRUN, see section C.3.2 OPMRUN Tools: Simulator Input/Keywords for further information on this application.
- The location of the ResInsight program, for loading the results of a simulation run for viewing.
- The editor command to used to edit the input deck and view the resulting simulator output files.
- The terminal console to be be used for background jobs. WSL ("Windows Subsystem for Linux) should be selected if OPMRUN is running under Windows 10 to enable jobs to be submitted to the installed Linux distribution.
- The User Information series of fields are used by various supplied tools and in some templates used by the Keyword Generator application (see C.3.2 OPMRUN Tools: Simulator Input/Keywords for more information on this application). Note if a User Information" field is not defined then the template variable will be output instead – this can easily be deleted in the application.
- One can also define the main OPMRUN windows configuration parameters define: input (Job List Element) and output panel's (OPM Flow Output Element) size, font and font size.

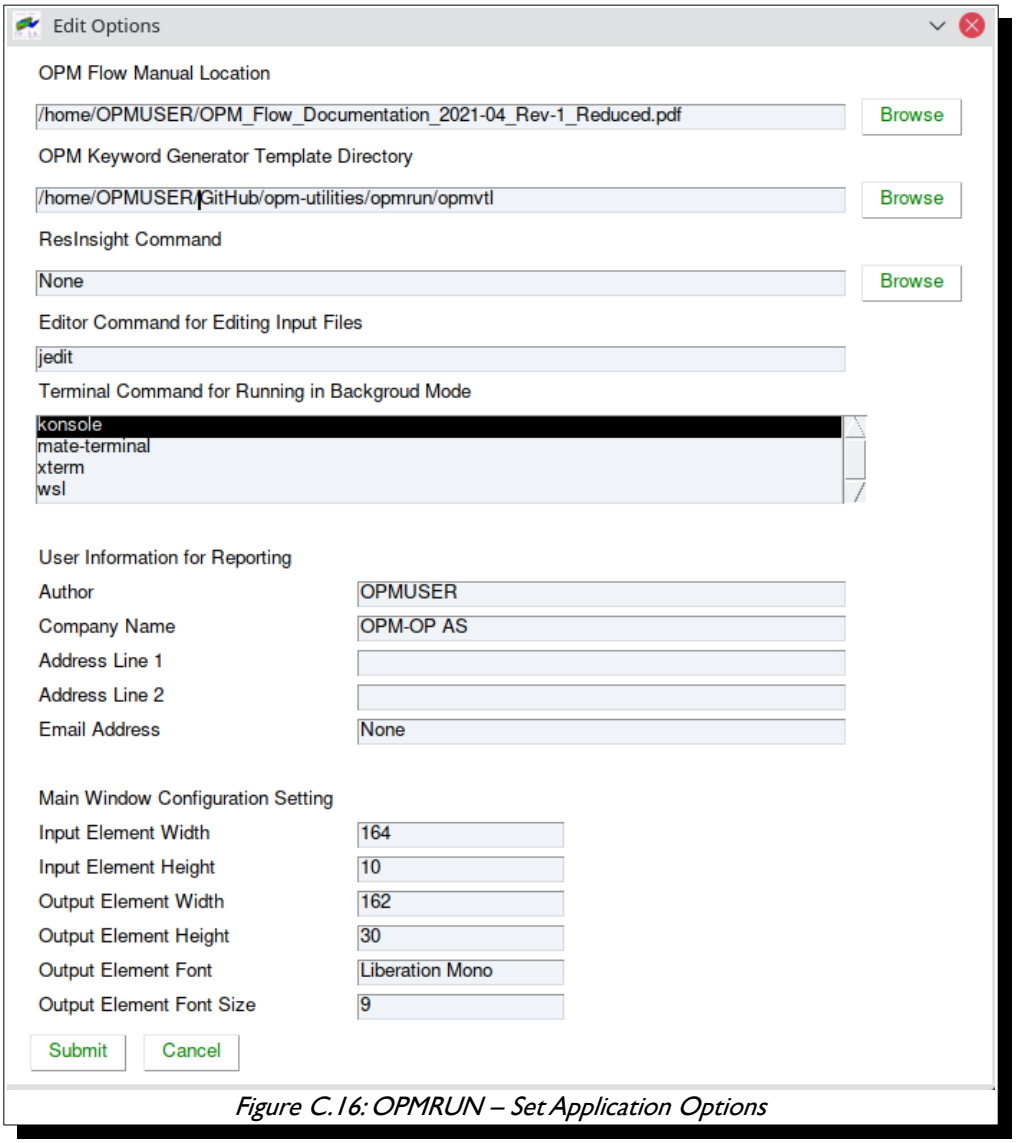


Figure C.16: OPMRUN – Set Application Options

In addition to the aforementioned options, the Edit/Projects menu item enables the setting of project directories that allows the user to set a default directory for loading and saving files within OPMRUN and the auxiliary applications (Figure C.17).

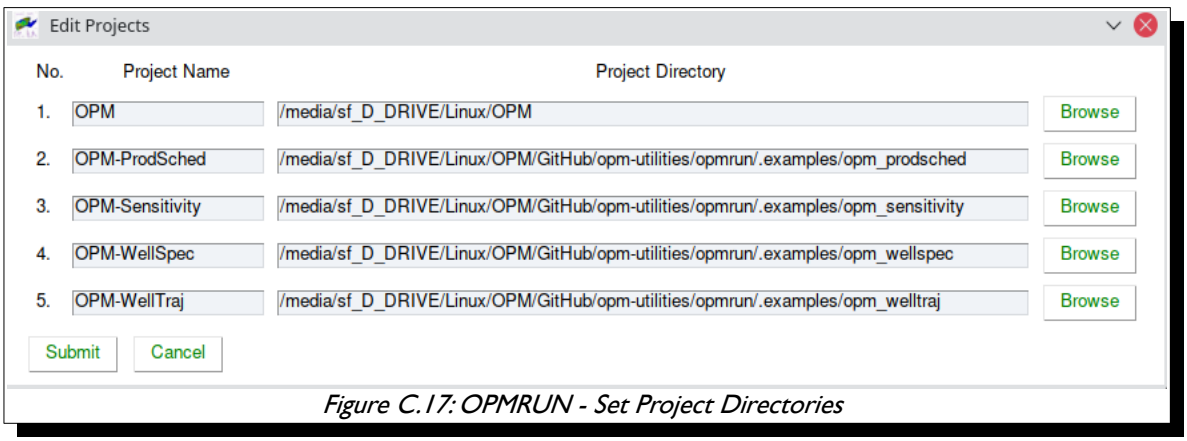
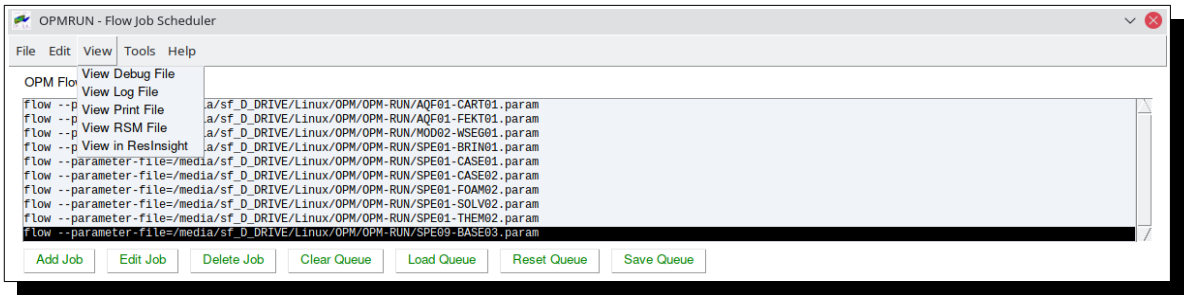


Figure C.17: OPMRUN - Set Project Directories

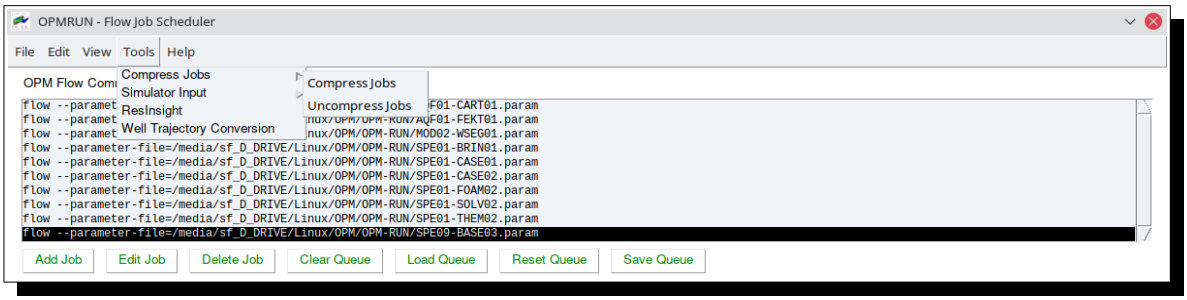
View Menu Options.

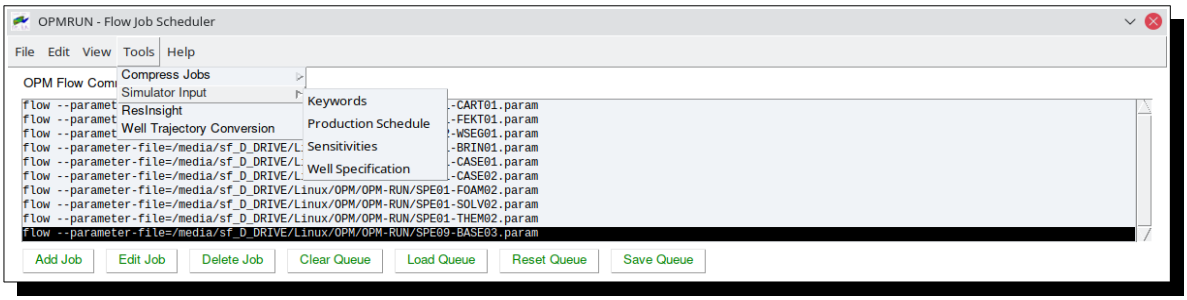
Allows the user to view the results of an OPM Flow simulation run using the default editor. The options are also available by right-clicking a job in the *Job List Element*.



Tools Menu Options

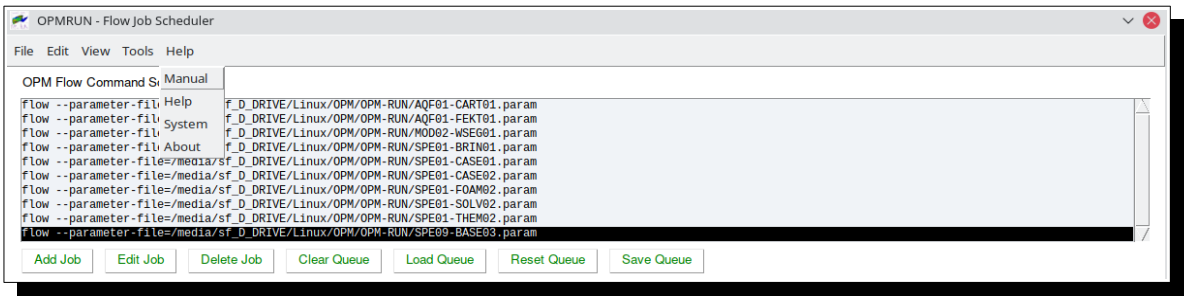
Contains various tool that may be useful in building a simulation model.





See the section C.3 OPMRUN Tools for further details on the available tools.

Help Menu Options



Use the Edit / Options menu option to select the location of the OPM Flow Manual.

C.3 OPMRUN TOOLS

C.3.1 OPMRUN TOOLS: JOB FILE COMPRESSION UTILITY FOR SAVING SPACE AND ARCHIVING.

Simulation input and output files can be extremely large, especially for large full field models. Running multiple cases in these circumstances can easily use up all available disk space, especially if multiple users are running multiple cases. The Tools/Compression Jobs option allows the user to compress a series of jobs into individual zip files (one zip file per job), as well as uncompressing previously zip job files.

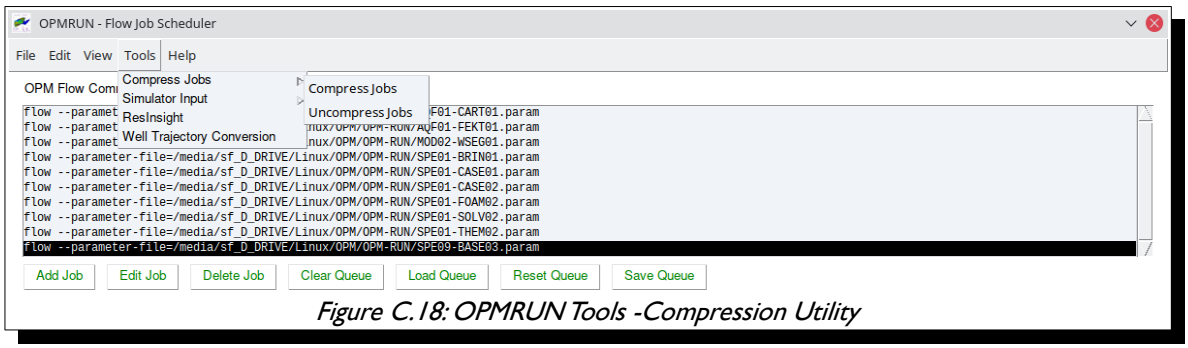


Figure C.18: OPMRUN Tools -Compression Utility

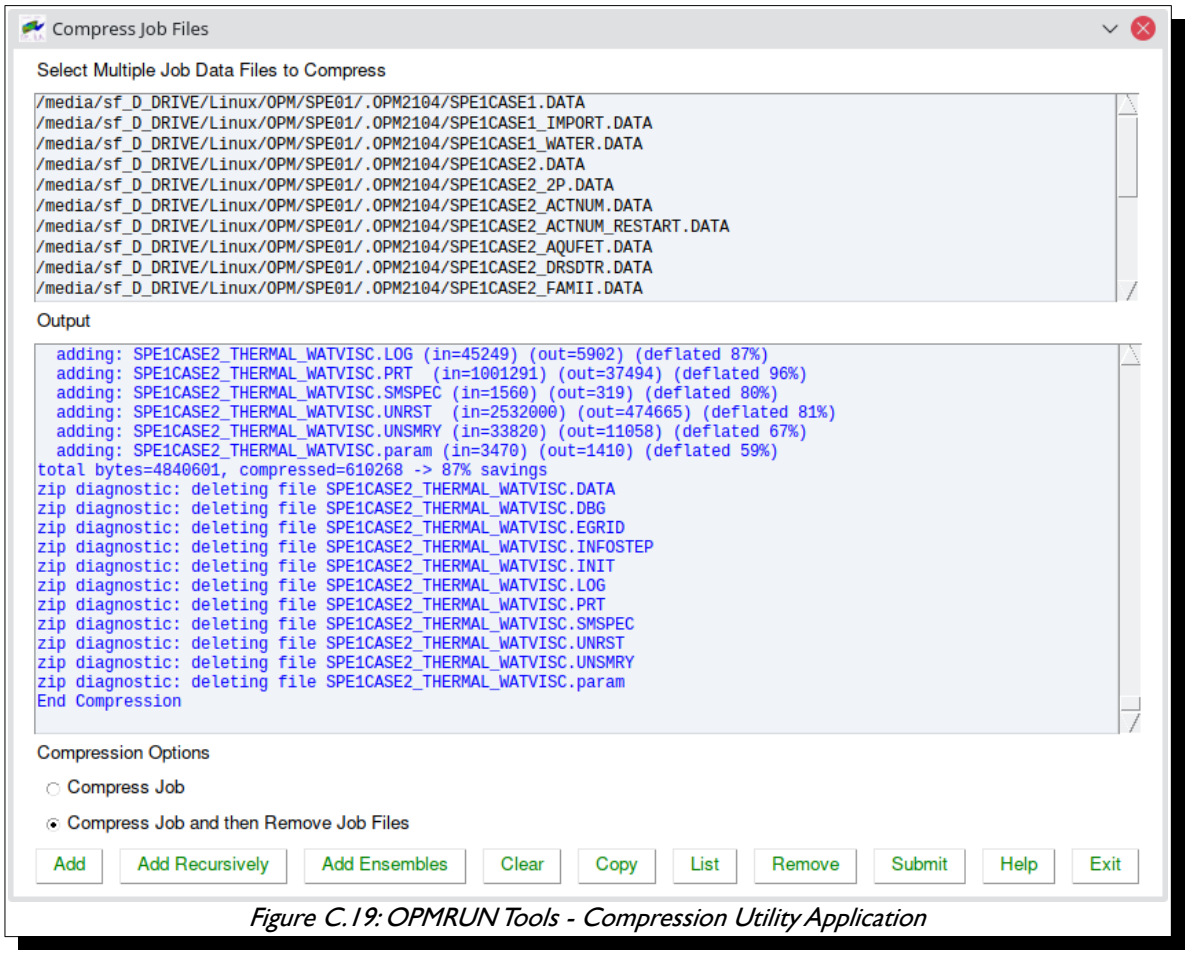


Figure C.19: OPMRUN Tools - Compression Utility Application

Note there is a similar application for uncompressing zip files and that the tool uses the Linux zip and unzip programs both on Linux host systems and Windows 10 systems using WSL.

C.3.2 OPMRUN TOOLS: SIMULATOR INPUT/KEYWORDS

The OPMRUN Keyword generator is an application that generates OPM Flow keywords that can be cut and pasted into any editor or saved to a separate file to form the basis of a new input deck (*.DATA files). The application utilizes templates based on the Apache Velocity Template Language ("VTL"), a commonly used template language used by software engineers. The templates can therefore also be used with any editor that supports VTL, jEdit for example, a popular open source Java based editor and PyCharm, a Python integrated development environment used in computer programming, specifically for the Python language.

There is one template per keyword, with formatting of the keywords being the same as the OPM Flow manual. Currently there are over 450 templates implemented and the intention is for additional keywords to be added as the their usage is implemented within the simulator and documented within the manual. The application allows one to customize the existing templates as well as creating user defined templates by including the templates in the template directory and following the VTL language syntax. One still has to edit the resulting keywords to match the data require, but the structure and comments are provided by the application.

The application is accessed via the Tool/Simulator Input/Keywords menu item and the tool can generate specific keywords, as well as complete sections (Figure C.20)

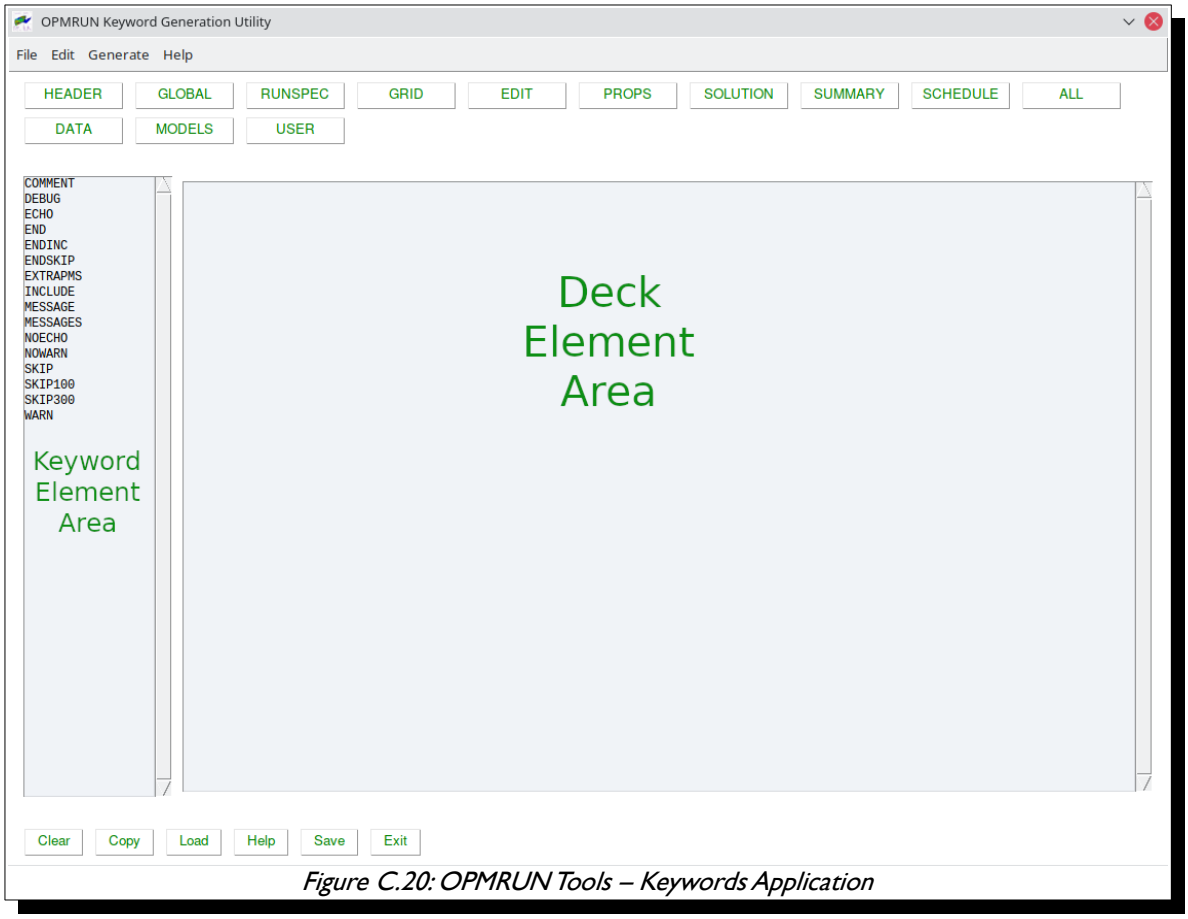


Figure C.20: OPMRUN Tools – Keywords Application

The application consists of several elements, a conventional menu system at the top, a *Deck Element Area* that will contain the resulting generated keywords, a *Keyword Element Area* for the user to select the keyword, data, models or user templates, and finally a series of buttons, *HEADER*, *GLOBAL*, etc., that are used to select the keywords in a OPM Flow section, specific data sets, models or user defined templates. The selection will appear in the *Keyword Element Area*.

Clicking on an item in the *Keyword Element Area* will generate the data for the item in the *Deck Element Area*, as shown below for the OPM Flow copyright header in Figure C.21.

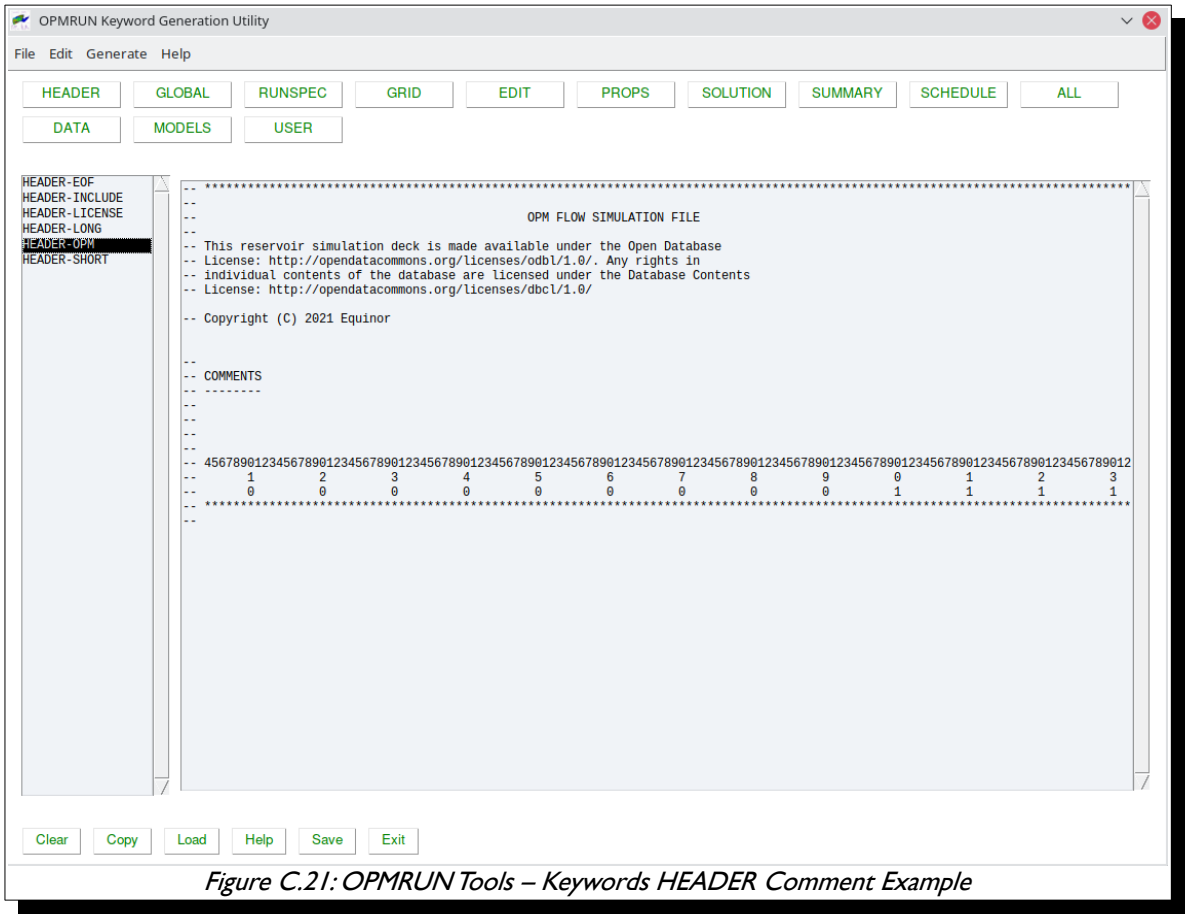


Figure C.21: OPMRUN Tools – Keywords HEADER Comment Example

The *Deck Element* Area is editable by simply clicking anywhere in the element and making changes. Use the *Clear* button to clear the *Deck Element Area* display, the *Copy* button to copy the *Deck Element Area* data to the clipboard, and the *Save* button to save the data to a file. The *Load* button allows one to load an existing file into the *Deck Element Area* for additional editing.

Note that the *HEADER* section is not an OPM Flow section, but a list of various comment block headers used to make the deck more readable.

Keywords: Menu Options

The various menu options include the File Menu

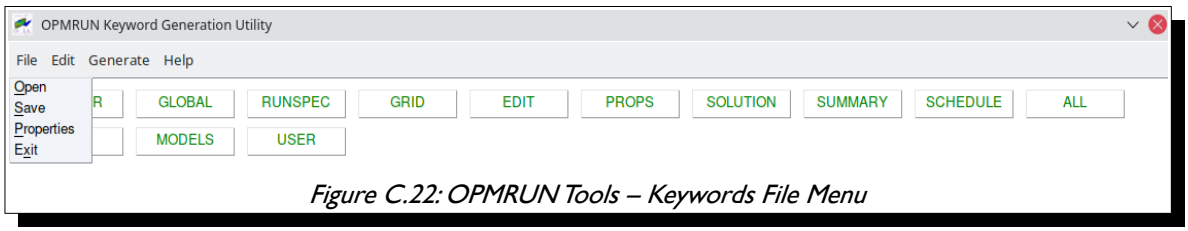


Figure C.22: OPMRUN Tools – Keywords File Menu

Where the *Open* and *Save* options load and save a file, and the *Properties* displays OPMRUN's properties.

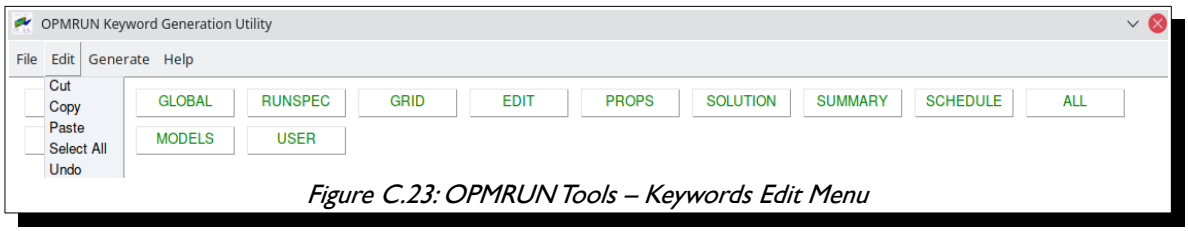


Figure C.23: OPMRUN Tools – Keywords Edit Menu

The Edit Menu provides some basic standard editing facilities

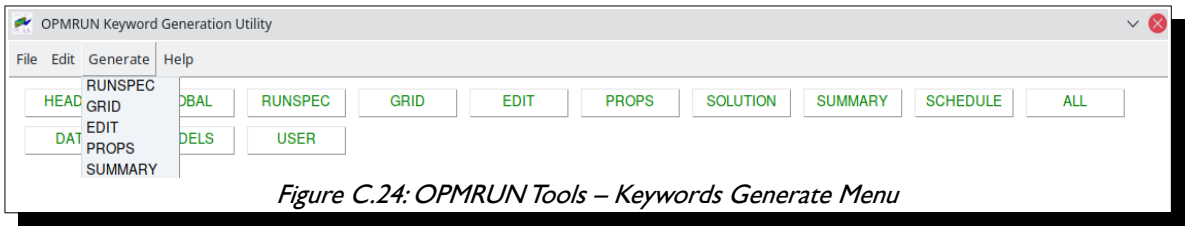


Figure C.24: OPMRUN Tools – Keywords Generate Menu

Next, the Generate Menu options allows one to generate a complete section of keywords, as described below. These options are equivalent to selecting the equivalent section keyword (RUNSPEC, GRID, etc.) in the *Keyword Element Area*.

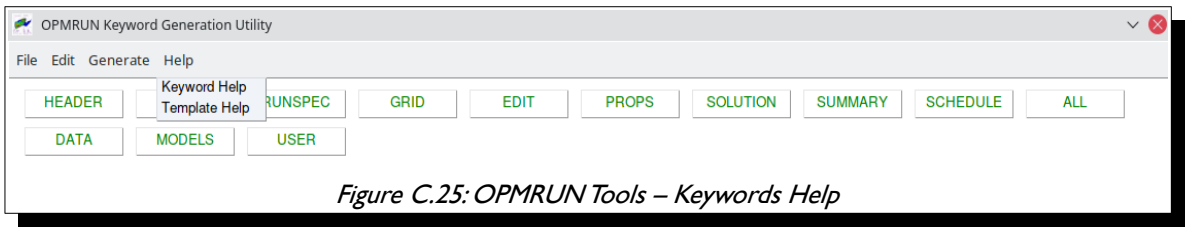


Figure C.25: OPMRUN Tools – Keywords Help

Finally, the Help Menu option display the Keyword Help information (Figure C.26):

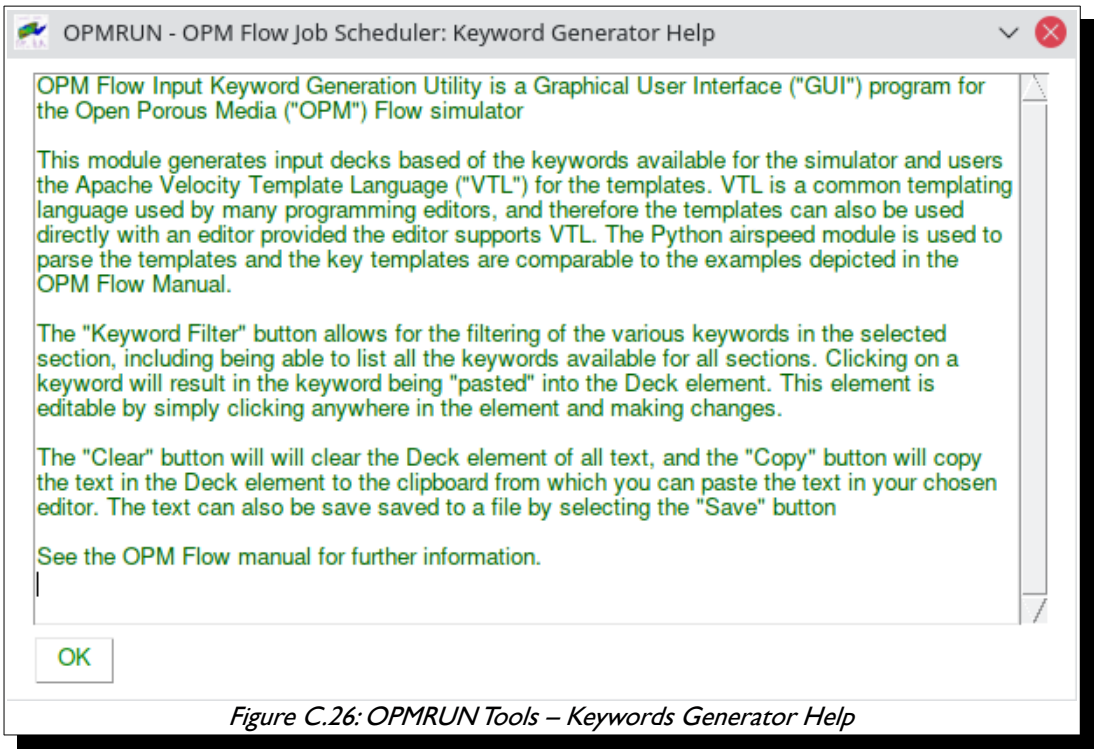


Figure C.26: OPMRUN Tools – Keywords Generator Help

And the Velocity Template Help (Figure C.27).

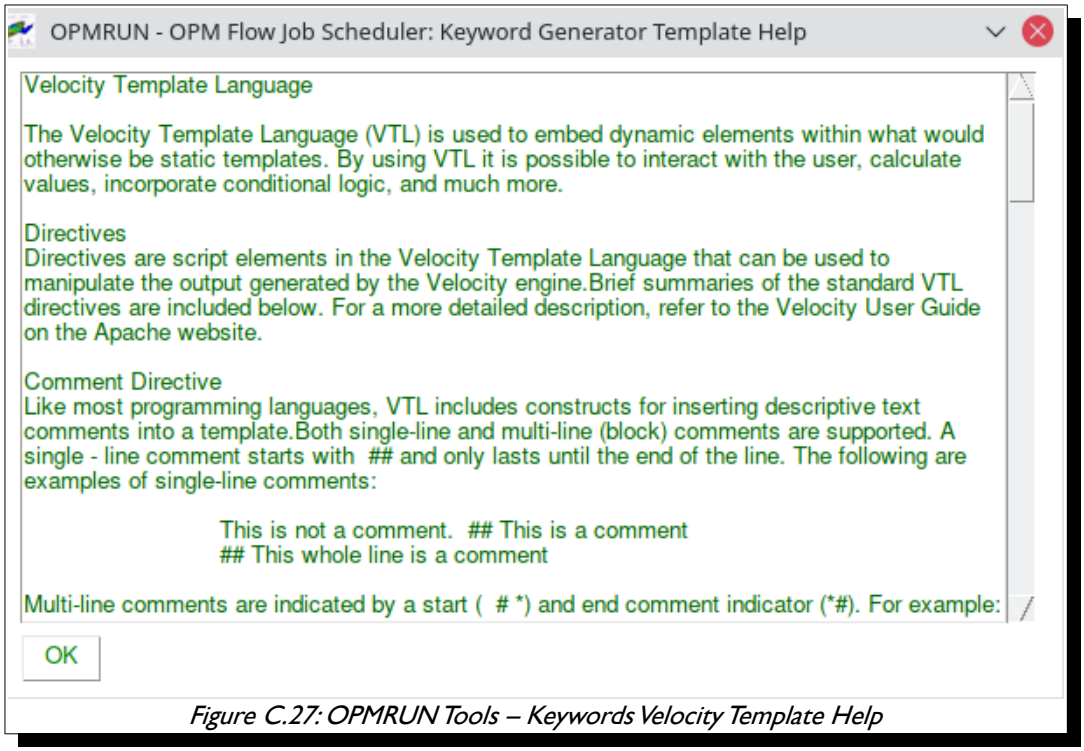


Figure C.27: OPMRUN Tools – Keywords Velocity Template Help

As mentioned previously, the tool uses the Apache Velocity Template Language ("VTL") for the templates, and therefore the templates can also be used directly with an editor, provided the editor supports VTL.

Keywords: File Imports

If a keyword requires a file, for example, the INCLUDE and LOAD keywords, then a dialog box is presented to enable the file to be selected. The application will also allow one to select the file name format, after the file has been selected.

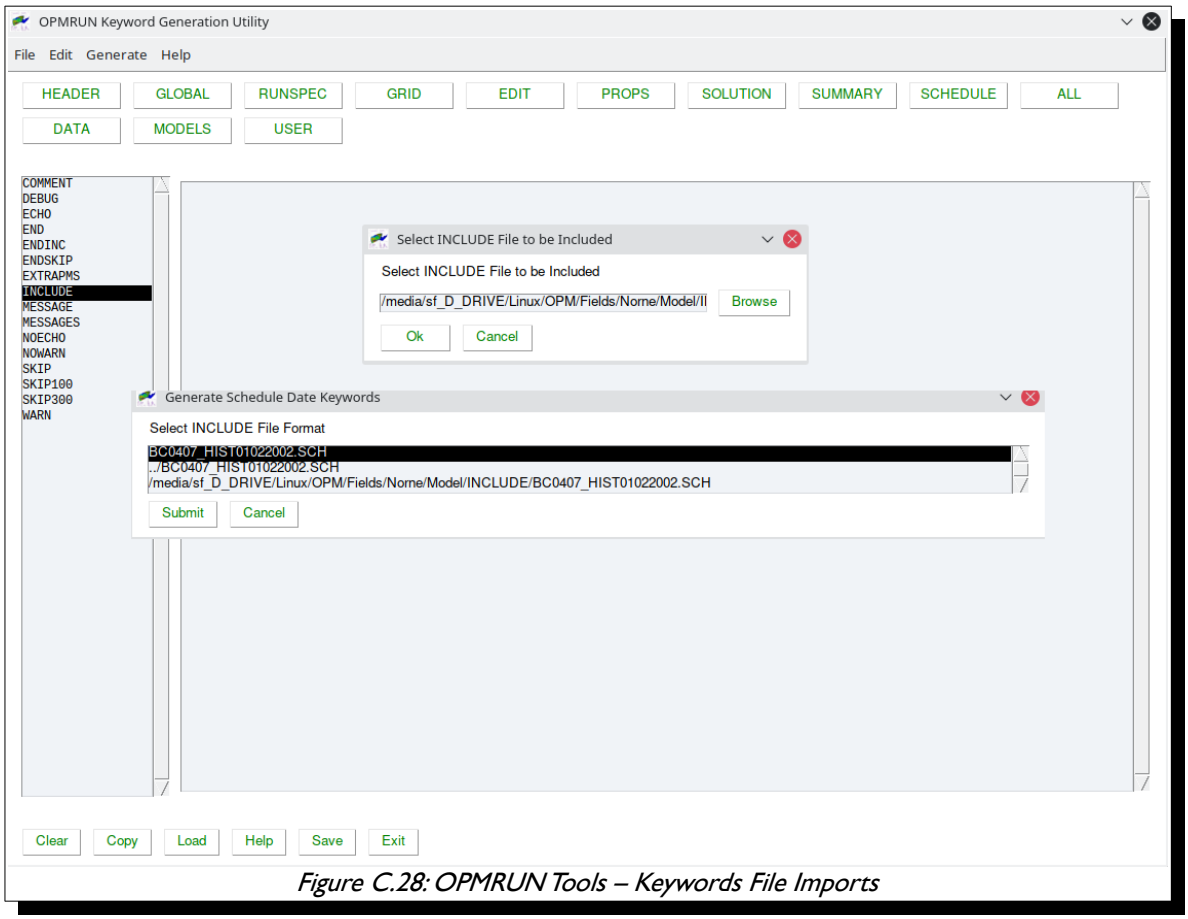


Figure C.28: OPMRUN Tools – Keywords File Imports

Note that COMMENT template is not an actual keyword, but a comment block to make the deck more readable for the user.

Keywords: Section Standard Set of Keywords

Selecting a Generate Menu option or a Section keyword (RUNSPEC, GRID, EDIT, PROPS, SOLUTION, SUMMARY, and SCHEDULE) in the *Keyword Element Area* will give an option to generate a representative set of keywords for that section, as per the RUNSPEC example in Figure C.29.

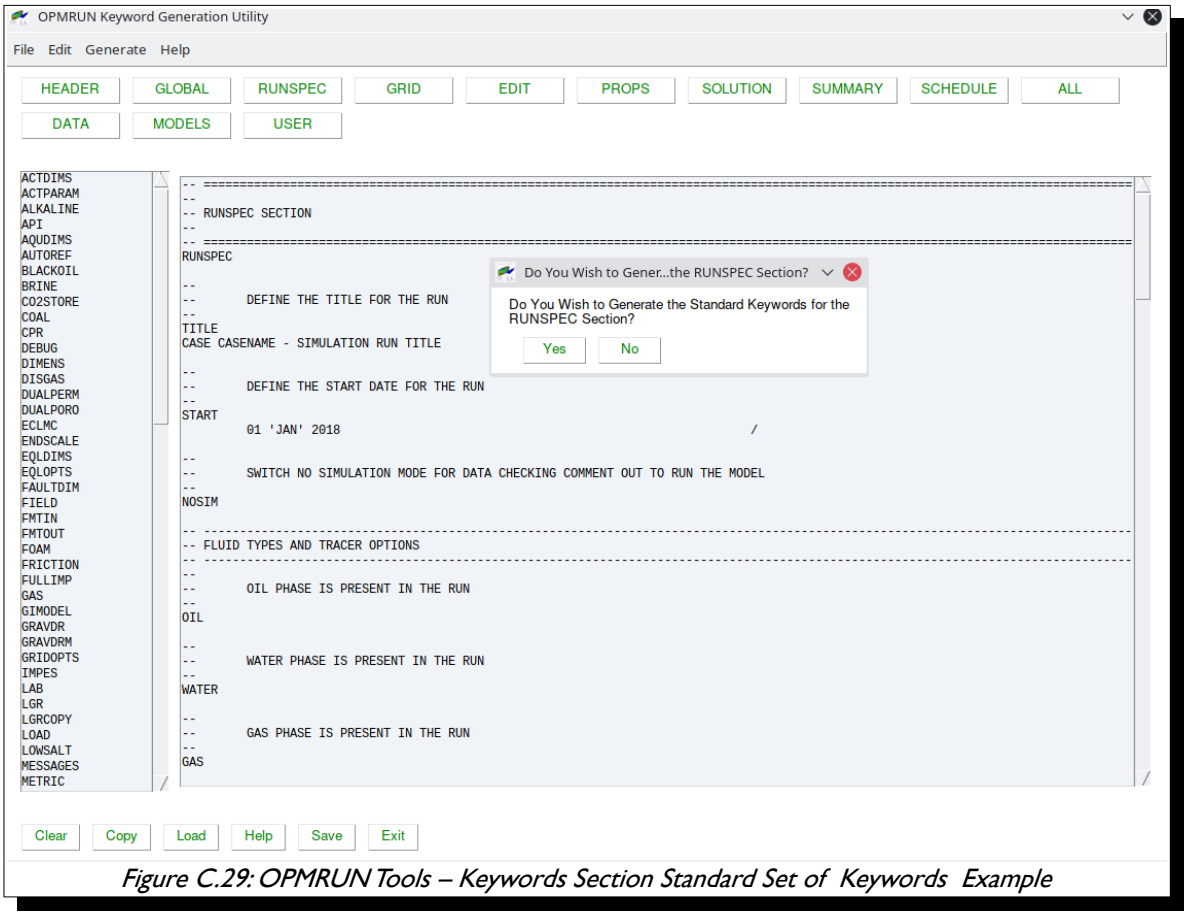


Figure C.29: OPMRUN Tools – Keywords Section Standard Set of Keywords Example

One can therefore generate a complete input deck in a matter of minutes; however, you still have to edit the generated input deck with your actual data.

Keywords: SUMMARY Section Variables

For the SUMMARY section keyword, one can also generate various sets of summary variables based on the options being used in the model. Note that not all the variables are currently available in OPM Flow, but additional variables are added at each release.

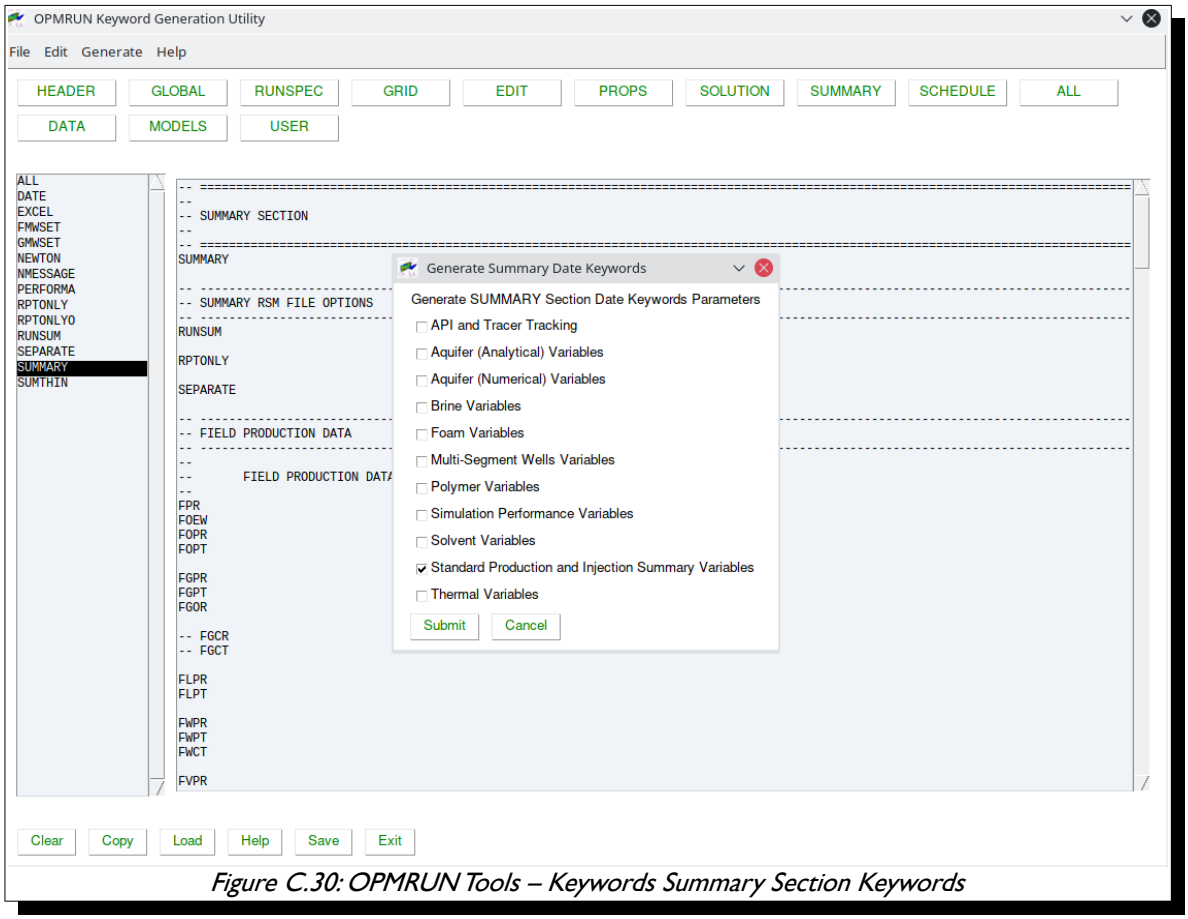


Figure C.30: OPMRUN Tools – Keywords Summary Section Keywords

For SUMMARY variables not recognized by OPM Flow, the simulator will issue a warning message and ignore those variables not implemented.

Keywords: SCHEDULE Section Keywords and Date Schedule

For the SCHEDULE Section keyword, one can also generate a representative set of SCHEDULE keywords, plus a date schedule from a start year to an end year, using Annual, Quarterly, or Monthly time steps.

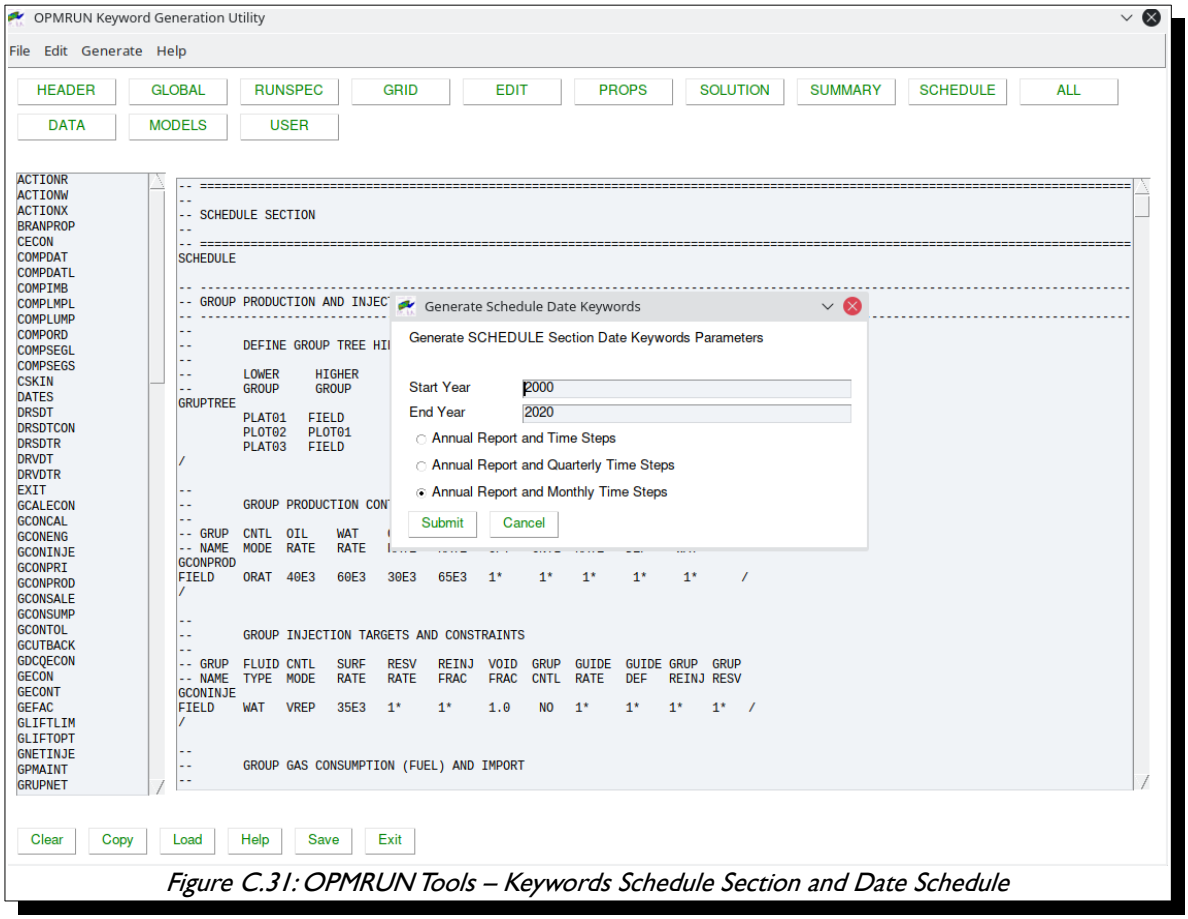


Figure C.31: OPMRUN Tools – Keywords Schedule Section and Date Schedule

This option also writes a standard report using the RPTSCHED keyword at the beginning of each year which is subsequently switch off for the intermediate Quarterly and Monthly time steps. A final report is written at the end of the run.

Keywords: DATA (Sets) Option

There is also a *DATA* option which is not an OPM Flow section, but a series of data sets, as shown in Figure C.32.

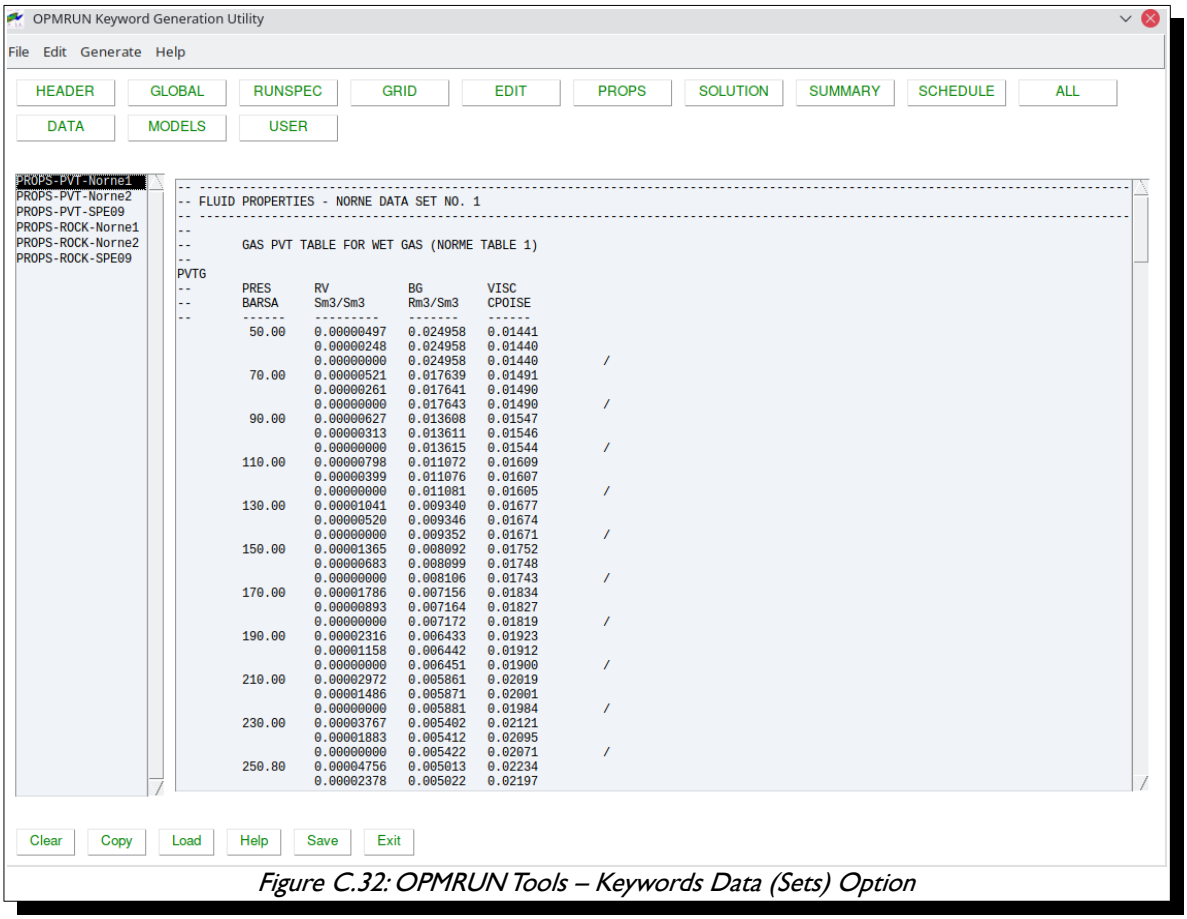


Figure C.32: OPMRUN Tools – Keywords Data (Sets) Option

The data sets are complete examples for a given type of data used in OPM Flow, for example a PVT data set for a wet gas reservoir, or three phase relative permeability data set. The data sets are intended to be used as a guide for generating ones own keyword input, or for building models for testing.

The intention is to expand the collection of data sets over time as more data becomes available.

Keywords: MODEL Option

Like the DATA option, the MODEL option is not an OPM Flow section, but is instead a collection of working models, as illustrated in Figure C.33.

The purpose of the models is to illustrate the functionality of various features implemented in OPM Flow and to act as guide for users in building their own models.

Figure C.33: OPMRUN Tools – Keywords Model Option

Additional models will be added when available.

Keywords: USER Templates Option

Finally, the *USER* option is where users can store their own templates. *USER* templates with the "vm" extension will automatically be listed by the *USER* button. To use this feature, after selecting a keyword, right clicking on the keyword allows one to load the actual template for the keyword. One can then edit the template and save the changes back to the same template or another template using the *Save* button.

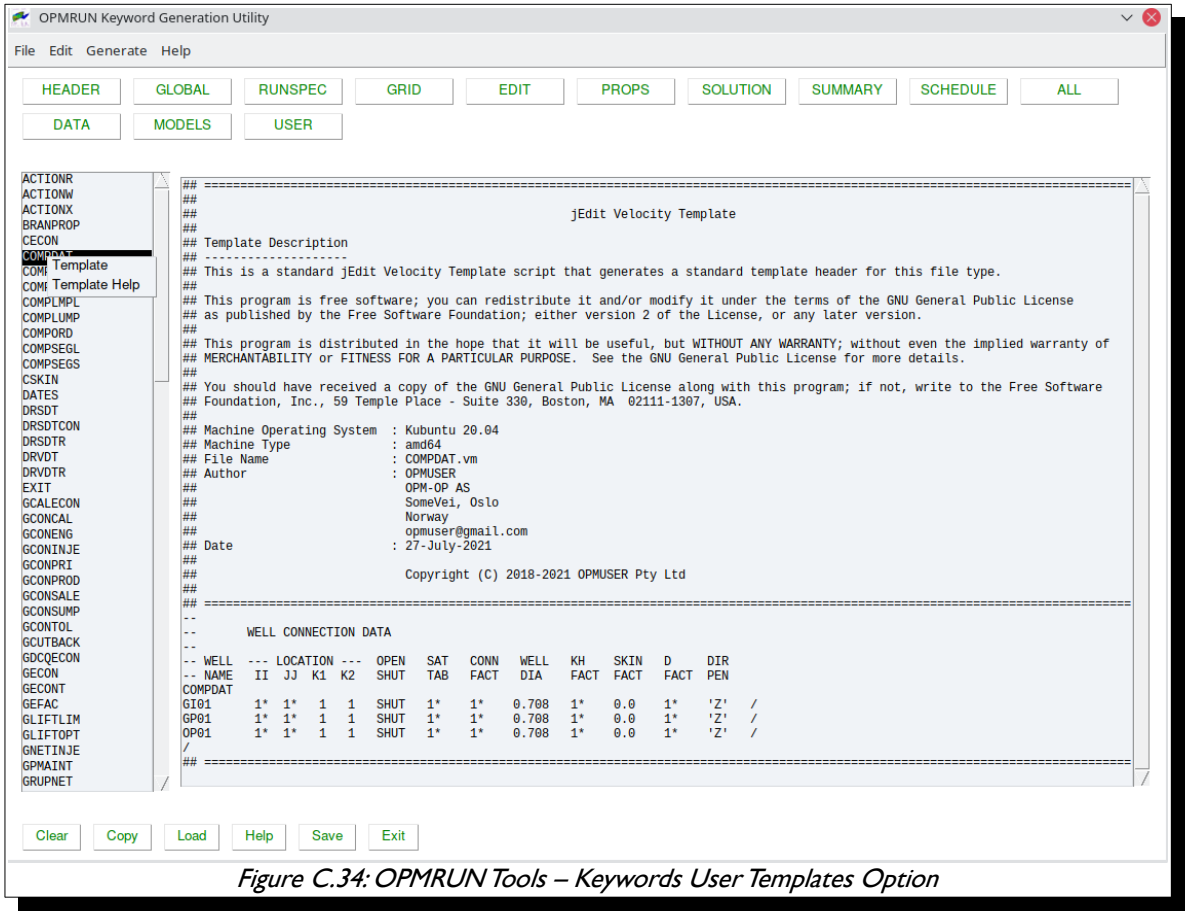


Figure C.34: OPMRUN Tools – Keywords User Templates Option

The Template Help option displays a brief introduction to VTL for further reference.

C.3.3 OPMRUN TOOLS: SIMULATOR INPUT/PRODUCTION SCHEDULE

The *Tools/Simulator Input/Production Schedule* application takes a comma delimited CSV file containing historical production and injection data and converts the data to an OPM Flow SCHEDULE file using the WCONHIST series of keywords. An example input file is shown below:

	A	B	C	D	E	F	G	H
1	Xy.Wellcompl	Date	Prd_dly.Time	Prd_dly.Cond	Prd_dly.Water	Prd_dly.Gas	Prd_dly.whp	Prd_dly.wht
2	GP-P1-01	03/01/2020	24	25	16	10528	551	133
3	GP-P1-01	03/02/2020	24	25	16	10507	551	133
4	GP-P1-01	03/03/2020	24	25	16	10513	550	131
5	GP-P1-01	03/04/2020	24	25	16	10506	549	132
6	GP-P1-01	03/05/2020	24	25	16	10503	550	133
7	GP-P1-01	03/06/2020	24	25	16	10506	549	132
8	GP-P1-01	03/07/2020	24	25	16	10416	556	131
9	GP-P1-01	03/08/2020	24	24	15	10054	595	132
10	GP-P1-01	03/09/2020	24	25	15	10180	584	131
11	GP-P1-01	03/10/2020	24	24	16	10493	549	132
12	GP-P1-01	03/11/2020	24	25	15	10493	549	133
13	GP-P1-01	03/12/2020	24	25	16	10476	550	133
14	GP-P1-01	03/13/2020	24	25	16	10243	607	132
15	GP-P1-01	03/14/2020	24	25	15	10276	596	130
16	GP-P1-01	03/15/2020	24	24	16	10414	548	131
17	GP-P1-01	03/16/2020	24	25	15	10400	549	132
18	GP-P1-01	03/17/2020	24	26	17	10471	601	130
19	GP-P1-01	03/18/2020	24	29	15	10819	647	131
20	GP-P1-01	03/19/2020	24	26	16	10348	653	132
21	GP-P1-01	03/20/2020	24	27	17	10843	626	132
22	GP-P1-01	03/21/2020	24	28	16	10929	649	131
23	GP-P1-01	03/22/2020	24	26	16	10901	648	131
24	GP-P1-01	03/23/2020	24	24	17	10693	575	133

Figure C.35: OPMRUN Tools – Production Schedule Input File Example

The first row in the input file is a header row that declares the data type for a column, the example shows typical Oil Field Manager ("OFM") header variable names, but various variable names can be used to define the data type.

The tool can convert daily production data to a: daily production schedule, monthly average, or monthly on-stream average production schedule, as shown below:

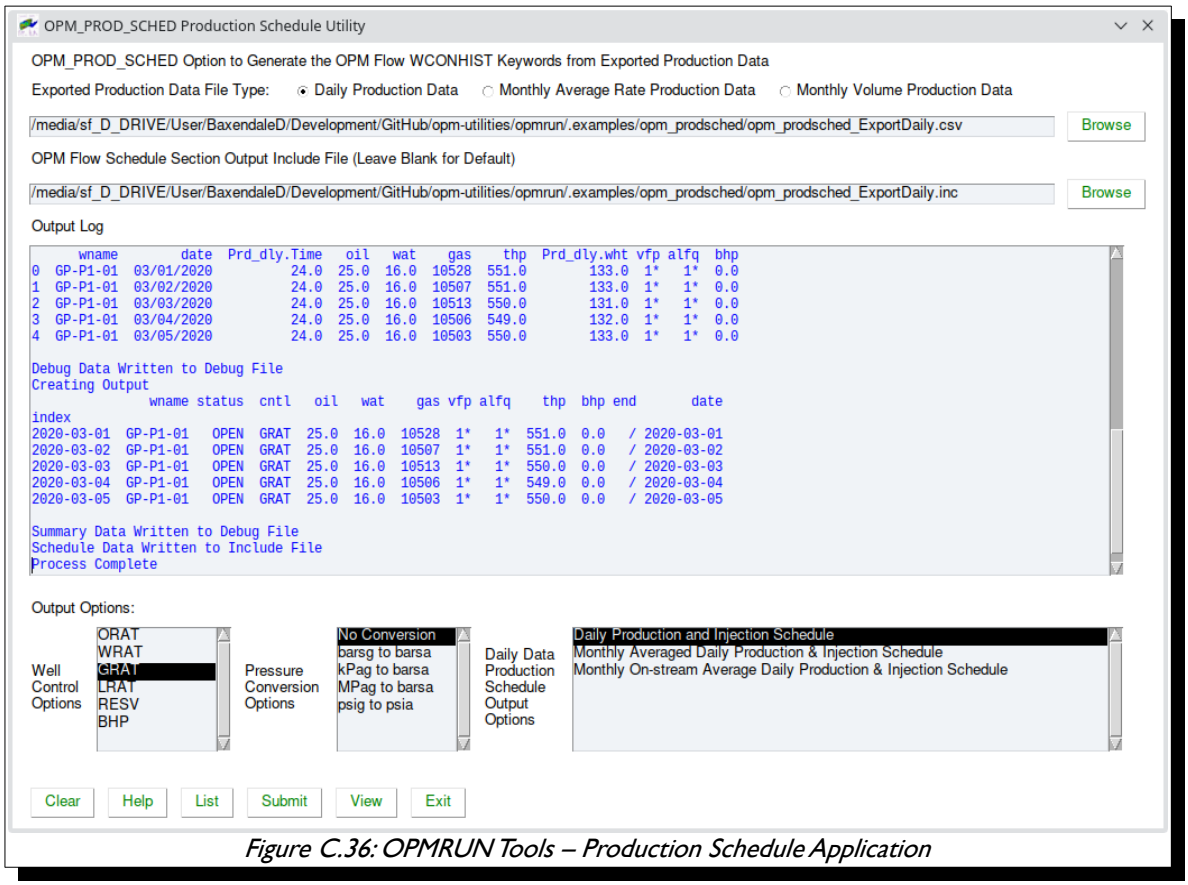


Figure C.36: OPMRUN Tools – Production Schedule Application

Notice that the application checks various variable names for the column headers. For example for the BHP data, the column names can be: bhp, bottom-hole pressure, BHP, or BOTTOM-HOLE PRESSURE.

A sample of the generated output file is shown in Figure C.37.

```
NOECHO
--
DATES
01 Mar 2020  /
/
--
-- WELL HISTORICAL PRODUCTION CONTROLS
--
-- WELL  OPEN CNTL    OIL   WAT   GAS   VFP  VFP     THP  BHP
-- NAME  SHUT MODE   RATE  RATE  RATE TABLE ALFQ    PRES PRES
WCONHIST
GP-P1-01 OPEN GRAT   25.0  16.0 10528    1*   1*   551.0  0.0 / 2020-03-01
GP-P1-02 OPEN GRAT   21.0  21.0 13680    1*   1*   557.0  0.0 / 2020-03-01
GP-P1-03 OPEN GRAT   39.0  34.0 16432    1*   1*   625.0  0.0 / 2020-03-01
GP-P1-05 OPEN GRAT  187.0  27.0 51043    1*   1*  1907.0  0.0 / 2020-03-01
GP-P1-06 OPEN GRAT   41.0  17.0  9427    1*   1*   619.0  0.0 / 2020-03-01
GP-P2-02 OPEN GRAT    0.0   0.0 13365    1*   1*   690.0  0.0 / 2020-03-01
GP-P2-03 OPEN GRAT    0.0   0.0 10486    1*   1*   663.0  0.0 / 2020-03-01
GP-P2-04 OPEN GRAT    0.0   0.0  8009    1*   1*   646.0  0.0 / 2020-03-01
/
DATES
02 Mar 2020  /
/
--
-- WELL HISTORICAL PRODUCTION CONTROLS
--
-- WELL  OPEN CNTL    OIL   WAT   GAS   VFP  VFP     THP  BHP
-- NAME  SHUT MODE   RATE  RATE  RATE TABLE ALFQ    PRES PRES
WCONHIST
GP-P1-01 OPEN GRAT   25.0  16.0 10507    1*   1*   551.0  0.0 / 2020-03-02
GP-P1-02 OPEN GRAT   21.0  21.0 13653    1*   1*   556.0  0.0 / 2020-03-02
GP-P1-03 OPEN GRAT   39.0  35.0 16400    1*   1*   624.0  0.0 / 2020-03-02
GP-P1-05 OPEN GRAT  186.0  25.0 51054    1*   1*  1905.0  0.0 / 2020-03-02
GP-P1-06 OPEN GRAT   41.0  17.0  9409    1*   1*   618.0  0.0 / 2020-03-02
GP-P2-02 OPEN GRAT    0.0   0.0 13485    1*   1*   681.0  0.0 / 2020-03-02
GP-P2-03 OPEN GRAT    0.0   0.0 10494    1*   1*   659.0  0.0 / 2020-03-02
GP-P2-04 OPEN GRAT    0.0   0.0  8024    1*   1*   641.0  0.0 / 2020-03-02
/
DATES
03 Mar 2020  /
/
```

Figure C.37: OPMRUN Tools – Production Schedule Output Example

Note

Note the current release only support production data via the WCONHIST keyword, injection data via WCONINJH keyword is not supported.

C.3.4 OPMRUN TOOLS: SIMULATOR INPUT/SENSITIVITIES

The *Tools/Simulator Input/Sensitivities* option generates sensitivity cases based on a "Base" case file. The Base file contains "Factors" (variable names), $X01, $X02, etc., that are substituted with user defined values using the data entered and the type of Sensitivity Scenario selected. Thus, the first step is to configure the Base file in a text editor by replacing actual values by the variable names, previously mentioned.

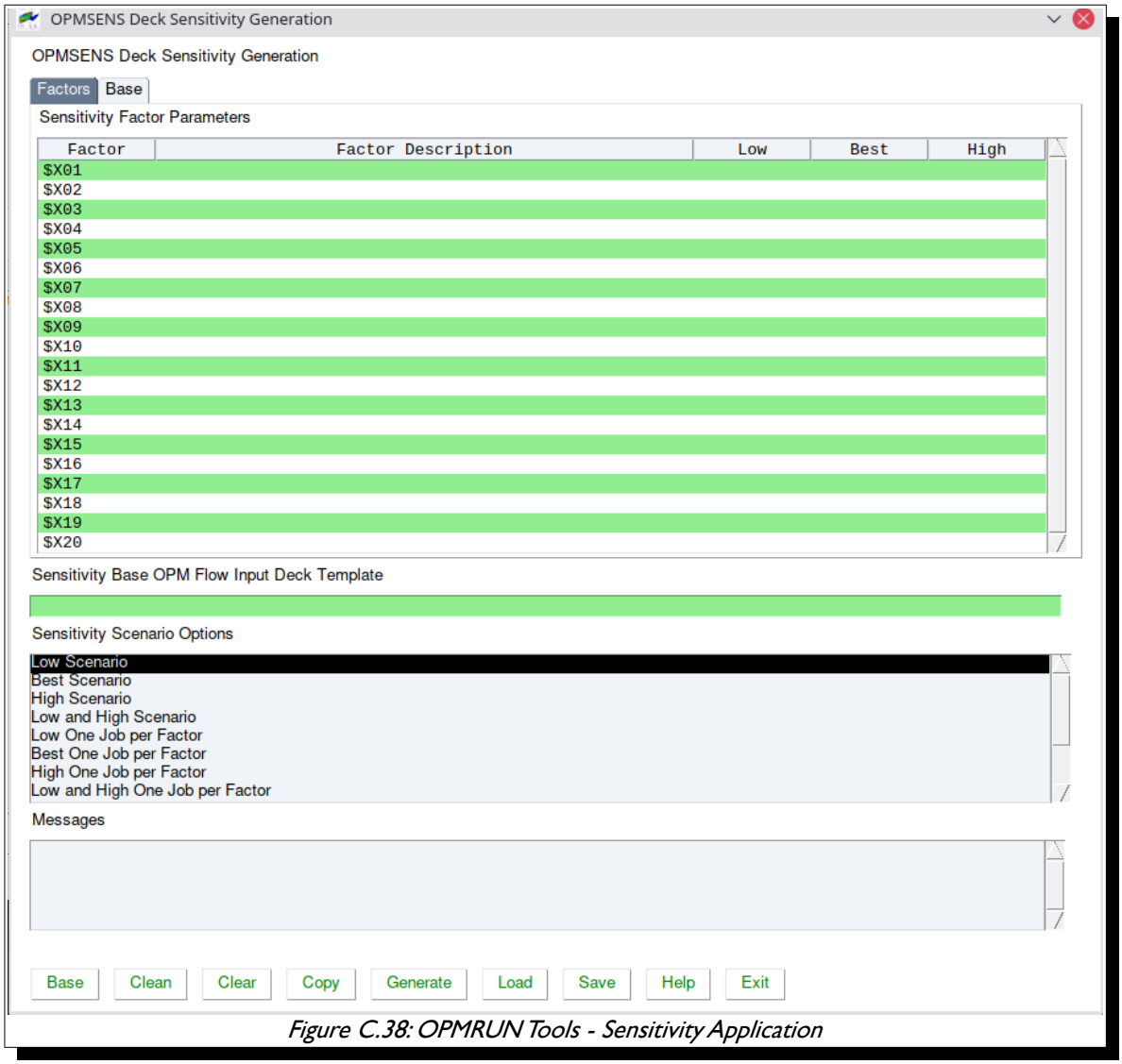


Figure C.38: OPMRUN Tools - Sensitivity Application

After editing the Base file, the next step is to load the Base file into the application using the *Base* button, which will prompt the user for the file to load and then display the file in the *Base* tab, as shown in Figure C.39

Figure C.39: OPMRUN Tools – Sensitivity Base Case Data File Example

Limited editing of the *Base* file is supported on the above screen.

The next step is to define the "Factors" and the factor values. A total of 20 factors are available and each factor consist of a Low, Best and High estimates. Note it is not necessary to enter all three estimates, if one wishes just to generate a limited sensitivity case. For example, if on wishes to only run a Low Scenario sensitivity then it is only necessary to enter data for the Low factor values.

Previously saved factor data can be loaded via the *Load* button, as shown below:

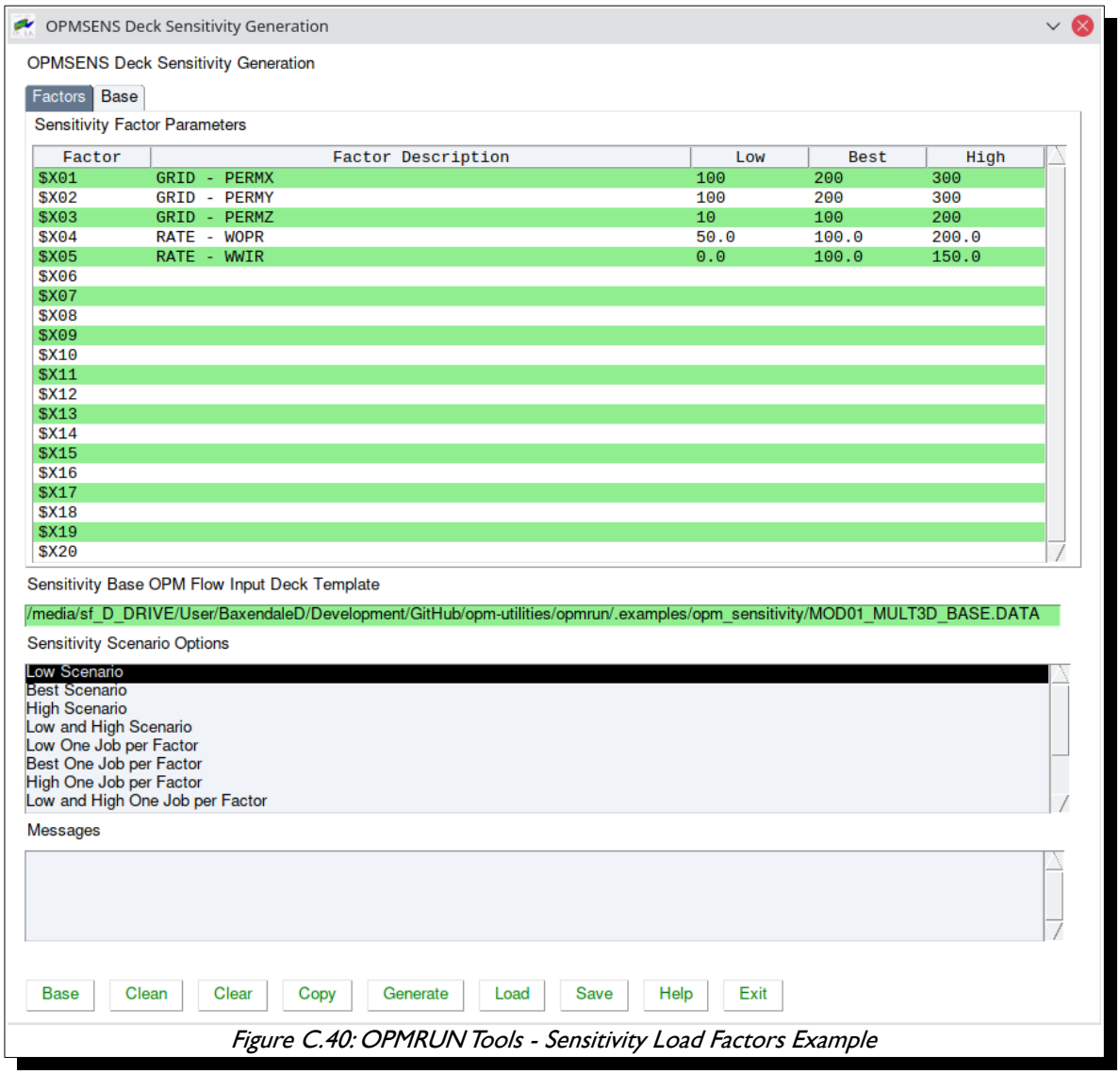


Figure C.40: OPMRUN Tools - Sensitivity Load Factors Example

Selecting a Factor Description row allows one to define a description for the factor variable, so for $X01 in the above figure the description is GRID - PERMX. When selecting a Factor Description, a popup dialog will be displayed to enter the data, and if one right-clicks on the popup's Factor Description field one can select a description for one of the pre-defined descriptions as illustrated in the next figure.

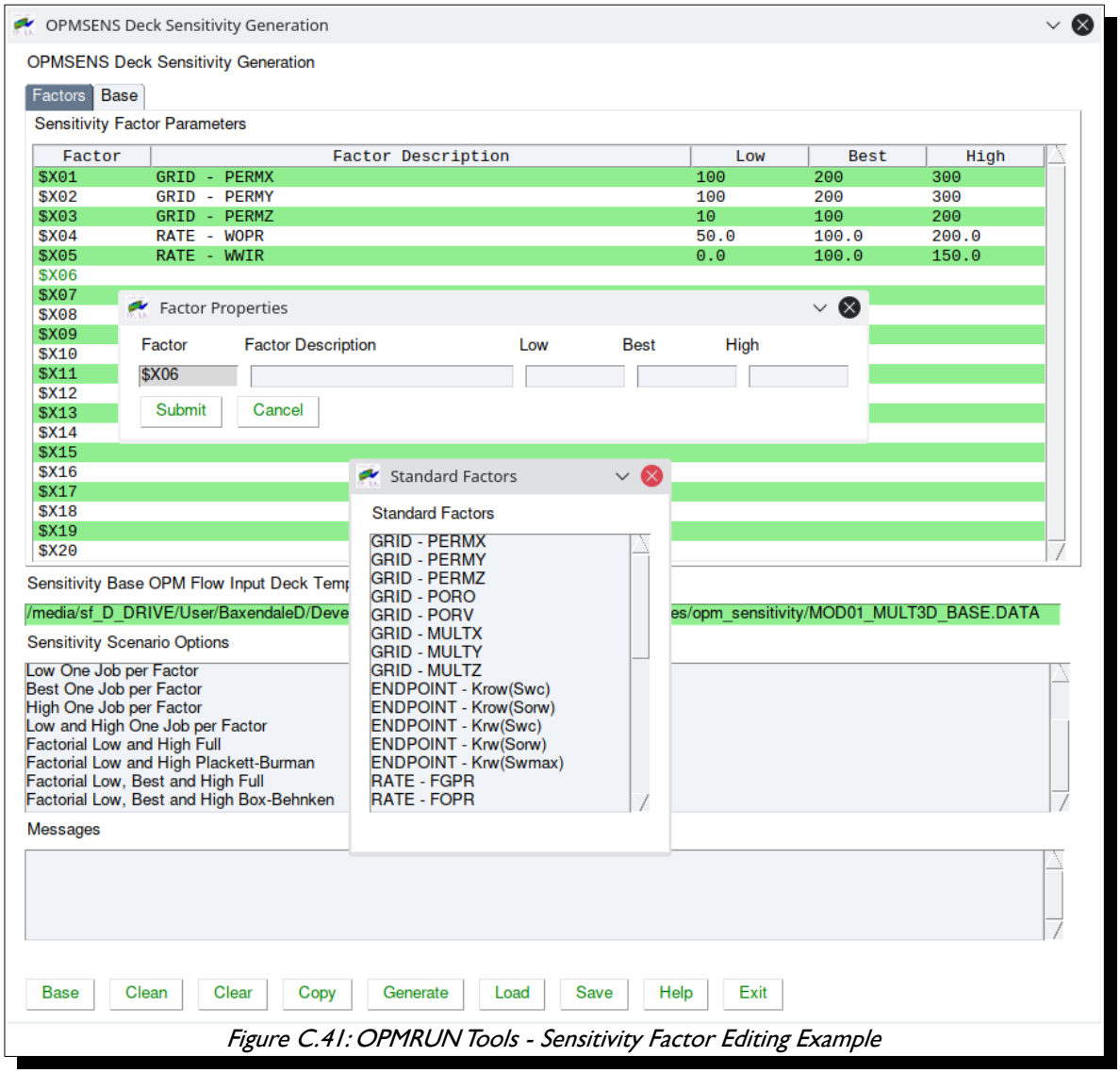


Figure C.41: OPMRUN Tools - Sensitivity Factor Editing Example

After the Sensitivity Factors have been entered one can then select the Sensitivity Scenario that one wishes to use generate the sensitivity cases. In the example in figure Figure C.42 the *Factorial: Low, Best and High Box-Behnken* DOE (Design of Experiments) has been selected. Selecting the *Generate* button, runs a series of checks, and if there are no errors the program will inquire if you wish to generate the set of cases (Figure C.42).

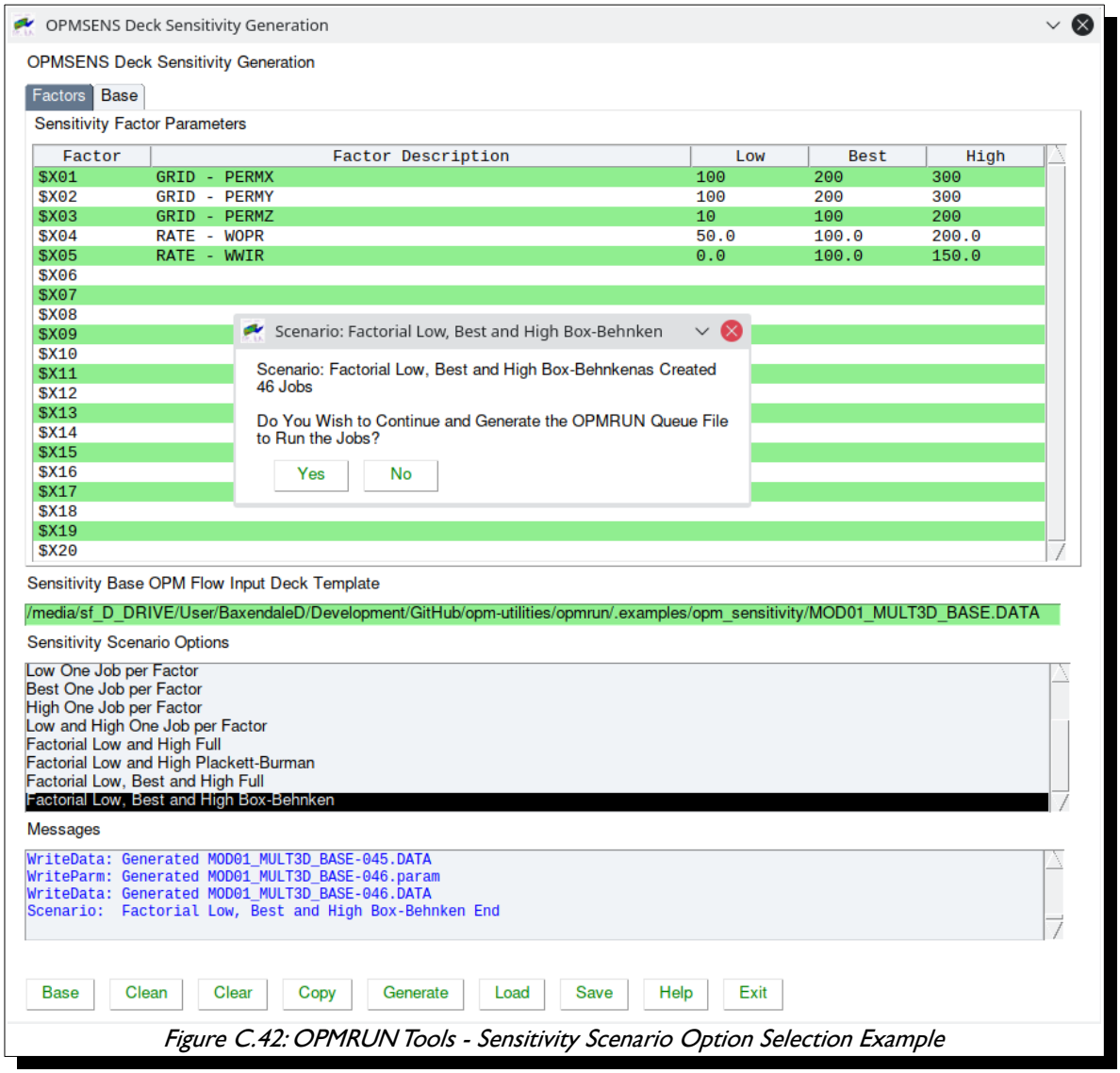


Figure C.42: OPMRUN Tools - Sensitivity Scenario Option Selection Example

If the Yes option is selected then the cases will be generated and the application will ask for the name of OPMRUN Queue file to write the jobs to, as depicted in

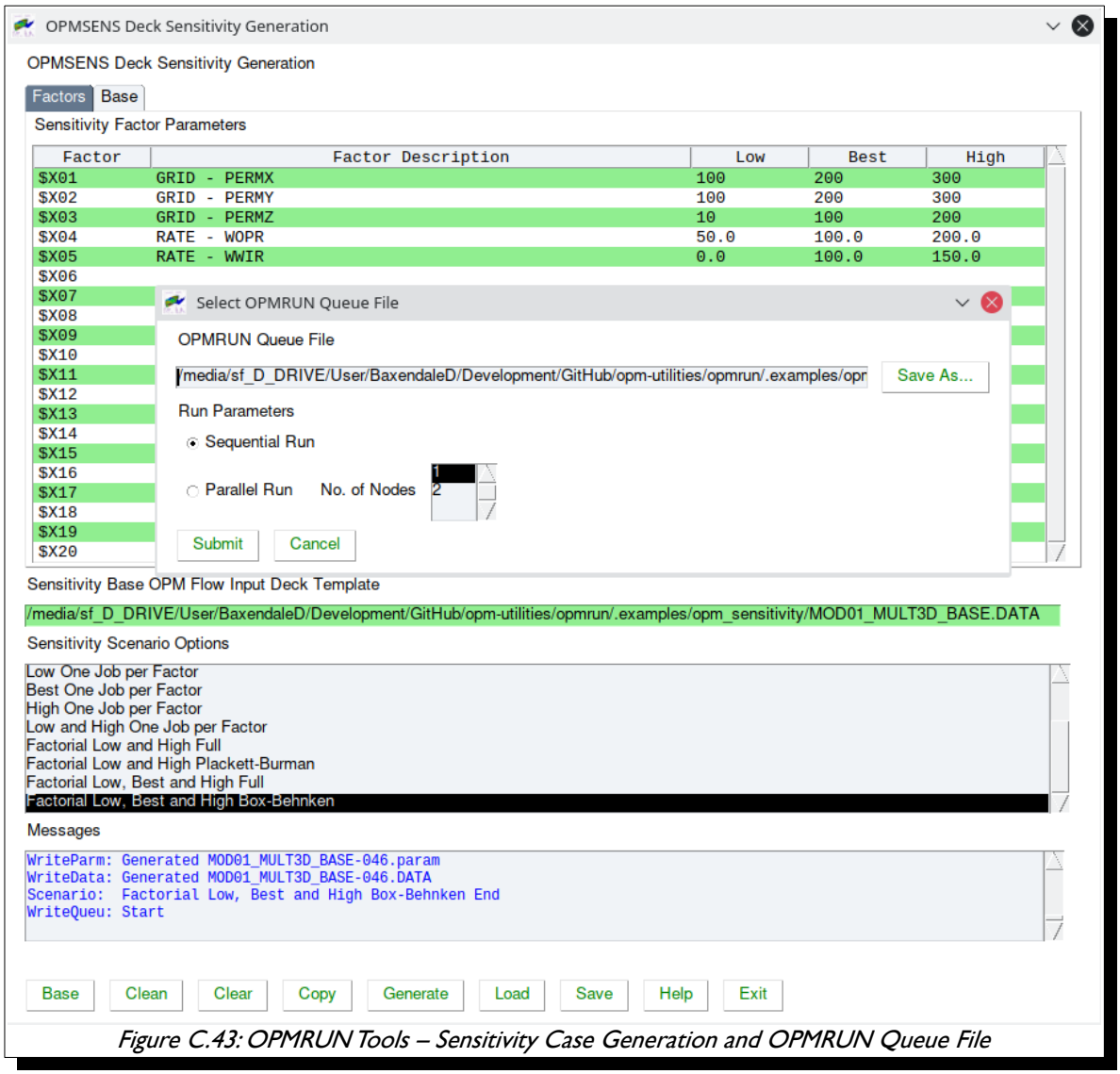


Figure C.43: OPMRUN Tools – Sensitivity Case Generation and OPMRUN Queue File

This allows the user to load the queue file into OPMRUN and to run all the jobs.

C.3.5 OPMRUN TOOLS: SIMULATOR INPUT/WELL SPECIFICATION

This tool, *Tools/Simulator Input/Well Specification*, uses the standard well export files from OPM ResInsight to reformat the data in a more user-friendly manner for the WELSPECS and COMPDAT keywords. Optionally, the application can generate the COMPLUMP keyword based on the OPM ResInsight layers file, with one completion per defined reservoir layer.

An example OPM ResInsight Exported Well Completion File Format(*.exp) is shown in Figure C.44

```
-- WELL          GROUP          BHP    PHASE  DRAIN  INFLOW  OPEN  CROSS  PVT    HYDS  FIP
-- NAME          NAME   I   J   DEPTH  FLUID  AREA   EQUANS  SHUT  FLOW   TABLE  DENS  REGN
WELSPECS
   GP-P1-06      1*     82  139 1*     OIL    0.0    STD     STOP  YES    0      SEG   0    /
   GP-P1-05      1*     72  89  1*     OIL    0.0    STD     STOP  YES    0      SEG   0    /
   GP-P1-01      1*     73  102 1*     OIL    0.0    STD     STOP  YES    0      SEG   0    /
   GP-P1-02      1*     79  112 1*     OIL    0.0    STD     STOP  YES    0      SEG   0    /
   GP-P2-04      1*     74  79  1*     OIL    0.0    STD     STOP  YES    0      SEG   0    /
    /
-- WELL                            OPEN   SAT   CONN           WELL      KH             SKIN       D      DIR
-- NAME          I    J     K1   K2  SHUT   TAB   FACT           DIA       FACT           FACT       FACT   PEN
COMPDAT
-- ---- Completions for completion type Perforation ----
-- Perforation Completion : MD In: 10115.8 - MD Out: 10125.6 Transmissibility: 0.0855562
   GP-P1-01       73   102   49   49   OPEN   1*    8.555621E-02   0.70900   8.712182E+01   2.00000    1*     'Z' /
-- Perforation Completion : MD In: 10125.6 - MD Out: 10136.6 Transmissibility: 0.153124
   GP-P1-01       73   102   50   50   OPEN   1*    1.531236E-01   0.70900   1.559257E+02   2.00000    1*     'Z' /
-- Perforation Completion : MD In: 10136.6 - MD Out: 10147.6 Transmissibility: 0.153119
   GP-P1-01       73   102   51   51   OPEN   1*    1.531194E-01   0.70900   1.559215E+02   2.00000    1*     'Z' /
-- Perforation Completion : MD In: 10147.6 - MD Out: 10154.2 Transmissibility: 0.141482
   GP-P1-01       73   102   52   52   OPEN   1*    1.414819E-01   0.70900   1.440710E+02   2.00000    1*     'Z' /
-- Perforation Completion : MD In: 10162.4 - MD Out: 10169.6 Transmissibility: 0.0383522
   GP-P1-01       73   102   53   53   OPEN   1*    3.835216E-02   0.70900   6.450968E+01   6.70000    1*     'Z' /
-- Perforation Completion : MD In: 10169.6 - MD Out: 10180.6 Transmissibility: 0.0356561
   GP-P1-01       73   102   54   54   OPEN   1*    3.565613E-02   0.70900   5.997487E+01   6.70000    1*     'Z' /
-- Perforation Completion : MD In: 10180.6 - MD Out: 10191.7 Transmissibility: 0.0030597
   GP-P1-01       73   102   55   55   OPEN   1*    3.059698E-03   0.70900   5.146520E+00   6.70000    1*     'Z' /
-- Perforation Completion : MD In: 10191.7 - MD Out: 10202.7 Transmissibility: 0.0116847
   GP-P1-01       73   102   56   56   OPEN   1*    1.168469E-02   0.70900   1.965406E+01   6.70000    1*     'Z' /
-- Perforation Completion : MD In: 10202.7 - MD Out: 10213.7 Transmissibility: 0.00139438
   GP-P1-01       73   102   57   57   OPEN   1*    1.394380E-03   0.70900   2.345397E+00   6.70000    1*     'Z' /
-- Perforation Completion : MD In: 10213.7 - MD Out: 10224.7 Transmissibility: 0.0584769
   GP-P1-01       73   102   58   58   OPEN   1*    5.847693E-02   0.70900   9.836029E+01   6.70000    1*     'Z' /
```

Figure C.44: OPMRUN Tools - Well Specification OPM ResInsight Well Completion File Format

And an OPM ResInsight Imported Formation Layer File (.Lyr) example is illustrated in Figure C.45

```
--
-- ResInsight Formation Names
-- Zone                Top   Base
-- Name                Layer Layer
--
'Pre-A'                 1 - 2
'Shale'                 3 - 3
'A Upper'               4 - 5
'Shale'                 6 - 6
'A Lower'               7 - 8
'A Lower Pay'           9 - 14
'Shale'                15 - 15
'B Upper'              16 - 18
'Shale'                19 - 19
'B Lower'              20 - 26
'Shale'                27 - 27
'C Upper'              28 - 30
'C Upper Inte'         31 - 36
'C Upper Lobe'         37 - 40
'Shale'                41 - 41
'C Lower'              42 - 47
'Shale'                48 - 48
'D Upper'              49 - 59
'D Middle'             60 - 72
'D Lower'              73 - 80
'Shale'                81 - 81
'D Lower Lobe 1'       82 - 87
'Shale'                88 - 88
'D Lower Lobe 2'       89 - 90
'Shale'                91 - 91
'E Sand'               92 - 97
'Shale'                98 - 98
```

Figure C.45: OPMRUN Tools - Well Specification OPM ResInsight Reservoir Layer File

The application also can generate a well a OPM ResInsight perforation file with the formation names for cross-checking the perforations.

The application user interface is shown in Figure C.46. Note that in Figure C.46 the *Output Header* options are used for comments only, no unit conversion is performed.

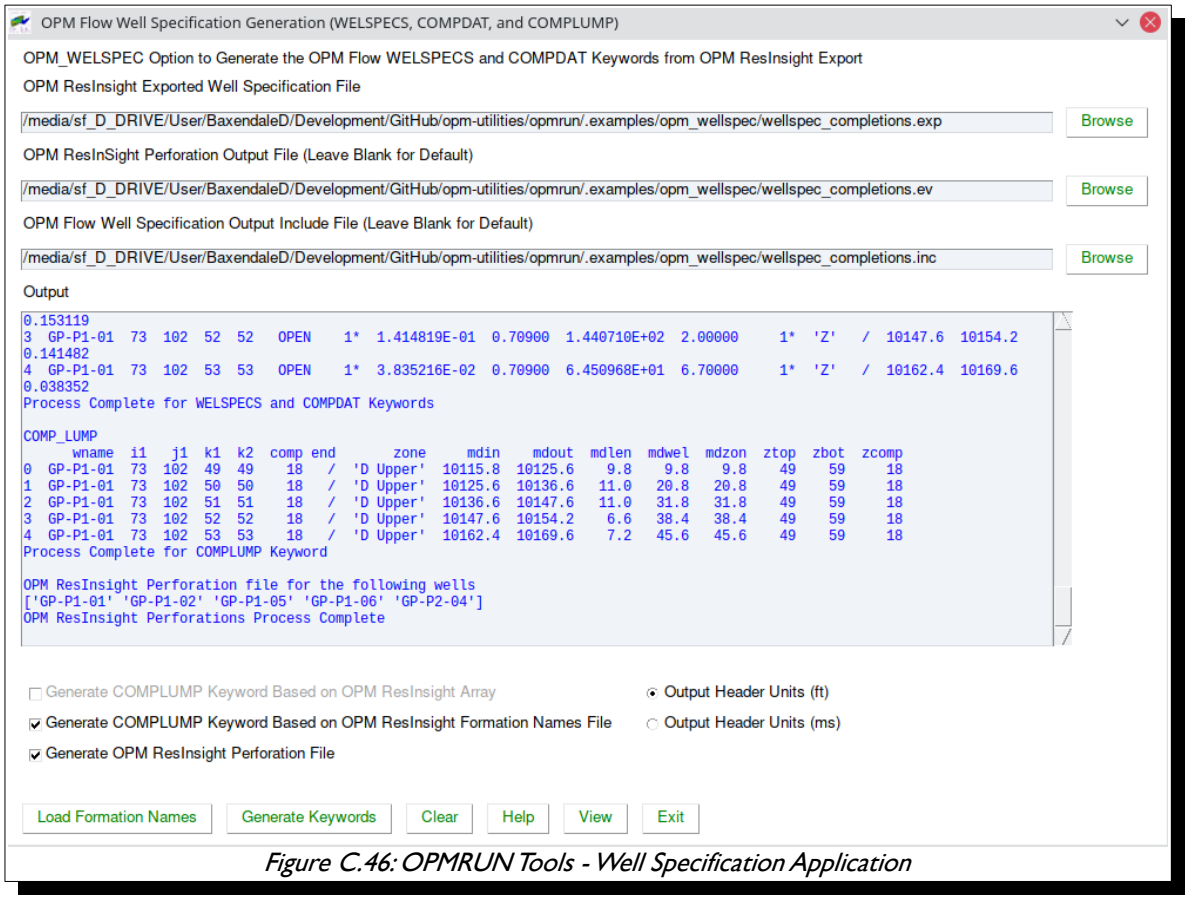


Figure C.46: OPMRUN Tools - Well Specification Application

In terms of output, the next figure shows the resulting well completion file to be used with OPM Flow, showing the WELSPECS and COMPDAT keywords (the COMPLUMP keyword is not shown in this example)

```
--
--
-- WELL SPECIFICATION DATA
--
-- WELL  GROUP LOC. LOC. BHP   PHASE DRAIN INFLOW OPEN CROSS PVT   HYDS FIP
-- NAME   NAME  I   J    DEPTH FLUID AREA  EQUANS SHUT FLOW  TABLE DENS REGN
WELSPECS
GP-P1-01    1*   73  102    1*   OIL   0.0    STD STOP   YES     0  SEG    0 /
GP-P1-02    1*   79  112    1*   OIL   0.0    STD STOP   YES     0  SEG    0 /
GP-P1-05    1*   72   89    1*   OIL   0.0    STD STOP   YES     0  SEG    0 /
GP-P2-04    1*   74   79    1*   OIL   0.0    STD STOP   YES     0  SEG    0 /
/
--
--  WELL CONNECTION DATA
--
-- WELL  LOC. LOC. LOC. LOC. OPEN SAT         CONN    WELL         KH       SKIN D    DIR      MD-IN   MD-OUT    CONN
-- NAME  II   JJ   K1   K2   SHUT TAB         FACT    DIA          FACT     FACT FACT PEN       MDFT    MDFT    FACTOR
COMPDAT
GP-P1-01   73  102   49   49 OPEN  1* 8.555621E-02 0.70900 8.712182E+01  2.00000   1* 'Z' /  10115.8  10125.6  0.085556
GP-P1-01   73  102   50   50 OPEN  1* 1.531236E-01 0.70900 1.559257E+02  2.00000   1* 'Z' /  10125.6  10136.6  0.153124
GP-P1-01   73  102   51   51 OPEN  1* 1.531194E-01 0.70900 1.559215E+02  2.00000   1* 'Z' /  10136.6  10147.6  0.153119
GP-P1-01   73  102   52   52 OPEN  1* 1.414819E-01 0.70900 1.440710E+02  2.00000   1* 'Z' /  10147.6  10154.2  0.141482
GP-P1-01   73  102   53   53 OPEN  1* 3.835216E-02 0.70900 6.450968E+01  6.70000   1* 'Z' /  10162.4  10169.6  0.038352
GP-P1-01   73  102   54   54 OPEN  1* 3.565613E-02 0.70900 5.997487E+01  6.70000   1* 'Z' /  10169.6  10180.6  0.035656
GP-P1-01   73  102   55   55 OPEN  1* 3.059698E-03 0.70900 5.146520E+00  6.70000   1* 'Z' /  10180.6  10191.7   0.00306
GP-P1-01   73  102   56   56 OPEN  1* 1.168469E-02 0.70900 1.965406E+01  6.70000   1* 'Z' /  10191.7  10202.7  0.011685
GP-P1-01   73  102   57   57 OPEN  1* 1.394380E-03 0.70900 2.345397E+00  6.70000   1* 'Z' /  10202.7  10213.7  0.001394
```

Figure C.47: OPMRUN Tools - Well Specification WELSPECS and COMPDAT Output Example

The final figure for this tool shows the resulting generated OPM ResInsight perforation file.

```
--
UNITS FIELD
--
--
-- DATE  PERFORATION      TOP      BOT    WELL    SKIN          ZONE
--                       MDFT     MDFT    DIAM    FACT
WELLNAME    GP-P1-01
"SOH"    perforation  10115.8  10125.6 0.70900 2.00000 -- 'D Upper'
"SOH"    perforation  10125.6  10136.6 0.70900 2.00000 -- 'D Upper'
"SOH"    perforation  10136.6  10147.6 0.70900 2.00000 -- 'D Upper'
"SOH"    perforation  10147.6  10154.2 0.70900 2.00000 -- 'D Upper'
"SOH"    perforation  10162.4  10169.6 0.70900 6.70000 -- 'D Upper'
"SOH"    perforation  10169.6  10180.6 0.70900 6.70000 -- 'D Upper'
"SOH"    perforation  10180.6  10191.7 0.70900 6.70000 -- 'D Upper'
"SOH"    perforation  10191.7  10202.7 0.70900 6.70000 -- 'D Upper'
"SOH"    perforation  10202.7  10213.7 0.70900 6.70000 -- 'D Upper'
"SOH"    perforation  10213.7  10224.7 0.70900 6.70000 -- 'D Upper'
"SOH"    perforation  10224.7  10228.7 0.70900 6.70000 -- 'D Upper'
```

Figure C.48: OPMRUN Tools - Well Specification OPM ResInsight Perforation File Output

C.3.6 OPMRUN TOOLS: RESINSIGHT

This option, *Tools/ResInsight*, loads the currently selected job into OPM ResInsight for viewing, this done via a Python sub-process call in OPMRUN, rather than using OPM ResInsight's Python API.

C.3.7 OPMRUN TOOLS: WELL TRAJECTORY CONVERSION

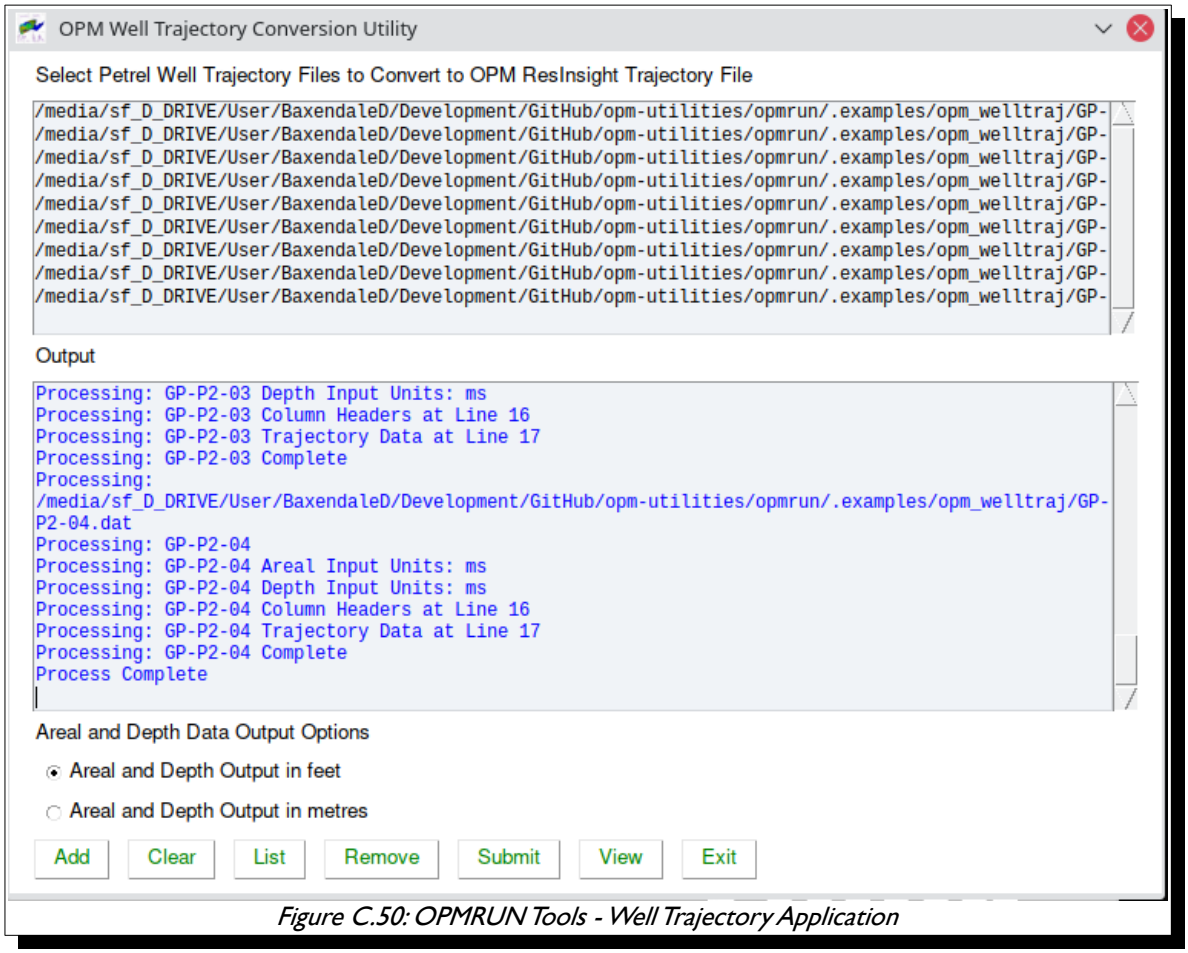
OPM ResInsight can read well trajectories in a given format into the program, the *Tools/Well Trajectory Conversion* option converts a Schlumberger Petrel exported well trajectory file, as shown Figure C.49, into a OPM ResInsight well trajectory file containing all the wells.

```
# WELL TRACE FROM PETREL
# WELL NAME:              GP-P1-01
# DEFINITIVE SURVEY:      Explicit survey 1
# WELL HEAD X-COORDINATE: 294774.82000000 (m)
# WELL HEAD Y-COORDINATE: 2623340.30000000 (m)
# WELL DATUM (KB, Kelly bushing, from MSL): 13.60000000 (m)
# WELL TYPE:              GAS
# MD AND TVD ARE REFERENCED (=0) AT WELL DATUM AND INCREASE DOWNWARDS
# ANGLES ARE GIVEN IN DEGREES
# XYZ TRACE IS GIVEN IN COORDINATE SYSTEM [Petrel]
# AZIM_TN: azimuth in True North
# AZIM_GN: azimuth in Grid North
# DX DY ARE GIVEN IN GRID NORTH IN m-UNITS
# DEPTH (Z, tvd_z) GIVEN IN m-UNITS
#=============================================================================================================================================
      MD           X           Y           Z          TVD          DX          DY       AZIM_TN        INCL         DLS        AZIM_GN
#=============================================================================================================================================
 0.0000000000 294774.82000 2623340.3000 13.600000000 0.0000000000 -0.000000001 -0.000000002 359.19018141 0.0000000000 0.0000000000 0.0000000000
 224.20000000 294774.88839 2623341.0993 -210.5980866 224.19808660 0.0683874065 0.7993438398 4.0801814067 0.4100000000 0.0548617306 4.8900000000
 252.80000000 294774.98621 2623341.2814 -239.1972995 252.79729950 0.1662053919 0.9814199335 47.230181407 0.4800000000 0.3500120164 48.040000000
 282.20000000 294775.19584 2623341.4957 -268.5957535 282.19575352 0.3758445395 1.1957111340 41.010181407 0.6900000000 0.2235585568 41.820000000
 311.10000000 294775.59178 2623341.9055 -297.4898516 311.08985156 0.7717802449 1.6054957099 44.170181407 1.5700000000 0.9154355591 44.980000000
 339.70000000 294776.51776 2623342.6035 -326.0654115 339.66541154 1.6977563415 2.3034936697 56.230181407 3.1000000000 1.6768888626 57.040000000
 368.60000000 294778.20113 2623343.6423 -354.8966452 368.49664522 3.3811279249 3.3422523835 58.350181407 4.7500000000 1.7191224028 59.160000000
 397.50000000 294780.74744 2623345.2481 -383.6372139 397.23721390 5.9274355074 4.9480945668 56.030181407 7.2100000000 2.5654105761 56.840000000
 426.10000000 294784.30376 2623347.6883 -411.9070216 425.50702158 9.4837615496 7.3882779982 53.810181407 10.140000000 3.0928640683 54.620000000
 454.90000000 294788.99742 2623351.1132 -440.1115661 453.71156609 14.177422776 10.813246789 52.500181407 13.140000000 3.1369063547 53.310000000
 483.90000000 294794.78585 2623355.5409 -468.1780891 481.77808913 19.965846340 15.240878427 51.180181407 15.970000000 2.9474446939 51.990000000
```

Figure C.49: OPMRUN Tools - Well Trajectory Conversion Input File

The utility allows for the multiple wells to be converted at once and for conversion of units.

Note in some areas of the world it is not uncommon for the areal units to be in UTM and the depth to be in feet. This configuration is also handled by the application.



Select Petrel Well Trajectory Files to Convert to OPM ResInsight Trajectory File

```
/media/sf_D_DRIVE/User/BaxendaleD/Development/GitHub/opm-utilities/opmrun/.examples/opm_welltraj/GP-
/media/sf_D_DRIVE/User/BaxendaleD/Development/GitHub/opm-utilities/opmrun/.examples/opm_welltraj/GP-
/media/sf_D_DRIVE/User/BaxendaleD/Development/GitHub/opm-utilities/opmrun/.examples/opm_welltraj/GP-
/media/sf_D_DRIVE/User/BaxendaleD/Development/GitHub/opm-utilities/opmrun/.examples/opm_welltraj/GP-
/media/sf_D_DRIVE/User/BaxendaleD/Development/GitHub/opm-utilities/opmrun/.examples/opm_welltraj/GP-
/media/sf_D_DRIVE/User/BaxendaleD/Development/GitHub/opm-utilities/opmrun/.examples/opm_welltraj/GP-
/media/sf_D_DRIVE/User/BaxendaleD/Development/GitHub/opm-utilities/opmrun/.examples/opm_welltraj/GP-
/media/sf_D_DRIVE/User/BaxendaleD/Development/GitHub/opm-utilities/opmrun/.examples/opm_welltraj/GP-
/media/sf_D_DRIVE/User/BaxendaleD/Development/GitHub/opm-utilities/opmrun/.examples/opm_welltraj/GP-
```

Output

```
Processing: GP-P2-03 Depth Input Units: ms
Processing: GP-P2-03 Column Headers at Line 16
Processing: GP-P2-03 Trajectory Data at Line 17
Processing: GP-P2-03 Complete
Processing:
/media/sf_D_DRIVE/User/BaxendaleD/Development/GitHub/opm-utilities/opmrun/.examples/opm_welltraj/GP-
P2-04.dat
Processing: GP-P2-04
Processing: GP-P2-04 Areal Input Units: ms
Processing: GP-P2-04 Depth Input Units: ms
Processing: GP-P2-04 Column Headers at Line 16
Processing: GP-P2-04 Trajectory Data at Line 17
Processing: GP-P2-04 Complete
Process Complete
```

Areal and Depth Data Output Options

● Areal and Depth Output in feet

○ Areal and Depth Output in metres

Figure C.50: OPMRUN Tools - Well Trajectory Application

An example output file is shown in Figure C.51

```
-- WELL TRACE FROM PETREL
-- WELL NAME:              GP-P1-01
-- DEFINITIVE SURVEY:      Explicit survey 1
-- WELL HEAD X-COORDINATE: 294774.82000000 (m)
-- WELL HEAD Y-COORDINATE: 2623340.30000000 (m)
-- WELL DATUM (KB, Kelly bushing, from MSL): 13.60000000 (m)
-- WELL TYPE:              GAS
-- MD AND TVD ARE REFERENCED (=0) AT WELL DATUM AND INCREASE DOWNWARDS
-- ANGLES ARE GIVEN IN DEGREES
-- XYZ TRACE IS GIVEN IN COORDINATE SYSTEM [Petrel]
-- AZIM_TN: azimuth in True North
-- AZIM_GN: azimuth in Grid North
-- DX DY ARE GIVEN IN GRID NORTH IN m-UNITS
-- DEPTH (Z, tvd_z) GIVEN IN m-UNITS
--
-- APPLIED AREAL CONVERSION FACTOR: 3.28083989501312
-- APPLIED DEPTH CONVERSION FACTOR: 3.28083989501312
--
WELLNAME:GP-P1-01
--          XCORD          YCORD      TVDSS         MD
--                                    DEPTH      DEPTH
--             ft             ft         ft         ft
-- -------------- -------------- ---------- ----------
     9.671090e+05   8.606760e+06     -44.62       0.00
     9.671092e+05   8.606762e+06     690.94     735.56
     9.671095e+05   8.606763e+06     784.77     829.40
     9.671102e+05   8.606763e+06     881.22     925.85
     9.671115e+05   8.606765e+06     976.02    1020.67
     9.671146e+05   8.606767e+06    1069.77    1114.50
     9.671201e+05   8.606770e+06    1164.36    1209.32
     9.671284e+05   8.606776e+06    1258.65    1304.13
     9.671401e+05   8.606784e+06    1351.40    1397.97
     9.671555e+05   8.606795e+06    1443.94    1492.45
     9.671745e+05   8.606810e+06    1536.02    1587.60
     9.671965e+05   8.606827e+06    1626.60    1682.41
     9.672208e+05   8.606846e+06    1715.81    1776.90
     9.672483e+05   8.606868e+06    1803.86    1871.72
```

Figure C.51: OPMRUN Tools - Well Trajectory Output Trajectory File.

APPENDIX D: PYTHON INTERFACE TO OPM FLOW

APPENDIX TABLE OF CONTENTS

D.1 INTRODUCTION

Some of the C++ code in OPM Flow has been "wrapped" in Python which means that one can invoke the simulator's C++ code from one's own Python programs. At the moment the wrappers for the input layer and the code for working with the result files is quite complete and usable. Work is also underway to be able to run the simulator and to interact with running simulations from within Python.

The goal is for the Python code to be structured like a native Python API, but some design decisions are certainly influenced by the underlying C++ implementation, and to get a deeper understanding of what is possible and how to achieve specialized tasks one might need to consult the simulator's C++ code.

Note

Observe that the Python bindings described here are not very mature. The API might change in future releases. It is in general quite simple to expose new functionality to Python.

D.2 WORKING WITH THE INPUT DECK

The reading of the input deck in OPM Flow is a two step process, first a data structure called a *"Deck"* is created - the *"Deck"* is essentially a collection of keywords where all elements have been converted to the correct type and default values have been injected into the keyword. The *"Deck"* is a quite low level data structure, and the actual simulation is based on higher level data structures where the origin in "keywords" from a *.DATA file is no longer so apparent. The most notable high level objects are the *"EclipseState"* and *"Schedule"* classes. Large parts of this functionality is available from Python.

D.2.1 THE PARSER OBJECT

The initial building block in the parsing process is the *"Parser"* object which knows how to interpret the input keywords and create data structures. Virtually all scripts working with the input files will start with creating a Parser object:

```
#!/usr/bin/env python3
from opm.io.parser import Parser
# Create a parser object which can be used to parse a string or input files
parser = Parser()
```

In most cases one will just create a default parser object and be done with it, but it is possible to both add your own keyword definitions to the parser and alternatively create a custom parser which only accepts a subset of keywords. These features will be demonstrated briefly in section *Special Parsing*.

D.2.2 LOADING AN INPUT DECK

The *"Deck"* data structure is essentially a list of keywords which have been loaded as *"DeckKeyword"* instances. The Deck can either be loaded from an input data file or from a string. The following example shows how to load two SCHEDULE section keywords, WELSPECS and COMPDATA:

```
#!/usr/bin/env python3
from opm.io.parser import Parser
deck_string = """
WELSPECS
   'W1'  'G1'   19    4      1*   WATER   1*   1*   SHUT  1*  1*    1*      /
/
COMPDAT
  'W1'   11   3    1    5   OPEN   1*   1*   0.216   1*   0    1*   Z    1* /
/
"""
parser = Parser()
deck = parser.parseString( deck_string )
```

After running this small script the deck variable will contain the two keywords WELSPECS and COMPDAT. For these keywords the '*' have been replaced with the correct default value and the items have been converted to the correct type. In addition, all numerical values have been converted to SI units, more about units can be found in section *Units*. Alternatively you can use the Parser.parseFile() method to parse an entire input file. In this case INCLUDE and IMPORT keywords will be resolved and everything will be coalesced into one large *"Deck*Keyword" data structure:

```
#!/usr/bin/env python3
 import os.path
 import sys
 from opm.io.parser import Parser
 data_file = sys.argv[1]
 print(f"Loading deck from {data_file}")
 parser = Parser()
 deck = parser.parseFile( data_file )
```

The code above loads a *.DATA file given by calling the Python script. For example, if the name of the input deck, that may contain a complete input deck or just a selection of keywords, is called NORNE.DATA, and the name of the above script is LoadDeck.py, then:

```
python LoadDeck NORNE.DATA
```

would load the NORNE.DATA into the simulator.

D.2.3 UNITS

Internally in OPM Flow all quantities are managed as SI units, whereas input decks use the unit systems METRIC, FIELD, or LAB (laboratory). The METRIC unit system is comparable to SI units, the most notable difference is that time is expressed in days instead of seconds, the FIELD unit system is based on historical units like barrels of oil (stb) and feet for length.

The important point is that when you access the elements in the Deck the values you will get are in SI units, whereas the input you provided has been in one of other set of units. Consider the example:

```
#!/usr/bin/env python3
from opm.io.parser import Parser
deck_string = """
WCONPROD
 'W1'  'OPEN'   'ORAT'  86400 /
/
"""
parser = Parser()
deck = parser.parseString( deck_string )
kw = deck[0]
record0 = kw[0]
orat = record0["ORAT"].value
well = record0["WELL"].value
print("==> The Oil rate in well:{} is {} m3/s".format(orat, well))
```

When this script is run it will print:

```
==> The Oil rate in well:W1 is 1.0 m3/s
```

because the input value of 86,400 m3/day is internally converted to 1.0 m3/second.

The default input unit system is METRIC, that means that if the input is to be interpreted in either FIELD or LAB units, then one needs to add a keyword to declare the units before the entering the data. Thus, to interpret the WCONPROD keyword's oil rate as stb/d in the previous example, the FIELD keyword needs to be added to the example, that is:

```
#!/usr/bin/env python3
   from opm.io.parser import Parser
   deck_string = """

   FIELD

   WCONPROD
    'W1'  'OPEN'   'ORAT'  86400 /
   /
   """
```

D.2.4 ACCESSING THE INPUT DECK

After you have loaded a Deck you can query it with normal the Python functions:

```
# Check if deck has keyword:
if "GRID" in deck:
    print("Deck contains 'GRID' keyword")
else
    print("Deck does not have 'GRID' keyword")

# Loop through all the keywords:
for kw in deck:
    print("kw: {}".format(kw.name)
```

To get a keyword from the deck you can access it with index, name or a combination of name and occurrence index:

```
# Get keyword 10
kw10 = deck[10]

# Get the last DATES keyword
last_dates = deck["DATES"]

# Get the third WELSPECS keyword
welspecs3 = deck[("WELSPECS", 3)]
```

A deck keyword is composed of *"records"*, which are again composed of *"items"*. The example below shows how one can iterate through the wells in a WELSPECS keyword:

```
kw = deck["WELSPECS"]
print(f"Keyword has {len(kw)} records")
for record in kw:
    well = record["WELL"].get_str()
    group = record[1].get_str()

    print(f"Well {num}: {well} is part of group{group})
```

D.2.5 SPECIAL PARSING

The keywords OPM Flow recognizes are configured via JSON files which are embedded in the source distribution. The JSON keywords for the simulator are in following the public GitHub repository:

https://github.com/OPM/opm-common/tree/master/src/opm/parser/eclipse/share/keywords

By default the Parser class will recognize all the keywords which are known to OPM Flow, but one can create your own custom parser with only a subset of keywords:

```
from opm.io.parser import Builtin
# Create a special parser which only recognizes the corner point grid
   # keywords
parser = Parser(add_default = False)
builtin = Builtin()
parser.add_keyword( builtin.COORD )
parser.add_keyword( builtin.ZCORN )
parser.add_keyword( builtin.ACTNUM )
```

The opposite is also possible, here is how one can add a special keyword GCLOSE which could be used to close all the wells in the group:

```
# Create a fictitious home mode keyword GCLOSE:
GCLOSE = {"name" : "GCLOSE",
          "sections" : ["SCHEDULE"],
          "items" : [
             {"name" : "GROUP", "value_type" : "STRING"}
           ]}

parser = Parser()
# Add the keyword description via a JSON string
parser.add_keyword(json.dumps(GCLOSE))
```

By default the parser used in OPM Flow is quite strict - a tad stricter than the one used by the commercial simulator. However you one configure how the parser should react to certain error conditions. By default the parser will fail with an exception if an INCLUDE file is not found, but one can for instance change this behavior to ignore that error condition, for example:

```
parse_context = ParseContext([('PARSE_MISSING_INCLUDE',
                               opm.io.action.ignore)])
parser = Parser()
deck = parser.parseFile(self.norne_fname, parse_context)
```

D.2.6 CREATING ECLIPSESTATE AND GRID OBJECTS

The *"Deck"* is a quite raw structure and not very user friendly. Before actually used in the simulator the properties from the *"Deck"* are coalesced to a form more easily usable format. Consider, for example the PORO array configured below where the configuration is given in three steps:

```
-- GRID Dimensions 10 x 10 x 3
DIMENS
  10 10 3 /
--
-- Set A Global Value 0.10 For All Of The Grid
--
PORO
   300*0.10 /
--
-- Set A Value Of 0.15 For All The Cells In The Middle Layer
--
BOX
   1 10 1 10 2 2 /
PORO
   100*0.15 /
ENDBOX
--
-- Scale The Bottom Layer By A Factor OF Two
--
MULTIPLY
   PORO 2 1 10 1 10 3 3 /
/
```

In the deck representation this will be a collection of five different keywords: {"PORO", "BOX", "PORO", "ENDBOX", "MULTIPLY"}, whereas in the *"EclipseState"* this will be one property "PORO" where all the modifications have been completed. In addition to the grid properties like "PERMX" and "PORO" the *"EclipseState"* object contains numerous other objects like the fault properties, PVT tables and more.

```
#!/usr/bin/env python3
import syshe
form opm.io.parser import Parser
from opm.io.ecl_state import EclipseState
parser = Parser()
data_file = sys.argv[1]
deck = parser.parseFile( data_file )
es = EclipseState( deck )
#
# Get the FieldPropsManager from the EclipseState. The FieldPropsManager
# is then used to query and look up grid properties like PERMX and PORO.
# Observe that the properties you get from the FieldPropsManager only have
# the active cels only.
#
fp = es.field_props()
poro = fp["PORO"]
satnum = fp["SATNUM"]
#
# Get the input grid from the EclipseState:
#
grid = es.grid()
# Get a manager for all the tables
tables = es.tables()
```

D.2.7 CREATING A SCHEDULE

The Schedule object contains all the dynamic information in the model, in particular that includes all information about wells and groups. The Schedule object is constructed from the *"Deck"* and the "*EclipseState"* objects.:

```
#!/usr/bin/env python3
import sys
from opm.io.parser import Parser
from opm.io.ecl_state import EclipseState
parser = Parser()
data_file = sys.argv[1]
deck = parser.parseFile( data_file )
es = EclipseState( deck )
schedule = Schedule(deck, es)
print(f"Schedule file has {len(schedule)} report steps")'
print("List of wells")
for well in schedule.well_names():
    print(well)
```

The example above prints the number of report time steps and all the well names in the SCHEDULE section.

D.3 WORKING WITH THE RESULT FILES

In addition to the functionality to load and inspect the input deck, OPM Flow also has a Python interface to load and inspect the various result files, SUMMARY and RESTART files. There is also some functionality to create files with the correct formatting.

D.3.1 LOADING A GRID FILE

Using the class *"EclipseGrid"* one can load a grid representation from a *.EGRID file on disk:

```
#!/usr/bin/env python3
import sys
from opm.io.ecl import EGrid
grid_file = sys.argv[1]
grid = EGrid(grid_file)
```

The return value from *"EclipseState.ecl_grid()"* is also of this type.

D.3.2 LOADING A SUMMARY CASE OF RESULTS

Using the class *"ESMry"* one can load the Extended SUMMARY file for a simulations that contains all the SUMMARY vectors.

```
#!/usr/bin/env python3
import sys
from opm.io.ecl import ESMry
case = sys.argv[1]
summary = Esmry(case)
```

D.3.3 LOADING A RESTART FILE

Finally, one load the solution arrays stored in the *.RESTART file using the Erst class, as shown below.

```
#!/usr/bin/env python
import sys
from opm.io.ecl import ERst

rst_file = sys.argv[1]
rst = Erst(rst_file)
```

Note that restart file can be large, so care must be taken on how these type of files are loaded into Python.

APPENDIX E: RUNNING PREVIOUS RELEASES OF OPM FLOW

APPENDIX TABLE OF CONTENTS

E.1 RUNNING OPM FLOW 2023-04

The command line syntax for running OPM Flow is:

```
flow [OPTIONS] CASENAME
```

and typing the following command from your terminal:

```
flow CASENAME
```

will start the simulator and run the case specified by CASENAME.DATA.

OPM Flow accepts command line options to control various aspects of the simulator's run time behavior, as described in the table below. To give a command line option use "key=value" syntax, with no spaces around the equals sign. It is also possible to put multiple options together in a parameter file. To do so, put one option on each line of the file with ".param" as the extension and pass that filename as a command line parameter to OPM Flow using:

```
flow --parameter-file=CASENAME.PARAM
```

to instruct OPM Flow to read the parameter file.

OPM Flow 2023-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
1	-h or --help	A character string that causes OPM Flow to print a help message that gives a brief description of the available command line parameters. **This now only prints the active user facing command line options.**	N/A
2	--help-all	Prints all the command line options included in the release, including obsolete, hidden and deprecated options.	N/A
3	--accelerator-mode	A defined character string that defines the usage of the GPU (cusparseSolver or openclSolver) or FPGA (fpgaSolver) as the linear solver, usage '--accelerator-mode=[none|cusparse|opencl|fpga|amgcl]'.	"none"
4	--allow-distributed-wells	A boolean value set to true of false that allows the perforations of a well to be distributed to interior of multiple processes in a parallel run (true), or not (false).	false
5	--alternative-well-rate-init	A boolean value set to true of false that causes the simulator to use an alternative well rate initialization procedure (true).	true
6	--bda-device-id	Choose device ID for cusparseSolver or openclSolver, use 'nvidia-smi' or 'clinfo' commands to determine valid IDs.	0
7	--continue-on-convergence-error	A Boolean value set to true or false that that instructs the simulator to continue (true) with a non-converged solution instead of giving up (false) if it encounter a time step size smaller than the minimum	false

OPM Flow 2023-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
		time step size.	
8	--cpr-reuse-interval	A positive integer that sets the reuse preconditioner interval. Used when *--cpr-reuse-setup* is set to 4, then the preconditioner will be fully recreated instead of reused every N linear solve, where N is this parameter.	30
9	--cpr-reuse-setup	A positive integer that defines if the CPR solver should re-use the AMG setup. Valid options are: 3 : Recreate the preconditioner for every linear solve. 4 : Recreate once every time step. 5 : Recreate if last linear solve took more than 10 iterations. 6 : Never recreate. 7 : Recreate every N linear solves, where N is the parameter *--cpr-reuse-interval*. Changed the default value from three to four.	4
10	--dbph-max-rel	A real positive value that sets maximum relative change of the bottom-hole pressure in a single iteration	1.0
11	--dp-max-rel	A real positive double precision value that sets the maximum allowed relative change of pressure per iteration.	0.3
12	--ds-max	A real positive double precision value that sets the maximum allowed change in saturation per iteration.	0.2
13	--dwell-fraction-max	A real positive value that sets the maximum allowed change in well's volume fraction per iteration.	0.2
14	--ecl-deck-file-name	A character string that defines the name of the OPM Flow input file which contains the simulator's ECLIPSE formatted input deck to be simulated. Note that there must be no spaces in the filename. If there are spaces, which can typically occur on Windows based systems, then the filename should be enclosed in quotes; thus instead of FILE NAME use "FILE NAME".	""
15	--ecl-enable-drift-compensation	A Boolean value set to true or false that enables (true) or disables (false) partial compensation of systematic mass losses via the source term of the next time step.	true

OPM Flow 2023-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
16	--ecl-output-double-precision	A Boolean value set to true or false that switches on (true) or off (false) double precision in restart files. Useful for 'perfect' restarts.	false
17	--ecl-output-interval	An integer value defining the number of report steps that ought to be skipped between two writes of restart files results.	-1
18	--edge-weights-method	A defined positive integer value that defines the edge-weighing strategy: 0 : For uniform, 1 : for trans, or 2 : for log(trans).	1
19	--enable-adaptive-time-stepping	A Boolean value set to true or false that turns on (true) or off (false) adaptive time stepping. If set to false the report time steps provided by the input deck are used.	true
20	--enable-async-ecl-output	A Boolean value set to true or false that sets the output mode to be asynchronously (true), letting the simulator continue computing the next time step while writing results to restart and summary files in the commercial simulators format.	true
21	--enable-async-vtk-output	A Boolean value set to true or false that sets the output mode to be asynchronously (true), letting the simulator continue computing the next time step while writing results to the VTK output files.	true
22	--enable-dry-run	A Boolean value set to true or false that specifies if the simulation should actually run, (true) or just check the input deck (false). This option is equivalent to activating the NOSIM keyword in the RUNSPEC section of the input deck (see section *5.3.97 NOSIM – Activate the No Simulation Mode for Data File Checking*). The default value of "auto" will use whatever is stipulated in the input deck via the NOSIM keyword.	"auto"
23	--enable-ecl-output	A Boolean value set to true or false that specifies if the binary output files (restart, summary files, etc.) should be written in the commercial simulator's format (true), or OPM Flow's format (false).	true
24	--enable-esmry	A Boolean value that switches on (true) or off (false) the output SUMMARY vectors to the ESMRY file for fast loading of summary data.	false

OPM Flow 2023-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
25	--enable-logging-fallout-warning	A Boolean value set to true or false that sets the developer option to see whether logging was on non-root processors (true). If set to true output will be appended to the *.DBG or *.PRT files.	false
26	--enable-opm-rst-file	A Boolean value set to true or false to write OPM specific data sets to the commercial simulators restart file to enable restart of an OPM Flow runs (true) by OPM Flow, or not to write the data (false).	false
27	--enable-storage-cache	A Boolean value set to true or false that turns on (true) or off (false) storing previous storage terms and avoid re-calculating them.	true
28	--enable-terminal-output	A Boolean value set to true or false that turns on (true) or off (false) high-level information about the simulation's progress to the terminal	true
29	--enable-tuning	A Boolean value set to true or false that instructs OPM Flow to read the time stepping parameters from the *TUNING - Numerical Tuning Control* keyword in the input deck, if set to true. Note that only the first record of the TUNING keyword is processed.	false
30	--enable-vtk-output	A Boolean value set to true or false that turns on (true) or off (false) a global switch for writing VTK files.	false
31	--enable-well-operability-check	A Boolean value set to true or false that enables (true) checking of a well's operating status, or disables (false) the checking.	true
32	--enable-well-operability-check-iter	A Boolean value set to true or false that enables (true) checking of a well's operating status during iterations, or disables (false) the checking during iterations.	false
33	--enable-write-all-solutions	A Boolean value set to true or false that turns on (true) or off (false) the writing of all solutions to disk instead of only the ones for the report steps.	false
34	--force-disable-fluid-in-place-output	A Boolean value set to true or false that instructs OPM Flow not to print the Fluid In-Place report after each report time step (true) or not (false). Note this parameter will override the print request in the input deck.	false
35	--force-disable-resv-fluid-in-place-output	A Boolean value set to true or false that	false

OPM Flow 2023-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
		instructs OPM Flow to not to print the Reservoir Volume Fluid In-Place report after each report time step (true) or not (false). Note this parameter will override the print request in the input deck.	
36	--full-time-step-initially	A Boolean value set to true or false that instructs OPM Flow to always attempt to finish a report step using a single time step (true) or not (false).	false
37	--ignore-keywords	A character string that defines a list of keywords which should be ignored by OPM Flow. The keyword string should be enclosed in quotes and each keyword separated by a colon, that is ':' .	""
	ILU Preconditioner Parameters		
38	--ilu-fillin-level	A positive integer value that sets the fill in level for the ILU preconditioner.	0
39	--ilu-redblack	A Boolean value set to true or false that instructs OPM Flow to use red-black partitioning for the ILU preconditioner. (true) or not (false).	false
40	--ilu-relaxation	A real positive double precision value that sets the relaxation factor of the linear solver's ILU preconditioner	0.9
41	--ilu-reorder-spheres	A Boolean value set to true or false that specifies OPM Flow to reorder the entries of the matrix in the red-black ILU preconditioner in spheres starting at an edge (true) or not (false) If false the original ordering is preserved in each color. Otherwise try to ensure D4 ordering (in a 2D structured grid, the diagonal elements are consecutive).	false
42	--initial-time-step-in-days	A real double precision value that sets the size of initial time step in days.	1.0
	Linear Solver Parameters		
43	--linear-solver	A defined quoted character string that sets the configuration of the linear solver, valid values are: 1) ilu0 (default), 2) cpr (an alias for cpw)[361] [362] and [363]	"ilu0"

[361] *Wallis, J. R., Little, T. E., and Nolen, J. S.: "Constrained Residual Acceleration of Conjugate Residual Methods," paper SPE 13536 presented at the SPE Reservoir Simulation Symposium, Dallas, Texas, USA (February 10-13, 1985).*

[362] *R. Scheichl, M. Roland, J. Wendebourg, Decoupling and block preconditioning for sedimentary basin simulations, Computational Geosciences 7 (2003) 295{318.*

OPM Flow 2023-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
		3) cpr_quasiimpes, 4) cpr_trueimpes, 5) cprw, 6) amg,[364] or 7) a file name that has the extension ".json", that contains the linear solver configuration parameters. Option (5) extends the existing Constrained Pressure Residual ("CPR") preconditioner to include wells. This option can also be invoked via the CPR keyword in the RUNSPEC section; however, the command line parameter takes precedence. For option (6) one enters a character string enclosed in quotes that defines the filename of a JSON configuration file for the flexible linear solver system. In this case the file extension should be '.json'. Note that the *.PRT file contains the "Property tree for the linear solver" listing, which is the JSON specification of the current case, and can be used to configure a user specific linear solver JSON file. The default is "ilu0". **The option "cpr" now is an alias for "cprw" instead of "cpr_trueimpes".**	
44	--linear-solver-ignore-convergence-failure	A Boolean value set to true or false that if set to true convergences failures in the linear solver are ignored. This option should be used with care, as the results might be unreliable.	false
45	--linear-solver-max-iter	A positive integer value that defines the maximum number of linear iterations.	200
46	--linear-solver-reduction	A real positive double precision value that sets the minimum reduction of the residual for the linear solver. The linear solver convergences when the residual is reduced sufficiently. **The simulator now overrides the default reduction to be 0.005 instead of 0.01, if the linear solver has been set to one of the cpr options, in a similar manner as how the default maximum number of linear iterations for the cpr and cprw options are changed to 20 instead of**	0.01

[363] *Klemetsdal, Ø.S., Møyner, O. & Lie, KA. Accelerating multiscale simulation of complex geomodels by use of dynamically adapted basis functions. Comput Geosci 24, 459–476 (2020). https://doi.org/10.1007/s10596-019-9827-z.*

[364] *M. Blatt, A parallel algebraic multigrid method for elliptic problems with highly discontinuous coefficients, Ph.D. thesis, Ruprecht-Karls-Universität Heidelberg (2010).*

OPM Flow 2023-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
		100, unless specified by the command line option by the user.	
47	--linear-solver-restart	A positive integer value that sets the number of iterations after which GMRES is restarted.	40
48	--linear-solver-verbosity	A positive integer value that defines the output from linear solver: 0 : No extra output, 1 : output per solution iteration, or 2 : output per iteration.	0
49	--load-step	An integer value that determines if the simulator should load the serialized state from OPM Flow's version of the restart file. and should be set to either a specific report step, or 0 to load the last stored report step. The default value of -1 does not load the data from the OPM Flow specific restart file. OPM Flow's version of the restart file, is written using the *--save-step=N* option.	-1
50	--matrix-add-well-contributions	A Boolean value set to true or false that if set to true explicitly specifies the influences of wells between cells in the Jacobian and preconditioner matrices.	false
51	--max-inner-iter-ms-wells	A positive integer value that defines the maximum number of inner iterations for multi-segment wells.	100
52	--max-inner-iter-wells	A positive integer value that defines the maximum number of inner iterations for standard wells.	50
53	--max-newton-iterations-with-inner-well-iterations	A positive integer that specifies the maximum newton iterations with inner well iterations.	8
54	--max-pressure-change-ms-wells	A real positive value that defines the maximum relative pressure change for a single iteration of the multi-segment well model.	1.0×10^{6}
55	--max-residual-allowed	A real positive value that sets the absolute maximum tolerance for residuals without cutting the time step size.	1.0×10^{7}
56	--max-single-precision-days	A real positive value that sets the maximum time step size where single precision floating point arithmetic can be used for solving the linear systems of equations.	20.0
57	--max-temperature-change	A real positive value that stipulates the	5

OPM Flow 2023-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
		maximum absolute change of temperature in a single iteration.	
58	--max-welleq-iter	A positive integer that defines the maximum number of iterations to determine the solution to the well equations.	30
59	--maximum-number-of-well-switches	A positive integer values that stipulates the maximum number of times a well can switch to the same control.	3
60	--milu-variant	A defined character string that specifies which variant of the modified ILU preconditioner ought to be used. Possible variants are: 1) ILU (default, plain ILU), 2) MILU_1 (lump diagonal with dropped row entries), 3) MILU_2 (lump diagonal with the sum of the absolute values of the dropped row entries), 4) MILU_3 (if diagonal is positive add sum of dropped row entries. Otherwise subtract them), 5) MILU_4 (if diagonal is positive add sum of dropped row entries. Otherwise do nothing. The default is "ILU"	"ILU"
61	--min-strict-cnv-ite	A positive integer that sets the minimum number of Newton iterations before relaxed tolerances are used for the CNV convergence criterion.	0
62	--min-time-step-based-on-newton-iterations	The minimum time step size (in days for field and metric unit and hours for lab unit) can be reduced to based on Newton iteration counts.	0
63	--min-time-step-before-shutting-problematic-wells-in-days	A real positive value that sets the minimum time step size in days for which problematic wells are not shut. Time steps below this value will result in problematic wells being shut-in by the simulator.	0.01
	Newton Solver Parameters		
64	--newton-max-iterations	A positive integer that defines the maximum number of Newton iterations per time step used by the simulator.	20
65	--newton-max-relax	A real positive value that sets the maximum relaxation factor of a Newton iteration used by the simulator.	0.5

OPM Flow 2023-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
66	--newton-min-iterations	A real positive integer that sets the minimum number of Newton iterations per time step used by the simulator. The default value of one ensures that at least one Newton iteration is performed after the previous time step.	1
67	--newton-relaxation-type	A character string that defines the type of relaxation used in Newton's method, the default is dampen.	"dampen"
68	--opencl-ilu-parallel	A Boolean value set to true or false that if set to true then parallelize the ILU decomposition and application on GPU, or not (false).	WAHR
69	--opencl-platform-id	A positive integer that specifies the platform identification ("ID") for the openSolver. Use the "clinfo" command to determine valid IDs.	0
70	--output-dir	A character string that defines the directory to which OPM Flow is to write the commercial simulator compatible output files (restart, summary files etc.). The default value results in the files be written to the same directory as the input file.	""
71	--output-extra-convergence-info	Provides additional convergence output to separate files for diagnostic purposes. The available options are: 1) "none" results in no extra output and overrides all other options. 2) "steps" writes out convergence information per time step, to a CASENAME.INFOSTEP file. The file is useful for identifying numerical issues. 3) "iterations" writes out non-linear convergence metrics, i.e., the MB and CNV values, per phase, for each non-linear iteration in each time step, to a CASENAME.*INFOITER* file. Options can be combined with commas, e.g."steps,iterations" for multiple outputs. The default value of "none" prevents the two files from being written out, for better compatibility with the commercial simulator.	none
72	--output-interval	A positive integer that specifies the number of report steps between two consecutive writes of restart data.	1
73	--output-mode	A defined character string that defines the output to the *.PRT and *.DBG files:	"all"

OPM Flow 2023-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**

		1) none: No output to the files.
		2) log or false: Output logging information only.
		3) all or true: Output everything.
		For example to just output logging information use:
		`--output-mode="log"`
		or
		`--output-mode=false`	

74	--owner-cells-first	A Boolean value set to true or false that determines if the cells owned by a rank should be ordered before ghost/overlapping the cells (true), or not (false).	true
75	--parameter-file	A character string that defines the name of a parameter file which contains the simulator's set of run-time parameters, as listed in this table.	""
76	--parsing-strictness	Set strictness of parsing process. Available options are: 1) "normal": stop for critical errors, and for unsupported keywords that would would change the simulator results if supported. 2) "high": stop for all errors, that is even for unsupported keywords that do not effect the results, for example ECHO and NOECHO. 3) "low": same as normal, except do not stop due to unsupported keywords that would change the simulator results if supported, and even if marked critical. Default: "normal"	"normal"
77	--pri-var-oscilation-threshold	A real positive value that defines the threshold value for the primary variable switching conditions after its meaning has switched to hinder oscillations.	1.0 x 10-5
78	--print-parameters	A positive integer value that request that the run time parameters be printed at the start of the run: 0 : No output to the files. 1 : Output *.DBG file. 2 : Output to *.DBG and *.PRT files (default).	2
79	--print-properties	A positive integer value that request that the compile time parameters be printed at the start of the run: 0 : No output to the files.	2

OPM Flow 2023-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
		1 : Output *.DBG file. 2 : Output to *.DBG and *.PRT files (default).	
80	--project-saturations	A Boolean value set to true or false that determines if all the fluid saturations should be scaled to ensure the values are in the interval (0, 1), including runs that use solvents (true), or not (false).	false
81	--regularization-factor-wells	A real positive value that defines the "regularization factor" for wells.	100
82	--relaxed-linear-solver-reduction	A real positive value that defines the minimum reduction of the residual which the linear solver need to achieve for the solution to be accepted.	0.01
83	--relaxed-max-pv-fraction	A real positive value that defines The fraction of the pore volume of the reservoir where the volumetric error (CNV) might be violated during strict Newton iterations.	0.03
84	--relaxed-pressure-tol-msw	A real positive value that sets the relaxation tolerance for the multi-segment well pressure solution in Pascals.	10000
85	--relaxed-well-flow-tol	A real positive value that sets the relaxation tolerance for the well flow residual in reservoir cubic metres (rm3).	0.001
86	--save-step	A character string that determines if the simulator should save the serialized state of the OPM Flow simulator at one or more report steps to a special *.OPMRST file. This is in addition to the normal restart files written, and consumes significantly more space than the normal restart files, but restarting OPM Flow from this file using the *--load-step* option deviates less from the original run, compared to restarting using the normal restart features. The files produced are not compatible with other simulators, and also will not be compatible between different releases of OPM Flow The parameter should be set to one of the following: 1) "all" to save all report steps, 2) ":x" to save every x'th step, or 3) "x" to save a specif time step. The default value of "" does not write anything to the OPM Flow specific restart file.	""

OPM Flow 2023-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
87	--scale-linear-system	A Boolean value set to true or false that switches on (true) or off (false) the scaling of linear system of equations according to the equation scale and primary variable types.	false
88	--sched-restart	A Boolean value that determines for a restart run if the case should initialize wells and groups from the historical SCHEDULE section (true), or from the well and group data on the restart file (false). Note that the commercial simulator always uses data from the restart file (false).	falsch
89	--serial-partitioning	A Boolean value that determines if partitioning for parallel runs on a single process (true), or not (false).	false
90	--shut-unsolvable-wells	A Boolean value that determines if the simulator should shut unsolvable wells (true), or not (false).	wahr
91	--solve-welleq-initially	A Boolean value set to true or false that determines if the simulator should fully solve the well equations before each iteration of the reservoir model (true), or not (false). Note that the well equations are always added to the full system and solved until converged.	true
	Solver Parameters		
92	--solver-continue-on-convergence-failure	A Boolean value that stipulates if the simulator should continue (true) instead of stopping (false) when the minimum solver time step is reached.	false
93	--solver-growth-factor	A real positive value that specifies the growth factor a time step can be increased by when recovering from one or more time step chops, subject to the maximum allowable time step size set by the *--solver-max-time-step-in-days* parameter. For example, if the current time step has converged at 5 days after a time step chop, and --solver-growth-factor is set to the default value of 2.0, then the next time step will be 2.0 * 5 days, that is at 10 days.	2.0
94	--solver-max-growth	A real positive value that specifies the maximum growth factor a time step can be increased by after a report time step. subject to the maximum allowable time step size set by the --solver-max-time-step-in-days parameter. Thus, if the current time step has	3.0

OPM Flow 2023-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
		converged at 5 days after at a report step, and --solver-*max*-growth is set to the default value of 3.0, then the next time step will be 3.0 * 5 days, that is at 15 days.	
95	--solver-max-restarts	A positive integer that defines the number of allowed consecutive restarts (or time step chops) before the simulation is terminated.	10
96	--solver-max-time-step-in-days	A real positive double precision value that specifies the maximum allowed time step size in days.	365
97	--solver-min-time-step	A real positive double precision value that specifies the minimum size of a time step in days for field and metric units, and hours for laboratory units If a time step cannot converge without getting cut below this time step size the simulator will stop.	1.0 x 10-12
98	--solver-restart-factor	A real positive double precision value that sets the time step chop factor of the time step after a convergence failure. For example, if the current non-convergent time step is 30 days and *--solver-restart-factor* is set to the default value, then the time step will be repeated using 0.33 * 30 days as the time step, that is 9.9 days.	0.33
99	--solver-verbosity	A positive integer that specifies the "chattiness" of the non-linear solver.	1
100	--strict-inner-iter-wells	A positive integer that specifies the number of inner well iterations with strict tolerance.	40
101	--strict-outer-iter-wells	A positive integer that specifies the number of newton iterations for which wells are checked with strict tolerance.	6
102	--temperature-max	A real positive value that sets the maximum absolute temperature.	1.0 x 109
103	--temperature-min	A real positive value that sets the minimum absolute temperature.	0
104	--threads-per-process	A positive integer value that stipulates the maximum number of threads to be instantiated per process ('-1' means 'automatic').	-1
	Time Stepping Control Algorithm Parameters		
105	--time-step-after-event-in-days	A real double precision value that sets maximum allowed time step after an event; for example, when a well is open or closed	-1

OPM Flow 2023-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
		etc. The default value of -1 means that events to do effect the time stepping.	
106	--time-step-control	A defined character string that defines the time stepping control algorithm and is set to one of the following: 1) pid: PID controller based adaptive time step control as suggested by Turek and Kuzmin[365]. 2) pid+iteration: Use PID and linear iteration numbers to guide the time step. 3) pid+newtoniteration: Use PID and non-linear iterations numbers to guide the time step. 4) Hardcoded: Use time steps supplied by user via the time-step-control-filename parameter.	"pid+newtoniteration"
107	--time-step-control-decay-damping-factor	A real positive value that specifies the decay rate a time step can be decreased by after the number target iterations has been exceeded	1.0
108	--time-step-control-decay-rate	A real positive value that specifies the decay rate a time step size can be decreased by after the number of target iterations has been exceeded	0.75
109	--time-step-control-file-name	A character string enclosed in quotes that specifies a filename where time steps are specified. The default is the character string timesteps For instance time steps can be generated by the ecl_summary application in libecl as per the following UNIX command line: `path_to_libecl_applications/ ecl_summary DECK TIME > filename` Where: DECK is the name of the data deck you want to get the time steps from, TIME tells the application to return the timing for the run, and "filename" is the name of the file the times are piped to.	"timesteps"
110	--time-step-control-growth-damping-factor	A real positive value that specifies the growth rate of the time step increase when the number of target iterations are	3.2

[365] *Algebraic Flux Correction III. Incompressible Flow Problems. Uni Dortmund, Turek and Kuzmin, January, 2006 (DOI: 10.1007/3-540-27206-2_8)*

OPM Flow 2023-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
		undercut.	
111	--time-step-control-growth-rate	A real positive value that specifies the growth rate of the time step size when the number of target iterations are undercut.	1.25
112	--time-step-control-target-iterations	A positive integer that defines the number of linear iterations which the time step control scheme should aim for (if applicable).	30
113	--time-step-control-target-newton-iterations	A positive integer that specifies the number of Newton iterations which the time step control scheme should aim for (if applicable).	8
114	--time-step-control-tolerance	A real double precision value that sets the tolerance for PID (only used with the pid and pid+ options defined by the --time-step-control option).	0.1
115	--time-step-verbosity	A positive integer that specifies the "chattiness" during the time integration.	1
	Convergence and Material Balance Tolerance Parameters		
116	--tolerance-cnv	A real positive double precision value that specifies the maximum non-linear tolerance error. This is the local convergence tolerance (maximum of local saturation errors).	0.01
117	--tolerance-cnv-relaxed	A real positive value that defines the relaxed local convergence tolerance that applies for iterations after the iterations with the strict tolerance.	1.0
118	--tolerance-mb	A real positive double precision value that sets the maximum mass balance error, that is the tolerated mass balance error relative to total mass present.	1.0 x 10-6
119	--tolerance-pressure-ms-wells	A real positive double precision value that specifies the tolerance for the pressure equations for multi-segment wells.	1000
120	--tolerance-well-control	A real positive double precision value that sets the maximum tolerance for the well control equations.	1.0×10^{-7}
121	--tolerance-wells	A real positive double precision value that defines the maximum non-linear error for the well equations.	1.0×10^{-4}
122	-update-equations-scaling	A Boolean value that switches on (true) or off (false) the updating of the scaling factors for mass balance equations during the simulation.	false

OPM Flow 2023-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
123	--use-gmres	A Boolean value that when set to true OPM Flow will use Generalized Minimal Residual ("GMRES")[366] and [367] solver instead of Biconjugate Gradient Stabilized ("BiCGSTAB")[368] and [369] as the linear solver within the Newton iterations.	false
124	--use-multisegment-well	A Boolean value that when set to true the simulator will use the well model for multi-segment wells instead of the one for single-segment wells.	true
125	--use-update-stabilization	A Boolean value set to true or false that switches on (true) or off (false) the stabilized Newton option, that attempts to detect and correct oscillations or stagnation during the Newton iterations. This option may improve convergence for some cases.	true
126	--zoltan-imbalance-tol	A real positive that defines the tolerable imbalance of the load balancing provided by Zoltan package.	1.1

Notes:

1) Cells colored green in the No. column are new command line parameters for this release. Similarly for the Default column, cells colored green indicate the default value has changed from the previous release.
2) Items shaded in gray are considered to be developer options that should be used with caution as the values associated with these command line parameters are subject to change.
3) VTK Graphics Command Line Parameters[370] set of commands are no longer listed via the basic help command line command:

```
flow --help
```

instead use:

```
flow --help-all
```

to get a list of supported command line Parameters.

4) The –enable-vtk-output option above, if set to "true" will write out the data as stipulated by the compile options. One may need to compile the source code to obtain the desired output.
5) As per all UNIX and LINUX based system the input is case dependent.
6) If a parameter file is used to set one of the above parameters, the leading "--" should not be incorporated in

[366] *Y. Saad, A flexible inner-outer preconditioned GMRES algorithm, SIAM J. Sci. Statist. Comput., 14, (1993).*

[367] *Y. Saad and M.H. Schultz, GMRES: A generalized minimum residual algorithm for solving nonsymmetric linear systems, SIAM J. Sci. Statist. Comput., 7 (1986), pp. 856{869.*

[368] *Van der Vorst, H. A. (1992). "Bi-CGSTAB: A Fast and Smoothly Converging Variant of Bi-CG for the Solution of Nonsymmetric Linear Systems". SIAM J. Sci. Stat. Comput. 13 (2): 631–644. doi:10.1137/0913035. hdl:10338.dmlcz/104566*

[369] *Sleijpen, G. L. G.; Fokkema, D. R. (November 1993). "BiCGstab(l) for linear equations involving unsymmetric matrices with complex spectrum" (PDF). Electronic Transactions on Numerical Analysis. Kent, OH: Kent State University. 1: 11–32. ISSN 1068-9613.*

[370] *Virtualization ToolKit ("VTK") files used to visualize the 3D results from the model using ParaView, an open-source, multi-platform general purpose data analysis and visualization application (https://www.paraview.org/overview/). For the commercial simulator's compatible output files OPM's ResInsight 3D visualization package can be used instead and is specifically tailored for OPM Flow and the commercial simulator.*

OPM Flow 2023-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
	the parameter file.		

Table E.1: OPM Flow 2023-04 Command Line Options

E.2 RUNNING OPM FLOW 2022-10

The command line syntax for running OPM Flow is:

```
flow [OPTIONS] CASENAME
```

and typing the following command from your terminal:

```
flow CASENAME
```

will start the simulator and run the case specified by CASENAME.DATA.

OPM Flow accepts command line options to control various aspects of the simulator's run time behavior, as described in the table below. To give a command line option use "key=value" syntax, with no spaces around the equals sign. It is also possible to put multiple options together in a parameter file. To do so, put one option on each line of the file with ".param" as the extension and pass that filename as a command line parameter to OPM Flow using:

```
flow --parameter-file=CASENAME.PARAM
```

to instruct OPM Flow to read the parameter file.

OPM Flow 2022-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
1	-h or --help	A character string that causes OPM Flow to print a help message that gives a brief description of the available command line parameters. **This now only prints the active user facing command line options.**	N/A
2	--help-all	Prints all the command line options included in the release, including obsolete, hidden and deprecated options.	N/A
3	--accelerator-mode	A defined character string that defines the usage of the GPU (cusparseSolver or openclSolver) or FPGA (fpgaSolver) as the linear solver, usage '--accelerator-mode=[none|cusparse|opencl|fpga|amgcl]'.	"none"
4	--allow-distributed-wells	A boolean value set to true of false that allows the perforations of a well to be distributed to interior of multiple processes in a parallel run (true), or not (false).	false
5	--alternative-well-rate-init	A boolean value set to true of false that causes the simulator to use an alternative well rate initialization procedure (true).	true
6	--bda-device-id	Choose device ID for cusparseSolver or openclSolver, use 'nvidia-smi' or 'clinfo' commands to determine valid IDs.	0

OPM Flow 2022-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
7	--continue-on-convergence-error	A Boolean value set to true or false that that instructs the simulator to continue (true) with a non-converged solution instead of giving up (false) if it encounter a time step size smaller than the minimum time step size.	false
8	--cpr-max-ell-iter	A positive integer that sets the maximum number of Iterations for the elliptic pressure part of the Constrained Pressure Residual ("CPR") solver.	20
9	--cpr-reuse-interval	A positive integer that sets the reuse preconditioner interval. Used when --cpr-reuse-setup is set to 4, then the preconditioner will be fully recreated instead of reused every N linear solve, where N is this parameter.	10
10	--cpr-reuse-setup	A positive integer that defines if the CPR solver should re-use the Amg setup. Valid options are: 0 : Recreate the preconditioner for every linear solve. 1 : Recreate once every time step. 2 : Recreate if last linear solve took more than 10 iterations. 3 : Never recreate.	3
11	--dbph-max-rel	A real positive value that sets maximum relative change of the bottom-hole pressure in a single iteration	1.0
12	--dp-max-rel	A real positive double precision value that sets the maximum allowed relative change of pressure per iteration.	0.3
13	--ds-max	A real positive double precision value that sets the maximum allowed change in saturation per iteration.	0.2
14	--dwell-fraction-max	A real positive value that sets the maximum allowed change in well's volume fraction per iteration.	0.2
15	--ecl-deck-file-name	A character string that defines the name of the OPM Flow input file which contains the simulator's ECLIPSE formatted input deck to be simulated. Note that there must be no spaces in the filename. If there are spaces, which can typically occur on Windows based systems, then the filename should be enclosed in quotes; thus instead of FILE NAME use "FILE NAME".	""

OPM Flow 2022-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
16	--ecl-enable-drift-compensation	A Boolean value set to true or false that enables (true) or disables (false) partial compensation of systematic mass losses via the source term of the next time step.	true
17	--ecl-output-double-precision	A Boolean value set to true or false that switches on (true) or off (false) double precision in restart files. Useful for 'perfect' restarts.	false
18	--ecl-output-interval	An integer value defining the number of report steps that ought to be skipped between two writes of restart files results.	-1
19	--ecl-strict-parsing	A Boolean value set to true or false that switches on (true) or off (false) strict parsing mode for parsing - all errors are collected before the application exists. Useful for debugging initial runs.	false
20	--edge-weights-method	A defined positive integer value that defines the edge-weighing strategy: 0 : For uniform, 1 : for trans, or 2 : for log(trans).	1
21	--enable-adaptive-time-stepping	A Boolean value set to true or false that turns on (true) or off (false) adaptive time stepping. If set to false the report time steps provided by the input deck are used.	true
22	--enable-async-ecl-output	A Boolean value set to true or false that sets the output mode to be asynchronously (true), letting the simulator continue computing the next time step while writing results to restart and summary files in the commercial simulators format.	true
23	--enable-async-vtk-output	A Boolean value set to true or false that sets the output mode to be asynchronously (true), letting the simulator continue computing the next time step while writing results to the VTK output files.	true
24	--enable-dry-run	A Boolean value set to true or false that specifies if the simulation should actually run, (true) or just check the input deck (false). This option is equivalent to activating the NOSIM keyword in the RUNSPEC section of the input deck (see section *5.3.97 NOSIM – Activate the No Simulation Mode for Data File Checking*). The default value of "auto' will use whatever is stipulated in the input deck via the NOSIM keyword.	"auto"

OPM Flow 2022-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
25	--enable-ecl-output	A Boolean value set to true or false that specifies if the binary output files (restart, summary files, etc.) should be written in the commercial simulator's format (true), or OPM Flow's format (false).	true
26	--enable-esmry	A Boolean value that switches on (true) or off (false) the output SUMMARY vectors to the ESMRY file for fast loading of summary data.	false
27	--enable-logging-fallout-warning	A Boolean value set to true or false that sets the developer option to see whether logging was on non-root processors (true). If set to true output will be appended to the *.DBG or *.PRT files.	false
28	--enable-opm-rst-file	A Boolean value set to true or false to write OPM specific data sets to the commercial simulators restart file to enable restart of an OPM Flow runs (true) by OPM Flow, or not to write the data (false).	false
29	--enable-storage-cache	A Boolean value set to true or false that turns on (true) or off (false) storing previous storage terms and avoid re-calculating them.	true
30	--enable-terminal-output	A Boolean value set to true or false that turns on (true) or off (false) high-level information about the simulation's progress to the terminal	true
31	--enable-tuning	A Boolean value set to true or false that instructs OPM Flow to read the time stepping parameters from the *TUNING - Numerical Tuning Control* keyword in the input deck, if set to true. Note that only the first record of the TUNING keyword is processed.	false
32	--enable-vtk-output	A Boolean value set to true or false that turns on (true) or off (false) a global switch for writing VTK files.	false
33	--enable-well-operability-check	A Boolean value set to true or false that enables (true) checking of a well's operating status, or disables (false) the checking.	true
34	--enable-well-operability-check-iter	A Boolean value set to true or false that enables (true) checking of a well's operating status during iterations, or disables (false) the checking during iterations.	false

OPM Flow 2022-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
35	--enable-write-all-solutions	A Boolean value set to true or false that turns on (true) or off (false) the writing of all solutions to disk instead of only the ones for the report steps.	false
36	--force-disable-fluid-in-place-output	A Boolean value set to true or false that instructs OPM Flow not to print the Fluid In-Place report after each report time step (true) or not (false). Note this parameter will override the print request in the input deck.	false
37	--force-disable-resv-fluid-in-place-output	A Boolean value set to true or false that instructs OPM Flow to not to print the Reservoir Volume Fluid In-Place report after each report time step (true) or not (false). Note this parameter will override the print request in the input deck.	false
38	--fpga-bitstream	A character string that specifies the bit stream file for the fpgaSolver (including path), usage: '--fpga-bitstream=<filename>'.	“”
39	--full-time-step-initially	A Boolean value set to true or false that instructs OPM Flow to always attempt to finish a report step using a single time step (true) or not (false).	false
40	--ignore-keywords	A character string that defines a list of keywords which should be ignored by OPM Flow. The keyword string should be enclosed in quotes and each keyword separated by a colon, that is ':' .	“”
	ILU Preconditioner Parameters		
41	--ilu-fillin-level	A positive integer value that sets the fill in level for the ILU preconditioner.	0
42	--ilu-redblack	A Boolean value set to true or false that instructs OPM Flow to use red-black partitioning for the ILU preconditioner. (true) or not (false).	false
43	--ilu-relaxation	A real positive double precision value that sets the relaxation factor of the linear solver's ILU preconditioner	0.9

OPM Flow 2022-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
44	--ilu-reorder-spheres	A Boolean value set to true or false that specifies OPM Flow to reorder the entries of the matrix in the red-black ILU preconditioner in spheres starting at an edge (true) or not (false) If false the original ordering is preserved in each color. Otherwise try to ensure D4 ordering (in a 2D structured grid, the diagonal elements are consecutive).	false
45	--initial-time-step-in-days	A real double precision value that sets the size of initial time step in days.	1.0
	Linear Solver Parameters		
46	--linear-solver	"ilu0"A defined quoted character string that sets the configuration of the solver, valid values are: 1) ilu0 (default), 2) cpr (an alias for cpr_trueimpes)[371] [372] and [373] 3) cpr_quasiimpes, 4) cpr_trueimpes, 5) cprw, 6) amg,[374] or 7) a file name that has the extension ".json", that contains the linear solver configuration parameters. Option (5) extends the existing Constrained Pressure Residual ("CPR") preconditioner to include wells. For option (6) one enters a character string enclosed in quotes that defines the filename of a JSON configuration file for the flexible linear solver system. In this case the file extension should be '.json'. Note that the *.PRT file contains the "Property tree for the linear solver" listing, which is the JSON specification of the current case, and can be used as a guide to configure a user specific linear solver JSON file. The default is "ilu0".	"ilu0"

[371] *Wallis, J. R., Little, T. E., and Nolen, J. S.: "Constrained Residual Acceleration of Conjugate Residual Methods," paper SPE 13536 presented at the SPE Reservoir Simulation Symposium, Dallas, Texas, USA (February 10-13, 1985).*

[372] *R. Scheichl, M. Roland, J. Wendebourg, Decoupling and block preconditioning for sedimentary basin simulations, Computational Geosciences 7 (2003) 295{318.*

[373] *Klemetsdal, Ø.S., Møyner, O. & Lie, KA. Accelerating multiscale simulation of complex geomodels by use of dynamically adapted basis functions. Comput Geosci 24, 459–476 (2020). https://doi.org/10.1007/s10596-019-9827-z.*

[374] *M. Blatt, A parallel algebraic multigrid method for elliptic problems with highly discontinuous coefficients, Ph.D. thesis, Ruprecht-Karls-Universität Heidelberg (2010).*

OPM Flow 2022-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
47	--linear-solver-ignore-convergence-failure	A Boolean value set to true or false that if set to true convergences failures in the linear solver are ignored. This option should be used with care, as the results may be unreliable.	false
48	--linear-solver-max-iter	A positive integer value that defines the maximum number of linear iterations.	200
49	--linear-solver-reduction	A real positive double precision value that sets the minimum reduction of the residual for the linear solver. The linear solver convergences when the residual is reduced sufficiently.	0.01
50	--linear-solver-restart	A positive integer value that sets the number of iterations after which GMRES is restarted.	40
51	--linear-solver-verbosity	A positive integer value that defines the output from linear solver: 0 : No extra output, 1 : output per solution iteration, or 2 : output per iteration.	0
52	--matrix-add-well-contributions	A Boolean value set to true or false that if set to true explicitly specifies the influences of wells between cells in the Jacobian and preconditioner matrices.	false
53	--max-inner-iter-ms-wells	A positive integer value that defines the maximum number of inner iterations for multi-segment wells.	100
54	--max-inner-iter-wells	A positive integer value that defines the maximum number of inner iterations for standard wells.	50
55	--max-newton-iterations-with-inner-well-iterations	A positive integer that specifies the maximum newton iterations with inner well iterations.	8
56	--max-pressure-change-ms-wells	A real positive value that defines the maximum relative pressure change for a single iteration of the multi-segment well model.	1.0×10^{6}
57	--max-residual-allowed	A real positive value that sets the absolute maximum tolerance for residuals without cutting the time step size.	1.0×10^{7}
58	--max-single-precision-days	A real positive value that sets the maximum time step size where single precision floating point arithmetic can be used for solving the linear systems of equations.	20.0

OPM Flow 2022-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
59	--max-temperature-change	A real positive value that stipulates the maximum absolute change of temperature in a single iteration.	5
60	--max-welleq-iter	A positive integer that defines the maximum number of iterations to determine the solution to the well equations.	30
61	--maximum-number-of-well-switches	A positive integer values that stipulates the maximum number of times a well can switch to the same control.	3
62	--milu-variant	A defined character string that specifies which variant of the modified ILU preconditioner ought to be used. Possible variants are: 1) ILU (default, plain ILU), 2) MILU_1 (lump diagonal with dropped row entries), 3) MILU_2 (lump diagonal with the sum of the absolute values of the dropped row entries), 4) MILU_3 (if diagonal is positive add sum of dropped row entries. Otherwise subtract them), 5) MILU_4 (if diagonal is positive add sum of dropped row entries. Otherwise do nothing. The default is "ILU"	"ILU"
63	--min-strict-cnv-ite	A positive integer that sets the minimum number of Newton iterations before relaxed tolerances are used for the CNV convergence criterion.	0
64	--min-time-step-based-on-newton-iterations	The minimum time step size (in days for field and metric unit and hours for lab unit) can be reduced to based on Newton iteration counts.	0
65	--min-time-step-before-shutting-problematic-wells-in-days	A real positive value that sets the minimum time step size in days for which problematic wells are not shut. Time steps below this value will result in problematic wells being shut-in by the simulator.	0.01
	Newton Solver Parameters		
66	--newton-max-iterations	A positive integer that defines the maximum number of Newton iterations per time step used by the simulator.	20
67	--newton-max-relax	A real positive value that sets the maximum relaxation factor of a Newton iteration used by the simulator.	0.5

OPM Flow 2022-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
68	--newton-min-iterations	A real positive integer that sets the minimum number of Newton iterations per time step used by the simulator. The default value of one ensures that at least one Newton iteration is performed after the previous time step.	1
69	--newton-relaxation-type	A character string that defines the type of relaxation used in Newton's method, the default is dampen.	"dampen"
70	--opencl-ilu-reorder	A defined character string that selects the reordering strategy for ILU for openclSolver and fpgaSolve, and should be set to: 1) "level_scheduling", or 2) "graph_coloring" Note that "level_scheduling" behaves like Dune and cusparse; whereas, "graph_coloring" is more aggressive and is likely to be faster, but is random-based and generally increases the number of linear solves and linear iterations significantly.	""
71	--opencl-platform-id	A positive integer that specifies the platform identification ("ID") for the openSolver. Use the "clinfo" command to determine valid IDs.	0
72	--output-dir	A character string that defines the directory to which OPM Flow is to write the commercial simulator compatible output files (restart, summary files etc.). The default value results in the files be written to the same directory as the input file.	""
73	--output-interval	A positive integer that specifies the number of report steps between two consecutive writes of restart data.	1

74	--output-mode	A defined character string that defines the output to the *.PRT and *.DBG files:
		1) none: No output to the files.
		2) log or false: Output logging information only.
		3) all or true: Output everything.
		For example to just output logging information use:
		`--output-mode="log"`
		or
		`--output-mode=false`
			"all"

OPM Flow 2022-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
75	--owner-cells-first	A Boolean value set to true or false that determines if the cells owned by a rank should be ordered before ghost/overlapping the cells (true), or not (false).	true
76	--parameter-file	A character string that defines the name of a parameter file which contains the simulator's set of run-time parameters, as listed in this table.	""
77	--pri-var-oscilation-threshold	A real positive value that defines the threshold value for the primary variable switching conditions after its meaning has switched to hinder oscillations.	1.0 x 10-5
78	--print-parameters	A positive integer value that request that the run time parameters be printed at the start of the run: 0 : No output to the files. 1 : Output *.DBG file. 2 : Output to *.DBG and *.PRT files (default).	2
79	--print-properties	A positive integer value that request that the compile time parameters be printed at the start of the run: 0 : No output to the files. 1 : Output *.DBG file. 2 : Output to *.DBG and *.PRT files (default).	2
80	--project-saturations	A Boolean value set to true or false that determines if all the fluid saturations should be scaled to ensure the values are in the interval (0, 1), including runs that use solvents (true), or not (false).	false
81	--regularization-factor	A real positive value that defines the "regularization factor" for wells.	100
82	--relaxed-max-pv-fraction	A real positive value that defines The fraction of the pore volume of the reservoir where the volumetric error (CNV) may be violated during strict Newton iterations.	0.03
83	--relaxed-pressure-tol-msw	A real positive value that sets the relaxation tolerance for the multi-segment well pressure solution in Pascals.	10000
84	--relaxed-well-flow-tol	A real positive value that sets the relaxation tolerance for the well flow residual in reservoir cubic metres (rm3).	0.001

OPM Flow 2022-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
85	--scale-linear-system	A Boolean value set to true or false that switches on (true) or off (false) the scaling of linear system of equations according to the equation scale and primary variable types.	false
86	--sched-restart	A Boolean value that determines for a restart run if the case should initialize wells and groups from the historical SCHEDULE section (true), or from the well and group data on the restart file (false). Note that the commercial simulator always uses data from the restart file (false).	falsch
87	--serial-partitioning	A Boolean value that determines if partitioning for parallel runs on a single process (true), or not (false).	false
88	--shut-unsolvable-wells	A Boolean value that determines if the simulator should shut unsolvable wells (true), or not (false).	wahr
89	--solve-welleq-initially	A Boolean value set to true or false that determines if the simulator should fully solve the well equations before each iteration of the reservoir model (true), or not (false). Note that the well equations are always added to the full system and solved until converged.	true
	Solver Parameters		
90	--solver-continue-on-convergence-failure	A Boolean value that stipulates if the simulator should continue (true) instead of stopping (false) when the minimum solver time step is reached.	false
91	--solver-growth-factor	A real positive value that specifies the growth factor a time step can be increased by when recovering from one or more time step chops, subject to the maximum allowable time step size set by the *--solver-max-time-step-in-days* parameter. For example, if the current time step has converged at 5 days after a time step chop, and --solver-growth-factor is set to the default value of 2.0, then the next time step will be 2.0 * 5 days, that is at 10 days.	2.0

OPM Flow 2022-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
92	--solver-max-growth	A real positive value that specifies the maximum growth factor a time step can be increased by after a report time step. subject to the maximum allowable time step size set by the --solver-max-time-step-in-days parameter. Thus, if the current time step has converged at 5 days after at a report step, and --solver-*max*-growth is set to the default value of 3.0, then the next time step will be 3.0 * 5 days, that is at 15 days.	3.0
93	--solver-max-restarts	A positive integer that defines the number of allowed consecutive restarts (or time step chops) before the simulation is terminated.	10
94	--solver-max-time-step-in-days	A real positive double precision value that specifies the maximum allowed time step size in days.	365
95	--solver-min-time-step	A real positive double precision value that specifies the minimum size of a time step in days for field and metric units, and hours for laboratory units If a time step cannot converge without getting cut below this time step size the simulator will stop.	1.0 x 10-12
96	--solver-restart-factor	A real positive double precision value that sets the time step chop factor of the time step after a convergence failure. For example, if the current non-convergent time step is 30 days and *--solver-restart-factor* is set to the default value, then the time step will be repeated using 0.33 * 30 days as the time step, that is 9.9 days.	0.33
97	--solver-verbosity	A positive integer that specifies the "chattiness" of the non-linear solver.	1
98	--strict-inner-iter-wells	A positive integer that specifies the number of inner well iterations with strict tolerance.	40
99	--strict-outer-iter-wells	A positive integer that specifies the number of newton iterations for which wells are checked with strict tolerance.	6
100	--temperature-max	A real positive value that sets the maximum absolute temperature.	1.0 x 109
101	--temperature-min	A real positive value that sets the minimum absolute temperature.	0

OPM Flow 2022-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
102	--threads-per-process	A positive integer value that stipulates the maximum number of threads to be instantiated per process ('-1' means 'automatic').	-1
	Time Stepping Control Algorithm Parameters		
103	--time-step-after-event-in-days	A real double precision value that sets maximum allowed time step after an event; for example, when a well is open or closed etc. The default value of -1 means that events to do effect the time stepping.	-1
104	--time-step-control	A defined character string that defines the time stepping control algorithm and is set to one of the following: 1) pid: PID controller based adaptive time step control as suggested by Turek and Kuzmin[375]. 2) pid+iteration: Use PID and linear iteration numbers to guide the time step. 3) pid+newtoniteration: Use PID and non-linear iterations numbers to guide the time step. 4) Hardcoded: Use time steps supplied by user via the time-step-control-filename parameter.	"pid+newtoniteration"
105	--time-step-control-decay-damping-factor	A real positive value that specifies the decay rate a time step can be decreased by after the number target iterations has been exceeded	1.0
106	--time-step-control-decay-rate	A real positive value that specifies the decay rate a time step size can be decreased by after the number of target iterations has been exceeded	0.75

[375] *Algebraic Flux Correction III. Incompressible Flow Problems. Uni Dortmund, Turek and Kuzmin, January, 2006 (DOI: 10.1007/3-540-27206-2_8)*

OPM Flow 2022-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
107	--time-step-control-file-name	A character string enclosed in quotes that specifies a filename where time steps are specified. The default is the character string timesteps For instance time steps can be generated by the ecl_summary application in libecl as per the following UNIX command line: `path_to_libecl_applications/ ecl_summary DECK TIME > filename` Where: DECK is the name of the data deck you want to get the time steps from, TIME tells the application to return the timing for the run, and "filename" is the name of the file the times are piped to.	"timesteps"
108	--time-step-control-growth-damping-factor	A real positive value that specifies the growth rate of the time step increase when the number of target iterations are undercut.	3.2
109	--time-step-control-growth-rate	A real positive value that specifies the growth rate of the time step size when the number of target iterations are undercut.	1.25
110	--time-step-control-target-iterations	A positive integer that defines the number of linear iterations which the time step control scheme should aim for (if applicable).	30
111	--time-step-control-target-newton-iterations	A positive integer that specifies the number of Newton iterations which the time step control scheme should aim for (if applicable).	8
112	--time-step-control-tolerance	A real double precision value that sets the tolerance for PID (only used with the pid and pid+ options defined by the --time-step-control option).	0.1
113	--time-step-verbosity	A positive integer that specifies the "chattiness" during the time integration.	1
	Convergence and Material Balance Tolerance Parameters		
114	--tolerance-cnv	A real positive double precision value that specifies the maximum non-linear tolerance error. This is the local convergence tolerance (maximum of local saturation errors).	0.01

OPM Flow 2022-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
115	--tolerance-cnv-relaxed	A real positive value that defines the relaxed local convergence tolerance that applies for iterations after the iterations with the strict tolerance.	1.0
116	--tolerance-mb	A real positive double precision value that sets the maximum mass balance error, that is the tolerated mass balance error relative to total mass present.	1.0 x 10-6
117	--tolerance-pressure-ms-wells	A real positive double precision value that specifies the tolerance for the pressure equations for multi-segment wells.	1000
118	--tolerance-well-control	A real positive double precision value that sets the maximum tolerance for the well control equations.	1.0×10^{-7}
119	--tolerance-wells	A real positive double precision value that defines the maximum non-linear error for the well equations.	1.0×10^{-4}
120	-update-equations-scaling	A Boolean value that switches on (true) or off (false) the updating of the scaling factors for mass balance equations during the simulation.	false
121	--use-gmres	A Boolean value that when set to true OPM Flow will use Generalized Minimal Residual ("GMRES")[376] and [377] solver instead of Biconjugate Gradient Stabilized ("BiCGSTAB")[378] and [379] as the linear solver within the Newton iterations.	false
122	--use-multisegment-well	A Boolean value that when set to true the simulator will use the well model for multi-segment wells instead of the one for single-segment wells.	true
123	--use-update-stabilization	A Boolean value set to true or false that switches on (true) or off (false) the stabilized Newton option, that attempts to detect and correct oscillations or stagnation during the Newton iterations. This option may improve convergence for some cases.	true

[376] *Y. Saad, A flexible inner-outer preconditioned GMRES algorithm, SIAM J. Sci. Statist. Comput., 14, (1993).*

[377] *Y. Saad and M.H. Schultz, GMRES: A generalized minimum residual algorithm for solving nonsymmetric linear systems, SIAM J. Sci. Statist. Comput., 7 (1986), pp. 856{869.*

[378] *Van der Vorst, H. A. (1992). "Bi-CGSTAB: A Fast and Smoothly Converging Variant of Bi-CG for the Solution of Nonsymmetric Linear Systems". SIAM J. Sci. Stat. Comput. 13 (2): 631–644. doi:10.1137/0913035. hdl:10338.dmlcz/104566*

[379] *Sleijpen, G. L. G.; Fokkema, D. R. (November 1993). "BiCGstab(l) for linear equations involving unsymmetric matrices with complex spectrum" (PDF). Electronic Transactions on Numerical Analysis. Kent, OH: Kent State University. 1: 11–32. ISSN 1068-9613.*

OPM Flow 2022-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
124	--zoltan-imbalance-tol	A real positive that defines the tolerable imbalance of the load balancing provided by Zoltan package.	1.1

Notes:

1) Cells colored green in the No. column are new command line parameters for this release. Similarly for the Default column, cells colored green indicate the default value has changed from the previous release.
2) Items shaded in gray are considered to be developer options that should be used with caution as the values associated with these command line parameters are subject to change.
3) VTK Graphics Command Line Parameters[380] set of commands are no longer listed via the basic help command line command:

   ```
   flow --help
   ```

   instead use:

   ```
   flow --help-all
   ```

   to get a list of supported command line Parameters. The –enable-vtk-output option above, if set to "true" will write out the data as stipulated by the compile options. One may need to compile the source code to obtain the desired output.
4) As per all UNIX and LINUX based system the input is case dependent.
5) If a parameter file is used to set one of the above parameters, the leading "--" should not be incorporated in the parameter file.

Table E.2: OPM Flow 2022-10 Command Line Options

[380] *Virtualization ToolKit ("VTK") files used to visualize the 3D results from the model using ParaView, an open-source, multi-platform general purpose data analysis and visualization application (https://www.paraview.org/overview/). For the commercial simulator's compatible output files OPM's ResInsight 3D visualization package can be used instead and is specifically tailored for OPM Flow and the commercial simulator.*

E.3 RUNNING OPM FLOW 2022-04

The command line syntax for running OPM Flow is:

```
flow [OPTIONS] CASENAME
```

and typing the following command from your terminal:

```
flow CASENAME
```

will start the simulator and run the case specified by CASENAME.DATA.

OPM Flow accepts command line options to control various aspects of the simulator's run time behavior, as described in the table below. To give a command line option use "key=value" syntax, with no spaces around the equals sign. It is also possible to put multiple options together in a parameter file. To do so, put one option on each line of the file with ".param" as the extension and pass that filename as a command line parameter to OPM Flow using:

```
flow --parameter-file=CASENAME.PARAM
```

to instruct OPM Flow to read the parameter file.

OPM Flow 2022-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
	General eWoms/ebos Command Line Parameters		
1	-h or --help	A character string that causes OPM Flow to print a help message that gives a brief description of the available command line parameters. **This now only prints the active user facing command line options.**	N/A
2	--help-all	Prints all the command line options included in the release, including obsolete, hidden and deprecated options.	N/A
3	--accelerator-mode	A defined character string that defines the usage of the GPU (cusparseSolver or openclSolver) or FPGA (fpgaSolver) as the linear solver, usage '--accelerator-mode=[none|cusparse|opencl|fpga|amgcl]'.	"none"
4	--allow-distributed-wells	A boolean value set to true of false that allows the perforations of a well to be distributed to interior of multiple processes in a parallel run (true), or not (false).	false
5	--alternative-well-rate-init	A boolean value set to true of false that causes the simulator to use an alternative well rate initialization procedure (true).	true
6	--bda-device-id	Choose device ID for cusparseSolver or openclSolver, use 'nvidia-smi' or 'clinfo' commands to determine valid IDs.	0

OPM Flow 2022-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
7	--continue-on-convergence-error	A Boolean value set to true or false that that instructs the simulator to continue (true) with a non-converged solution instead of giving up (false) if it encounter a time step size smaller than the minimum time step size.	false
8	--cpr-max-ell-iter	A positive integer that sets the maximum number of Iterations for the elliptic pressure part of the Constrained Pressure Residual ("CPR") solver.	20
9	--cpr-reuse-setup	A positive integer that defines if the CPR solver should re-use the Amg setup. Valid options are: 0 : Recreate the preconditioner for every linear solve. 1 : Recreate once every time step. 2 : Recreate if last linear solve took more than 10 iterations. 3 : Never recreate.	3
10	--dbph-max-rel	A real positive value that sets maximum relative change of the bottom-hole pressure in a single iteration	1.0
11	--dp-max-rel	A real positive double precision value that sets the maximum allowed relative change of pressure per iteration.	0.3
12	--ds-max	A real positive double precision value that sets the maximum allowed change in saturation per iteration.	0.2
13	--dwell-fraction-max	A real positive value that sets the maximum allowed change in well's volume fraction per iteration.	0.2
14	--ecl-deck-file-name	A character string that defines the name of the OPM Flow input file which contains the simulator's ECLIPSE formatted input deck to be simulated. Note that there must be no spaces in the filename. If there are spaces, which can typically occur on Windows based systems, then the filename should be enclosed in quotes; thus instead of FILE NAME use "FILE NAME".	""
15	--ecl-enable-drift-compensation	A Boolean value set to true or false that enables (true) or disables (false) partial compensation of systematic mass losses via the source term of the next time step.	true

OPM Flow 2022-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
16	--ecl-output-double-precision	A Boolean value set to true or false that switches on (true) or off (false) double precision in restart files. Useful for 'perfect' restarts.	false
17	--ecl-output-interval	An integer value defining the number of report steps that ought to be skipped between two writes of restart files results.	-1
18	--ecl-strict-parsing	A Boolean value set to true or false that switches on (true) or off (false) strict parsing mode for parsing - all errors are collected before the application exists. Useful for debugging initial runs.	false
19	--edge-weights-method	A defined positive integer value that defines the edge-weighing strategy: 0 : For uniform, 1 : for trans, or 2 : for log(trans).	1
20	--enable-adaptive-time-stepping	A Boolean value set to true or false that turns on (true) or off (false) adaptive time stepping. If set to false the report time steps provided by the input deck are used.	true
21	--enable-async-ecl-output	A Boolean value set to true or false that sets the output mode to be asynchronously (true), letting the simulator continue computing the next time step while writing results to restart and summary files in the commercial simulators format.	true
22	--enable-async-vtk-output	A Boolean value set to true or false that sets the output mode to be asynchronously (true), letting the simulator continue computing the next time step while writing results to the VTK output files.	true
23	--enable-dry-run	A Boolean value set to true or false that specifies if the simulation should actually run, (true) or just check the input deck (false). This option is equivalent to activating the NOSIM keyword in the RUNSPEC section of the input deck (see section *5.3.97 NOSIM – Activate the No Simulation Mode for Data File Checking*). The default value of "auto' will use whatever is stipulated in the input deck via the NOSIM keyword.	"auto"

OPM Flow 2022-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
24	--enable-ecl-output	A Boolean value set to true or false that specifies if the binary output files (restart, summary files, etc.) should be written in the commercial simulator's format (true), or OPM Flow's format (false).	true
25	--enable-esmry	A Boolean value that switches on (true) or off (false) the output SUMMARY vectors to the ESMRY file for fast loading of summary data.	false
26	--enable-logging-fallout-warning	A Boolean value set to true or false that sets the developer option to see whether logging was on non-root processors (true). If set to true output will be appended to the *.DBG or *.PRT files.	false
27	--enable-opm-rst-file	A Boolean value set to true or false to write OPM specific data sets to the commercial simulators restart file to enable restart of an OPM Flow runs (true) by OPM Flow, or not to write the data (false).	false
28	--enable-storage-cache	A Boolean value set to true or false that turns on (true) or off (false) storing previous storage terms and avoid re-calculating them.	true
29	--enable-terminal-output	A Boolean value set to true or false that turns on (true) or off (false) high-level information about the simulation's progress to the terminal	true
30	--enable-tuning	A Boolean value set to true or false that instructs OPM Flow to read the time stepping parameters from the *TUNING - Numerical Tuning Control* keyword in the input deck, if set to true. Note that only the first record of the TUNING keyword is processed.	false
31	--enable-vtk-output	A Boolean value set to true or false that turns on (true) or off (false) a global switch for writing VTK files.	false
32	--enable-well-operability-check	A Boolean value set to true or false that enables (true) checking of a well's operating status, or disables (false) the checking.	true
33	--enable-well-operability-check-iter	A Boolean value set to true or false that enables (true) checking of a well's operating status during iterations, or disables (false) the checking during iterations.	false

OPM Flow 2022-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
34	--enable-write-all-solutions	A Boolean value set to true or false that turns on (true) or off (false) the writing of all solutions to disk instead of only the ones for the report steps.	false
	OPM Flow Specific Command Line Parameters		
35	--flow-linear-solver-verbosity	A positive integer value that defines the output from linear solver: 0 : No extra output, 1 : output per solution iteration, or 2 : output per iteration.	0
36	--flow-newton-max-iterations	A positive integer that defines the maximum number of Newton iterations per time step used by the simulator.	20
37	--flow-newton-min-iterations	A real positive value that sets the minimum number of Newton iterations per time step used by the simulator.	1
38	--force-disable-fluid-in-place-output	A Boolean value set to true or false that instructs OPM Flow not to print the Fluid In-Place report after each report time step (true) or not (false).	false
39	--force-disable-resv-fluid-in-place-output	A Boolean value set to true or false that instructs OPM Flow to not to print the Reservoir Volume Fluid In-Place report after each report time step (true) or not (false).	false
	General eWoms/ebos Command Line Parameters		
40	--fpga-bitstream	A character string that specifies the bit stream file for the fpgaSolver (including path), usage: '--fpga-bitstream=<filename>'.	“”
41	--full-time-step-initially	A Boolean value set to true or false that instructs OPM Flow to always attempt to finish a report step using a single time step (true) or not (false).	false
42	--ignore-keywords	A character string that defines a list of keywords which should be ignored by OPM Flow. The keyword string should be enclosed in quotes and each keyword separated by a colon, that is ':' .	“”
43	--ilu-fillin-level	A positive integer value that sets the fill in level for the ILU preconditioner.	0

OPM Flow 2022-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
44	--ilu-redblack	A Boolean value set to true or false that instructs OPM Flow to use red-black partitioning for the ILU preconditioner. (true) or not (false).	false
45	--ilu-relaxation	A real positive double precision value that sets the relaxation factor of the linear solver's ILU preconditioner	0.9
46	--ilu-reorder-spheres	A Boolean value set to true or false that specifies OPM Flow to reorder the entries of the matrix in the red-black ILU preconditioner in spheres starting at an edge (true) or not (false) If false the original ordering is preserved in each color. Otherwise try to ensure D4 ordering (in a 2D structured grid, the diagonal elements are consecutive).	false
47	--initial-time-step-in-days	A real double precision value that sets the size of initial time step in days.	1.0
48	--linear-solver-ignore-convergence-failure	A Boolean value set to true or false that if set to true convergences failures in the linear solver are ignored. This option should be used with care, as the results may be unreliable.	false
49	--linear-solver-max-iter	A positive integer value that defines the maximum number of linear iterations.	200
50	--linear-solver-reduction	A real positive double precision value that sets the tolerance for the linear solver. The linear solver convergences when the residual is reduced sufficiently.	0.01
51	--linear-solver-require-full-sparsity-pattern	A Boolean value set to true or false that if set to true the simulator will produce the full sparsity pattern for the linear solver.	false
52	--linear-solver-restart	A positive integer value that sets the number of iterations after which GMRES is restarted.	40

OPM Flow 2022-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
53	--linsolver	A defined quoted character string that sets the configuration of the solver, valid values are: 1) ilu0 (default), 2) cpr_quasiimpes, 3) cpr_trueimpes, or 4) a file specified by the LinearSolverConfigurationJsonFile parameter. The default is "ilu0". Alternatively one can enter a character string enclosed in quotes that defines the filename of a JSON configuration file for the flexible linear solver system. In this case the file extension should be '.json'. Command line parameter changed from --linear-solver to --lin-solver	"ilu0"
54	--matrix-add-well-contributions	A Boolean value set to true or false that if set to true explicitly specifies the influences of wells between cells in the Jacobian and preconditioner matrices.	false
55	--max-inner-iter-ms-wells	A positive integer value that defines the maximum number of inner iterations for multi-segment wells.	100
56	--max-inner-iter-wells	A positive integer value that defines the maximum number of inner iterations for standard wells.	50
57	--max-newton-iterations-with-inner-well-iterations	A positive integer that specifies the maximum newton iterations with inner well iterations.	8.0
58	--max-pressure-change-ms-wells	A real positive value that defines the maximum relative pressure change for a single iteration of the multi-segment well model.	1.0×10^{6}
59	--max-residual-allowed	A real positive value that sets the absolute maximum tolerance for residuals without cutting the time step size.	1.0×10^{7}
60	--max-single-precision-days	A real positive value that sets the maximum time step size where single precision floating point arithmetic can be used for solving the linear systems of equations.	20.0
61	--max-strict-iter	A positive integer that sets the maximum number of Newton iterations before relaxed tolerances are used for the CNV convergence criterion.	0

OPM Flow 2022-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
62	--max-temperature-change	A real positive value that stipulates the maximum absolute change of temperature in a single iteration.	5
63	--max-welleq-iter	A positive integer that defines the maximum number of iterations to determine the solution to the well equations.	30
64	--maximum-number-of-well-switches	A positive integer values that stipulates the maximum number of times a well can switch to the same control.	3
65	--milu-variant	A defined character string that specifies which variant of the modified-ILU preconditioner ought to be used. Possible variants are: 1) ILU (default, plain ILU), 2) MILU_1 (lump diagonal with dropped row entries), 3) MILU_2 (lump diagonal with the sum of the absolute values of the dropped row entries), 4) MILU_3 (if diagonal is positive add sum of dropped row entries. Otherwise subtract them), 5) MILU_4 (if diagonal is positive add sum of dropped row entries. Otherwise do nothing. The default is "ILU"	"ILU"
66	--min-time-step-based-on-newton-iterations	The minimum time step size (in days for field and metric unit and hours for lab unit) can be reduced to based on newton iteration counts.	0
67	--min-time-step-before-shutting-problematic-wells-in-days	A real positive value that sets the minimum time step size in days for which problematic wells are not shut. Time steps below this value will result in problematic wells being shut-in by the simulator.	0.01
68	--newton-max-relax	A real positive value that sets the maximum relaxation factor of a Newton iteration used by the simulator.	0.5
69	--newton-relaxation-type	A character string that defines the type of relaxation used in Newton's method, default is dampen.	"dampen"

OPM Flow 2022-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
70	--opencl-ilu-reorder	A defined character string that selects the reordering strategy for ILU for openclSolver and fpgaSolve, and should be set to: 1) "level_scheduling", or 2) "graph_coloring" Note that "level_scheduling" behaves like Dune and cusparse; whereas, "graph_coloring" is more aggressive and is likely to be faster, but is random-based and generally increases the number of linear solves and linear iterations significantly.	""
71	--opencl-platform-id	A positive integer that specifies the platform identification ("ID") for the openSolver. Use the "clinfo" command to determine valid IDs.	0
72	--output-dir	A character string that defines the directory to which OPM Flow is to write the commercial simulator compatible output files (restart, summary files etc.). The default value results in the files be written to the same directory as the input file.	""
73	--output-interval	A positive integer that specifies the number of report steps between two consecutive writes of restart data.	1

74	--output-mode	A defined character string that defines the output to the *.PRT and *.DBG files:
		1) none: No output to the files.
		2) log or false: Output logging information only.
		3) all or true: Output everything.
		For example to just output logging information use:
		`--output-mode="log"`
		or
		`--output-mode=false`
			"all"

75	--owner-cells-first	A Boolean value set to true or false that determines if the cells owned by a rank should be ordered before ghost/overlapping the cells (true), or not (false).	true
76	--parameter-file	A character string that defines the name of a parameter file which contains the simulator's set of run-time parameters, as listed in this table.	""

OPM Flow 2022-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
77	--pri-var-oscilation-threshold	A real positive value that defines the threshold value for the primary variable switching conditions after its meaning has switched to hinder oscillations.	1.0 x 10-5
78	--print-parameters	A positive integer value that request that the run time parameters be printed at the start of the run: 0 : No output to the files. 1 : Output *.DBG file. 2 : Output to *.DBG and *.PRT files (default).	2
79	--print-properties	A positive integer value that request that the compile time parameters be printed at the start of the run: 0 : No output to the files. 1 : Output *.DBG file. 2 : Output to *.DBG and *.PRT files (default).	2
80	--project-saturations	A Boolean value set to true or false that determines if all the fluid saturations should be scaled to ensure the values are in the interval (0, 1), including runs that use solvents (true), or not (false).	false
81	--regularization-factor-msw	A real positive value that defines the "regularization factor" for multi-segment wells.	1.0
82	--relaxed-max-pv-fraction	A real positive value that defines The fraction of the pore volume of the reservoir where the volumetric error (CNV) may be violated during strict Newton iterations.	0.03
83	--relaxed-pressure-tol-msw	A real positive value that sets the relaxation tolerance for the multi-segment well pressure solution.	50000
84	--relaxed-well-flow-tol	A real positive value that sets the relaxation tolerance for the well flow residual.	1.0
85	--scale-linear-system	A Boolean value set to true or false that switches on (true) or off (false) the scaling of linear system of equations according to the equation scale and primary variable types.	false

OPM Flow 2022-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
86	--sched-restart	A Boolean value that determines for a restart run if the case should initialize wells and groups from the historical SCHEDULE section (true), or from the well and group data on the restart file (false). Note that the commercial simulator always uses data from the restart file (false).	falsch
87	--serial-partitioning	A Boolean value that determines if partitioning for parallel runs on a single process (true), or not (false).	false
88	--shut-unsolvable-wells	A Boolean value that determines if the simulator should shut unsolvable wells (true), or not (false).	false
89	--solve-welleq-initially	A Boolean value set to true or false that determines if the simulator should fully solve the well equations before each iteration of the reservoir model (true), or not (false). Note that the well equations are always added to the full system and solved until converged.	true
90	--solver-continue-on-convergence-failure	A Boolean value that stipulates if the simulator should continue (true) instead of stopping (false) when the minimum solver time step is reached.	false
91	--solver-growth-factor	A real positive value that specifies the allowed value a time step can be increased by, subject to the maximum allowable time step size set by the --flow-solver-max-time-step-in-days parameter. For example, if the current time step has converged at 10 days and --flow-solver-growth-factor is set to the default value, then the next time step will be 2.0 * 10 days, that is at 20 days.	2.0
92	--solver-max-growth	A real positive value that specifies the maximum allowed value a time step can be increased by after a report time step. subject to the maximum allowable time step size set by the --flow-solver-max-time-step-in-days parameter.	3.0
93	--solver-max-restarts	A positive integer that defines the number of allowed consecutive restarts (or time step chops) before the simulation is terminated.	10
94	--solver-max-time-step-in-days	A real positive double precision value that specifies the maximum allowed time step size in days.	365

OPM Flow 2022-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
95	--solver-min-time-step	A real positive double precision value that specifies the minimum size of a time step in seconds. If a step cannot converge without getting cut below this step size the simulator will stop.	1.0×10^{-12}
96	--solver-restart-factor	A real positive double precision value that sets the time step chop factor of the time step after a convergence failure. For example, if the current non-convergent time step is 30 days and --flow-solver-restart-factor is set to the default value, then the time step will be repeated using 0.33 * 30 days as the time step, that is 9.9 days.	0.33
97	--solver-verbosity	A positive integer that specifies the "chattiness" of the non-linear solver.	1
98	--strict-inner-iter-wells	A positive integer that specifies the number of inner well iterations with strict tolerance.	40
99	--strict-outer-iter-wells	A positive integer that specifies the number of newton iterations for which wells are checked with strict tolerance.	99
100	--temperature-max	A real positive value that sets the maximum absolute temperature.	400
101	--temperature-min	A real positive value that sets the minimum absolute temperature.	280
102	--threads-per-process	A positive integer value that stipulates the maximum number of threads to be instantiated per process ('-1' means 'automatic').	-1
103	--time-step-after-event-in-days	A real double precision value that sets maximum allowed time step after an event; for example, when a well is open or closed etc. The default value of -1 means that events to do effect the time stepping.	-1

OPM Flow 2022-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
104	--time-step-control	A defined character string that defines the time stepping control algorithm and is set to one of the following: 1) pid: PID controller based adaptive time step control as suggested by Turek and Kuzmin[381]. 2) pid+iteration: Use PID and linear iteration numbers to guide the time step. 3) pid+newtoniteration: Use PID and non-linear iterations numbers to guide the time step. 4) Hardcoded: Use time steps supplied by user via the time-step-control-filename parameter.	"pid+newtoniteration"
105	--time-step-control-decay-damping-factor	A real positive value that specifies the decay rate a time step can be decreased by after the number target iterations has been exceeded	1.0
106	--time-step-control-decay-rate	A real positive value that specifies the decay rate a time step size can be decreased by after the number of target iterations has been exceeded	0.75
107	--time-step-control-file-name	A character string enclosed in quotes that specifies a filename where time steps are specified. The default is the character string timesteps For instance time steps can be generated by the ecl_summary application in libecl as per the following UNIX command line: `path_to_libecl_applications/ecl_summary DECK TIME > filename` Where: DECK is the name of the data deck you want to get the time steps from, TIME tells the application to return the timing for the run, and "filename" is the name of the file the times are piped to.	"timesteps"
108	--time-step-control-growth-damping-factor	A real positive value that specifies the growth rate of the time step increase when the number of target iterations are undercut.	3.2
109	--time-step-control-growth-rate	A real positive value that specifies the growth rate of the time step size when the number of target iterations are undercut.	1.25

[381] *Algebraic Flux Correction III. Incompressible Flow Problems. Uni Dortmund, Turek and Kuzmin, January, 2006 (DOI: 10.1007/3-540-27206-2_8)*

OPM Flow 2022-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
110	--time-step-control-target-iterations	A positive integer that defines the number of linear iterations which the time step control scheme should aim for (if applicable).	30
111	--time-step-control-target-newton-iterations	A positive integer that specifies the number of Newton iterations which the time step control scheme should aim for (if applicable).	8
112	--time-step-control-tolerance	A real double precision value that sets the tolerance for PID (only used with the pid and pid+ options defined by the --time-step-control option).	0.1
113	--time-step-verbosity	A positive integer that specifies the "chattiness" during the time integration.	1
114	--tolerance-cnv	A real positive double precision value that specifies the maximum non-linear tolerance error. This is the local convergence tolerance (maximum of local saturation errors).	0.01
115	--tolerance-cnv-relaxed	A real positive value that defines the relaxed local convergence tolerance that applies for iterations after the iterations with the strict tolerance.	1.0
116	--tolerance-mb	A real positive double precision value that sets the maximum mass balance error, that is the tolerated mass balance error relative to total mass present.	1.0 x 10-6
117	--tolerance-pressure-ms-wells	A real positive double precision value that specifies the tolerance for the pressure equations for multi-segment wells.	1000
118	--tolerance-well-control	A real positive double precision value that sets the maximum tolerance for the well control equations.	1.0×10^{-7}
119	--tolerance-wells	A real positive double precision value that defines the maximum non-linear error for the well equations.	1.0×10^{-4}
120	-update-equations-scaling	A Boolean value that switches on (true) or off (false) the updating of the scaling factors for mass balance equations during the simulation.	false
121	--use-gmres	A Boolean value that when set to true OPM Flow will use Generalized Minimal Residual ("GMRES") solver instead of Biconjugate Gradient Stabilized ("BiCGSTAB") as the linear solver within the Newton iterations.	false

OPM Flow 2022-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
122	--use-multisegment-well	A Boolean value that when set to true the simulator will use the well model for multi-segment wells instead of the one for single-segment wells.	true
123	--use-update-stabilization	A Boolean value set to true or false that switches on (true) or off (false) the stabilized Newton option, that attempts to detect and correct oscillations or stagnation during the Newton iterations. This option may improve convergence for some cases.	true
124	--vtk-write-diffusion-coefficients	A Boolean value that switches on (true) or off (false) the output of the molecular diffusion coefficients to the VTK[382] output files.	false
125	--vtk-write-effective-diffusion-coefficients	A Boolean value that switches on (true) or off (false) the output of the effective molecular diffusion coefficients for the medium to the VTK output files.	false
126	--vtk-write-tortuosities	A Boolean value that switches on (true) or off (false) the output of the tortuosity for each phase to the VTK output files.	false
127	--zoltan-imbalance-tol	A real positive that defines the tolerable imbalance of the load balancing provided by Zoltan package.	1.1

Notes:

1) Cells colored green in the No. column are new command line parameters for this release. Similarly for the Default column, cells colored green indicate the default value has changed from the previous release.
2) Items shaded in gray are considered to be developer options that should be used with caution as the values associated with these command line parameters are subject to change.
3) VTK Graphics Command Line Parameters[383] set of commands are no longer listed via the basic help command line command:

```
flow --help
```

instead use:

```
flow --help-all
```

to get a list of supported command line Parameters. The –enable-vtk-output option above, if set to "true" will write out the data as stipulated by the compile options. One may need to compile the source code to obtain the desired output.

4) As per all UNIX and LINUX based system the input is case dependent.
5) If a parameter file is used to set one of the above parameters, the leading "--" should not be incorporated in the parameter file.

Table E.3: OPM Flow 2022-04 Command Line Options

[382] *Virtualization ToolKit ("VTK") files used to visualize the 3D results from the model using ParaView, an open-source, multi-platform general purpose data analysis and visualization application (https://www.paraview.org/overview/). For the commercial simulator's compatible output files OPM's ResInsight 3D visualization package can be used instead and is specifically tailored for OPM Flow and the commercial simulator.*

[383] *Virtualization ToolKit ("VTK") files used to visualize the 3D results from the model using ParaView, an open-source, multi-platform general purpose data analysis and visualization application (https://www.paraview.org/overview/). For the commercial simulator's compatible output files OPM's ResInsight 3D visualization package can be used instead and is specifically tailored for OPM Flow and the commercial simulator.*

E.4 RUNNING OPM FLOW 2021-10

The command line syntax for running OPM Flow is:

```
flow [OPTIONS] CASENAME
```

and typing the following command from your terminal:

```
flow CASENAME
```

will start the simulator and run the case specified by CASENAME.DATA.

OPM Flow accepts command line options to control various aspects of the simulator's run time behavior, as described in the table below. To give a command line option use "key=value" syntax, with no spaces around the equals sign. It is also possible to put multiple options together in a parameter file. To do so, put one option on each line of the file with ".param" as the extension and pass that filename as a command line parameter to OPM Flow using:

```
flow --parameter-file=CASENAME.PARAM
```

to instruct OPM Flow to read the parameter file.

OPM Flow 2021-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
	General eWoms/ebos Command Line Parameters		
1	-h or --help	A character string that causes OPM Flow to print a help message that gives a brief description of the available command line parameters. **This now only prints the active user facing command line options.**	N/A
2	--help-all	Prints all the command line options included in the release, including obsolete, hidden and deprecated options.	N/A
3	--accelerator-mode	A defined character string that defines the usage of the GPU (cusparseSolver or openclSolver) or FPGA (fpgaSolver) as the linear solver, usage '--accelerator-mode=[none|cusparse|opencl|fpga|amgcl]'.	"none"
4	--allow-distributed-wells	A boolean value set to true of false that allows the perforations of a well to be distributed to interior of multiple processes in a parallel run (true), or not (false).	false
5	--alternative-well-rate-init	A boolean value set to true of false that causes the simulator to use an alternative well rate initialization procedure (true).	true
6	--bda-device-id	Choose device ID for cusparseSolver or openclSolver, use 'nvidia-smi' or 'clinfo' commands to determine valid IDs.	0

OPM Flow 2021-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
7	--continue-on-convergence-error	A Boolean value set to true or false that that instructs the simulator to continue (true) with a non-converged solution instead of giving up (false) if it encounter a time step size smaller than the minimum time step size.	false
8	--cpr-max-ell-iter	A positive integer that sets the maximum number of Iterations for the elliptic pressure part of the Constrained Pressure Residual ("CPR") solver.	20
9	--cpr-reuse-setup	A positive integer that defines if the CPR solver should re-use the Amg setup. Valid options are: 0 : Recreate the preconditioner for every linear solve. 1 : Recreate once every time step. 2 : Recreate if last linear solve took more than 10 iterations. 3 : Never recreate.	3
10	--dbph-max-rel	A real positive value that sets maximum relative change of the bottom-hole pressure in a single iteration	1.0
11	--dp-max-rel	A real positive double precision value that sets the maximum allowed relative change of pressure per iteration.	0.3
12	--ds-max	A real positive double precision value that sets the maximum allowed change in saturation per iteration.	0.2
13	--dwell-fraction-max	A real positive value that sets the maximum allowed change in well's volume fraction per iteration.	0.2
14	--ecl-deck-file-name	A character string that defines the name of the OPM Flow input file which contains the simulator's ECLIPSE formatted input deck to be simulated. Note that there must be no spaces in the filename. If there are spaces, which can typically occur on Windows based systems, then the filename should be enclosed in quotes; thus instead of FILE NAME use "FILE NAME".	""
15	--ecl-enable-drift-compensation	A Boolean value set to true or false that enables (true) or disables (false) partial compensation of systematic mass losses via the source term of the next time step.	true

OPM Flow 2021-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
16	--ecl-output-double-precision	A Boolean value set to true or false that switches on (true) or off (false) double precision in restart files. Useful for 'perfect' restarts.	false
17	--ecl-output-interval	An integer value defining the number of report steps that ought to be skipped between two writes of restart files results.	-1
18	--ecl-strict-parsing	A Boolean value set to true or false that switches on (true) or off (false) strict parsing mode for parsing - all errors are collected before the application exists. Useful for debugging initial runs.	false
19	--edge-weights-method	A defined positive integer value that defines the edge-weighing strategy: 0 : For uniform, 1 : for trans, or 2 : for log(trans).	1
20	--enable-adaptive-time-stepping	A Boolean value set to true or false that turns on (true) or off (false) adaptive time stepping. If set to false the report time steps provided by the input deck are used.	true
21	--enable-async-ecl-output	A Boolean value set to true or false that sets the output mode to be asynchronously (true), letting the simulator continue computing the next time step while writing results to restart and summary files in the commercial simulators format.	true
22	--enable-async-vtk-output	A Boolean value set to true or false that sets the output mode to be asynchronously (true), letting the simulator continue computing the next time step while writing results to the VTK output files.	true
23	--enable-dry-run	A Boolean value set to true or false that specifies if the simulation should actually run, (true) or just check the input deck (false). This option is equivalent to activating the NOSIM keyword in the RUNSPEC section of the input deck (see section *5.3.97 NOSIM – Activate the No Simulation Mode for Data File Checking*). The default value of "auto' will use whatever is stipulated in the input deck via the NOSIM keyword.	"auto"

OPM Flow 2021-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
24	--enable-ecl-output	A Boolean value set to true or false that specifies if the binary output files (restart, summary files, etc.) should be written in the commercial simulator's format (true), or OPM Flow's format (false).	true
25	--enable-esmry	A Boolean value that switches on (true) or off (false) the output SUMMARY vectors to the ESMRY file for fast loading of summary data.	false
26	--enable-logging-fallout-warning	A Boolean value set to true or false that sets the developer option to see whether logging was on non-root processors (true). If set to true output will be appended to the *.DBG or *.PRT files.	false
27	--enable-opm-rst-file	A Boolean value set to true or false to write OPM specific data sets to the commercial simulators restart file to enable restart of an OPM Flow runs (true) by OPM Flow, or not to write the data (false).	false
28	--enable-storage-cache	A Boolean value set to true or false that turns on (true) or off (false) storing previous storage terms and avoid re-calculating them.	true
29	--enable-terminal-output	A Boolean value set to true or false that turns on (true) or off (false) high-level information about the simulation's progress to the terminal	true
30	--enable-tracer-model	A Boolean value set to true or false that turns on (true) or off (false) transport tracer calculations for when tracers have been declared in the input deck.	true
31	--enable-tuning	A Boolean value set to true or false that instructs OPM Flow to read the time stepping parameters from the *TUNING - Numerical Tuning Control* keyword in the input deck, if set to true. Note that only the first record of the TUNING keyword is processed.	false
32	--enable-vtk-output	A Boolean value set to true or false that turns on (true) or off (false) a global switch for writing VTK files.	false
33	--enable-well-operability-check	A Boolean value set to true or false that enables (true) checking of a well's operating status, or disables (false) the checking.	true

OPM Flow 2021-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
34	--enable-well-operability-check-iter	A Boolean value set to true or false that enables (true) checking of a well's operating status during iterations, or disables (false) the checking during iterations.	false
35	--enable-write-all-solutions	A Boolean value set to true or false that turns on (true) or off (false) the writing of all solutions to disk instead of only the ones for the report steps.	false
	OPM Flow Specific Command Line Parameters		
36	--flow-linear-solver-verbosity	A positive integer value that defines the output from linear solver: 0 : No extra output, 1 : output per solution iteration, or 2 : output per iteration.	0
37	--flow-newton-max-iterations	A positive integer that defines the maximum number of Newton iterations per time step used by the simulator.	20
38	--flow-newton-min-iterations	A real positive value that sets the minimum number of Newton iterations per time step used by the simulator.	1
39	--force-disable-fluid-in-place-output	A Boolean value set to true or false that instructs OPM Flow not to print the Fluid In-Place report after each report time step (true) or not (false).	false
40	--force-disable-resv-fluid-in-place-output	A Boolean value set to true or false that instructs OPM Flow to not to print the Reservoir Volume Fluid In-Place report after each report time step (true) or not (false).	false
	General eWoms/ebos Command Line Parameters		
41	--fpga-bitstream	A character string that specifies the bit stream file for the fpgaSolver (including path), usage: '--fpga-bitstream=<filename>'.	""
42	--full-time-step-initially	A Boolean value set to true or false that instructs OPM Flow to always attempt to finish a report step using a single time step (true) or not (false).	false
43	--ignore-keywords	A character string that defines a list of keywords which should be ignored by OPM Flow. The keyword string should be enclosed in quotes and each keyword separated by a colon, that is ':' .	""
44	--ilu-fillin-level	A positive integer value that sets the fill in level for the ILU preconditioner.	0

OPM Flow 2021-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
45	--ilu-redblack	A Boolean value set to true or false that instructs OPM Flow to use red-black partitioning for the ILU preconditioner. (true) or not (false).	false
46	--ilu-relaxation	A real positive double precision value that sets the relaxation factor of the linear solver's ILU preconditioner	0.9
47	--ilu-reorder-spheres	A Boolean value set to true or false that specifies OPM Flow to reorder the entries of the matrix in the red-black ILU preconditioner in spheres starting at an edge (true) or not (false) If false the original ordering is preserved in each color. Otherwise try to ensure D4 ordering (in a 2D structured grid, the diagonal elements are consecutive).	false
48	--initial-time-step-in-days	A real double precision value that sets the size of initial time step in days.	1.0
49	--linear-solver-ignore-convergence-failure	A Boolean value set to true or false that if set to true convergences failures in the linear solver are ignored. This option should be used with care, as the results may be unreliable.	false
50	--linear-solver-max-iter	A positive integer value that defines the maximum number of linear iterations.	200
51	--linear-solver-reduction	A real positive double precision value that sets the tolerance for the linear solver. The linear solver convergences when the residual is reduced sufficiently.	0.01
52	--linear-solver-require-full-sparsity-pattern	A Boolean value set to true or false that if set to true the simulator will produce the full sparsity pattern for the linear solver.	false
53	--linear-solver-restart	A positive integer value that sets the number of iterations after which GMRES is restarted.	40

OPM Flow 2021-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
54	--linsolver	A defined quoted character string that sets the configuration of the solver, valid values are: 1) ilu0 (default), 2) cpr_quasiimpes, 3) cpr_trueimpes, or 4) a file specified by the LinearSolverConfigurationJsonFile parameter. The default is "ilu0". Alternatively one can enter a character string enclosed in quotes that defines the filename of a JSON configuration file for the flexible linear solver system. In this case the file extension should be '.json'. Command line parameter changed from --linear-solver to --lin-solver	"ilu0"
55	--matrix-add-well-contributions	A Boolean value set to true or false that if set to true explicitly specifies the influences of wells between cells in the Jacobian and preconditioner matrices.	false
56	--max-inner-iter-ms-wells	A positive integer value that defines the maximum number of inner iterations for multi-segment wells.	100
57	--max-inner-iter-wells	A positive integer value that defines the maximum number of inner iterations for standard wells.	50
58	--max-newton-iterations-with-inner-well-iterations	A positive integer that specifies the maximum newton iterations with inner well iterations.	8.0
59	--max-pressure-change-ms-wells	A real positive value that defines the maximum relative pressure change for a single iteration of the multi-segment well model.	1.0×10^{6}
60	--max-residual-allowed	A real positive value that sets the absolute maximum tolerance for residuals without cutting the time step size.	1.0×10^{7}
61	--max-single-precision-days	A real positive value that set the maximum time step size where single precision floating point arithmetic can be used solving for the linear systems of equations.	20.0
62	--max-strict-iter	A positive integer that sets the maximum number of Newton iterations before relaxed tolerances are used for the CNV convergence criterion.	0

OPM Flow 2021-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
63	--max-temperature-change	A real positive value that stipulates the maximum absolute change of temperature in a single iteration.	5
64	--max-welleq-iter	A positive integer that defines the maximum number of iterations to determine the solution to the well equations.	30
65	--maximum-number-of-well-switches	A positive integer values that stipulates the maximum number of times a well can switch to the same control.	3
66	--milu-variant	A defined character string that specifies which variant of the modified-ILU preconditioner ought to be used. Possible variants are: 1) ILU (default, plain ILU), 2) MILU_1 (lump diagonal with dropped row entries), 3) MILU_2 (lump diagonal with the sum of the absolute values of the dropped row entries), 4) MILU_3 (if diagonal is positive add sum of dropped row entries. Otherwise subtract them), 5) MILU_4 (if diagonal is positive add sum of dropped row entries. Otherwise do nothing. The default is "ILU"	"ILU"
67	--min-time-step-based-on-newton-iterations	The minimum time step size (in days for field and metric unit and hours for lab unit) can be reduced to based on newton iteration counts.	0
68	--min-time-step-before-shutting-problematic-wells-in-days	A real positive value that sets the minimum time step size in days for which problematic wells are not shut. Time steps below this value will result in problematic wells being shut-in by the simulator.	0.01
69	--newton-max-relax	A real positive value that sets the maximum relaxation factor of a Newton iteration used by the simulator.	0.5
70	--newton-relaxation-type	A character string that defines the type of relaxation used in Newton's method, default is dampen.	"dampen"

OPM Flow 2021-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
71	--opencl-ilu-reorder	A defined character string that selects the reordering strategy for ILU for openclSolver and fpgaSolve, and should be set to: 1) "level_scheduling", or 2) "graph_coloring" Note that "level_scheduling" behaves like Dune and cusparse; whereas, "graph_coloring" is more aggressive and is likely to be faster, but is random-based and generally increases the number of linear solves and linear iterations significantly.	""
72	--opencl-platform-id	A positive integer that specifies the platform identification ("ID") for the openSolver. Use the "clinfo" command to determine valid IDs.	0
73	--output-dir	A character string that defines the directory to which OPM Flow is to write the commercial simulator compatible output files (restart, summary files etc.). The default value results in the files be written to the same directory as the input file.	""
74	--output-interval	A positive integer that specifies the number of report steps between two consecutive writes of restart data.	1

75	--output-mode	A defined character string that defines the output to the *.PRT and *.DBG files:
		1) none: No output to the files.
		2) log or false: Output logging information only.
		3) all or true: Output everything.
		For example to just output logging information use:
		`--output-mode="log"`
		or
		`--output-mode=false`
			"all"

76	--owner-cells-first	A Boolean value set to true or false that determines if the cells owned by a rank should be ordered before ghost/overlapping the cells (true), or not (false).	true
77	--parameter-file	A character string that defines the name of a parameter file which contains the simulator's set of run-time parameters, as listed in this table.	""

OPM Flow 2021-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
78	--pri-var-oscilation-threshold	A real positive value that defines the threshold value for the primary variable switching conditions after its meaning has switched to hinder oscillations.	1.0 x 10-5
79	--print-parameters	A positive integer value that request that the run time parameters be printed at the start of the run: 0 : No output to the files. 1 : Output *.DBG file. 2 : Output to *.DBG and *.PRT files (default).	2
80	--print-properties	A positive integer value that request that the compile time parameters be printed at the start of the run: 0 : No output to the files. 1 : Output *.DBG file. 2 : Output to *.DBG and *.PRT files (default).	2
81	--project-saturations	A Boolean value set to true or false that determines if all the fluid saturations should be scaled to ensure the values are in the interval (0, 1), including runs that use solvents (true), or not (false).	false
82	--regularization-factor-msw	A real positive value that defines the "regularization factor" for multi-segment wells.	1.0
83	--relaxed-max-pv-fraction	A real positive value that defines The fraction of the pore volume of the reservoir where the volumetric error (CNV) may be violated during strict Newton iterations.	0.03
84	--relaxed-pressure-tol-msw	A real positive value that sets the relaxation tolerance for the multi-segment well pressure solution.	50000
85	--relaxed-well-flow-tol	A real positive value that sets the relaxation tolerance for the well flow residual.	1.0
86	--scale-linear-system	A Boolean value set to true or false that switches on (true) or off (false) the scaling of linear system of equations according to the equation scale and primary variable types.	false

OPM Flow 2021-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
87	--sched-restart	A Boolean value that determines for a restart run if the case should initialize wells and groups from the historical SCHEDULE section (true), or from the well and group data on the restart file (false). Note that the commercial simulator always uses data from the restart file (false).	true
88	--serial-partitioning	A Boolean value that determines if partitioning for parallel runs on a single process (true), or not (false).	false
89	--shut-unsolvable-wells	A Boolean value that determines if the simulator should shut unsolvable wells (true), or not (false).	false
90	--solve-welleq-initially	A Boolean value set to true or false that determines if the simulator should fully solve the well equations before each iteration of the reservoir model (true), or not (false). Note that the well equations are always added to the full system and solved until converged.	true
91	--solver-continue-on-convergence-failure	A Boolean value that stipulates if the simulator should continue (true) instead of stopping (false) when the minimum solver time step is reached.	false
92	--solver-growth-factor	A real positive value that specifies the allowed value a time step can be increased by, subject to the maximum allowable time step size set by the --flow-solver-max-time-step-in-days parameter. For example, if the current time step has converged at 10 days and --flow-solver-growth-factor is set to the default value, then the next time step will be 2.0 * 10 days, that is at 20 days.	2.0
93	--solver-max-growth	A real positive value that specifies the maximum allowed value a time step can be increased by after a report time step. subject to the maximum allowable time step size set by the --flow-solver-max-time-step-in-days parameter.	3.0
94	--solver-max-restarts	A positive integer that defines the number of allowed consecutive restarts (or time step chops) before the simulation is terminated.	10
95	--solver-max-time-step-in-days	A real positive double precision value that specifies the maximum allowed time step size in days.	365

OPM Flow 2021-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
96	--solver-min-time-step	A real positive double precision value that specifies the minimum size of a time step in seconds. If a step cannot converge without getting cut below this step size the simulator will stop.	1.0×10^{-12}
97	--solver-restart-factor	A real positive double precision value that sets the time step chop factor of the time step after a convergence failure. For example, if the current non-convergent time step is 30 days and --flow-solver-restart-factor is set to the default value, then the time step will be repeated using 0.33 * 30 days as the time step, that is 9.9 days.	0.33
98	--solver-verbosity	A positive integer that specifies the "chattiness" of the non-linear solver.	1
99	--strict-inner-iter-wells	A positive integer that specifies the number of inner well iterations with strict tolerance.	40
100	--strict-outer-iter-wells	A positive integer that specifies the number of newton iterations for which wells are checked with strict tolerance.	99
101	--temperature-max	A real positive value that sets the maximum absolute temperature.	400
102	--temperature-min	A real positive value that sets the minimum absolute temperature.	280
103	--threads-per-process	A positive integer value that stipulates the maximum number of threads to be instantiated per process ('-1' means 'automatic').	-1
104	--time-step-after-event-in-days	A real double precision value that sets maximum allowed time step after an event; for example, when a well is open or closed etc. The default value of -1 means that events to do effect the time stepping.	-1

OPM Flow 2021-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
105	--time-step-control	A defined character string that defines the time stepping control algorithm and is set to one of the following: 1) pid: PID controller based adaptive time step control as suggested by Turek and Kuzmin[384]. 2) pid+iteration: Use PID and linear iteration numbers to guide the time step. 3) pid+newtoniteration: Use PID and non-linear iterations numbers to guide the time step. 4) Hardcoded: Use time steps supplied by user via the time-step-control-filename parameter.	"pid+newtoniteration"
106	--time-step-control-decay-damping-factor	A real positive value that specifies the decay rate a time step can be decreased by after the number target iterations has been exceeded	1.0
107	--time-step-control-decay-rate	A real positive value that specifies the decay rate a time step size can be decreased by after the number of target iterations has been exceeded	0.75
108	--time-step-control-file-name	A character string enclosed in quotes that specifies a filename where time steps are specified. The default is the character string timesteps For instance time steps can be generated by the ecl_summary application in libecl as per the following UNIX command line: `path_to_libecl_applications/ecl_summary DECK TIME > filename` Where: DECK is the name of the data deck you want to get the time steps from, TIME tells the application to return the timing for the run, and "filename" is the name of the file the times are piped to.	"timesteps"
109	--time-step-control-growth-damping-factor	A real positive value that specifies the growth rate of the time step increase when the number of target iterations are undercut.	3.2
110	--time-step-control-growth-rate	A real positive value that specifies the growth rate of the time step size when the number of target iterations are undercut.	1.25

[384] *Algebraic Flux Correction III. Incompressible Flow Problems. Uni Dortmund, Turek and Kuzmin, January, 2006 (DOI: 10.1007/3-540-27206-2_8)*

OPM Flow 2021-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
111	--time-step-control-target-iterations	A positive integer that defines the number of linear iterations which the time step control scheme should aim for (if applicable).	30
112	--time-step-control-target-newton-iterations	A positive integer that specifies the number of Newton iterations which the time step control scheme should aim for (if applicable).	8
113	--time-step-control-tolerance	A real double precision value that sets the tolerance for PID (only used with the pid and pid+ options defined by the --time-step-control option).	0.1
114	--time-step-verbosity	A positive integer that specifies the "chattiness" during the time integration.	1
115	--tolerance-cnv	A real positive double precision value that specifies the maximum non-linear tolerance error. This is the local convergence tolerance (maximum of local saturation errors).	0.01
116	--tolerance-cnv-relaxed	A real positive value that defines the relaxed local convergence tolerance that applies for iterations after the iterations with the strict tolerance.	1.0
117	--tolerance-mb	A real positive double precision value that sets the maximum mass balance error, that is the tolerated mass balance error relative to total mass present.	1.0 x 10-6
118	--tolerance-pressure-ms-wells	A real positive double precision value that specifies the tolerance for the pressure equations for multi-segment wells.	1000
119	--tolerance-well-control	A real positive double precision value that sets the maximum tolerance for the well control equations.	1.0×10^{-7}
120	--tolerance-wells	A real positive double precision value that defines the maximum non-linear error for the well equations.	1.0×10^{-4}
121	-update-equations-scaling	A Boolean value that switches on (true) or off (false) the updating of the scaling factors for mass balance equations during the simulation.	false

OPM Flow 2021-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
122	--use-gmres	A Boolean value that when set to true OPM Flow will use Generalized Minimal Residual ("GMRES") solver instead of Biconjugate Gradient Stabilized ("BiCGSTAB") as the linear solver within the Newton iterations.	false
123	--use-multisegment-well	A Boolean value that when set to true the simulator will use the well model for multi-segment wells instead of the one for single-segment wells.	true
124	--use-update-stabilization	A Boolean value set to true or false that switches on (true) or off (false) the stabilized Newton option, that attempts to detect and correct oscillations or stagnation during the Newton iterations. This option may improve convergence for some cases.	true
125	--vtk-write-diffusion-coefficients	A Boolean value that switches on (true) or off (false) the output of the molecular diffusion coefficients to the VTK[385] output files.	false
126	--vtk-write-effective-diffusion-coefficients	A Boolean value that switches on (true) or off (false) the output of the effective molecular diffusion coefficients for the medium to the VTK output files.	false
127	--vtk-write-tortuosities	A Boolean value that switches on (true) or off (false) the output of the tortuosity for each phase to the VTK output files.	false
128	--zoltan-imbalance-tol	A real positive that defines the tolerable imbalance of the load balancing provided by Zoltan package.	1.1

[385] *Virtualization ToolKit ("VTK") files used to visualize the 3D results from the model using ParaView, an open-source, multi-platform general purpose data analysis and visualization application (https://www.paraview.org/overview/). For the commercial simulator's compatible output files OPM's ResInsight 3D visualization package can be used instead and is specifically tailored for OPM Flow and the commercial simulator.*

OPM Flow 2021-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**

Notes:

1) Cells colored green in the No. column are new command line parameters for this release. Similarly for the Default column, cells colored green indicate the default value has changed from the previous release.
2) Items shaded in gray are considered to be developer options that should be used with caution as the values associated with these command line parameters are subject to change.
3) VTK Graphics Command Line Parameters[386] set of commands are no longer listed via the basic help command line command:

```
flow --help
```

instead use:

```
flow --help-all
```

to get a list of supported command line Parameters. The –enable-vtk-output option above, if set to "true" will write out the data as stipulated by the compile options. One may need to compile the source code to obtain the desired output.

4) As per all UNIX and LINUX based system the input is case dependent.
5) If a parameter file is used to set one of the above parameters, the leading "--" should not be incorporated in the parameter file.

Table E.4: OPM Flow 2021-10 Command Line Options

[386] *Virtualization ToolKit ("VTK") files used to visualize the 3D results from the model using ParaView, an open-source, multi-platform general purpose data analysis and visualization application (https://www.paraview.org/overview/). For the commercial simulator's compatible output files OPM's ResInsight 3D visualization package can be used instead and is specifically tailored for OPM Flow and the commercial simulator.*

E.5 RUNNING OPM FLOW 2021-04

The command line syntax for running OPM Flow is:

```
flow [OPTIONS] CASENAME
```

and typing the following command from your terminal:

```
flow CASENAME
```

will start the simulator and run the case specified by CASENAME.DATA.

OPM Flow accepts command line options to control various aspects of the simulator's run time behavior, as described in the table below. To give a command line option use "key=value" syntax, with no spaces around the equals sign. It is also possible to put multiple options together in a parameter file. To do so, put one option on each line of the file with ".param" as the extension and pass that filename as a command line parameter to OPM Flow using:

```
flow --parameter-file=CASENAME.PARAM
```

to instruct OPM Flow to read the parameter file.

The available command line options for this release are listed in Table E.5.

OPM Flow 2021-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
	General eWoms/ebos Command Line Parameters		
1	-h or --help	A character string that causes OPM Flow to print a help message that gives a brief description of the available command line parameters. **This now only prints the active user facing command line options.**	N/A
2	--help-all	Prints all the command line options included in the release, including obsolete, hidden and deprecated options.	N/A
3	--allow-distributed-wells	A boolean value set to true of false that allows the perforations of a well to be distributed to interior of multiple processes in a parallel run (true), or not (false).	false
4	--alternative-well-rate-init	A boolean value set to true of false that causes the simulator to use an alternative well rate initialization procedure (true).	true
5	--bda-device-id	Choose device ID for cusparseSolver or openclSolver, use 'nvidia-smi' or 'clinfo' commands to determine valid IDs.	0
6	--continue-on-convergence-error	A Boolean value set to true or false that that instructs the simulator to continue (true) with a non-converged solution instead of giving up (false) if it encounter a time step size smaller than the minimum time step size.	false

OPM Flow 2021-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
7	--cpr-max-ell-iter	A positive integer that sets the maximum number of Iterations for the elliptic pressure part of the Constrained Pressure Residual ("CPR") solver.	20
8	--cpr-reuse-setup	A positive integer that defines if the CPR solver should re-use the Amg setup. Valid options are: 0 : Recreate the preconditioner for every linear solve. 1 : Recreate once every time step. 2 : Recreate if last linear solve took more than 10 iterations. 3 : Never recreate.	0
9	--dbph-max-rel	A real positive value that sets maximum relative change of the bottom-hole pressure in a single iteration	1.0
10	--dp-max-rel	A real positive double precision value that sets the maximum allowed relative change of pressure per iteration.	0.3
11	--ds-max	A real positive double precision value that sets the maximum allowed change in saturation per iteration.	0.2
12	--dwell-fraction-max	A real positive value that sets the maximum allowed change in well's volume fraction per iteration.	0.2
13	--ecl-deck-file-name	A character string that defines the name of the OPM Flow input file which contains the simulator's ECLIPSE formatted input deck to be simulated.	""
14	--ecl-enable-drift-compensation	A Boolean value set to true or false that enables (true) or disables (false) partial compensation of systematic mass losses via the source term of the next time step.	true
15	--ecl-output-double-precision	A Boolean value set to true or false that switches on (true) or off (false) double precision in restart files. Useful for 'perfect' restarts.	false
16	--ecl-output-interval	An integer value defining the number of report steps that ought to be skipped between two writes of restart files results.	-1
17	--ecl-strict-parsing	A Boolean value set to true or false that switches on (true) or off (false) strict parsing mode for parsing - all errors are collected before the application exists. Useful for debugging initial runs.	false

OPM Flow 2021-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
18	--edge-weights-method	A defined positive integer value that defines the edge-weighing strategy: 0 : For uniform, 1 : for trans, or 2 : for log(trans).	1
19	--enable-adaptive-time-stepping	A Boolean value set to true or false that turns on (true) or off (false) adaptive time stepping. If set to false the report time steps provided by the input deck are used.	true
20	--enable-async-ecl-output	A Boolean value set to true or false that sets the output mode to be asynchronously (true), letting the simulator continue computing the next time step while writing results to restart and summary files in the commercial simulators format.	true
21	--enable-async-vtk-output	A Boolean value set to true or false that sets the output mode to be asynchronously (true), letting the simulator continue computing the next time step while writing results to the VTK output files.	true
22	--enable-dry-run	A Boolean value set to true or false that specifies if the simulation should actually run, (true) or just check the input deck (false). This option is equivalent to activating the NOSIM keyword in the RUNSPEC section of the input deck (see section *5.3.97 NOSIM – Activate the No Simulation Mode for Data File Checking*). The default value of "auto' will use whatever is stipulated in the input deck via the NOSIM keyword.	"auto"
23	--enable-ecl-output	A Boolean value set to true or false that specifies if the binary output files (restart, summary files, etc.) should be written in the commercial simulator's format (true), or OPM Flow's format (false).	true
24	--enable-logging-fallout-warning	A Boolean value set to true or false that sets the developer option to see whether logging was on non-root processors (true). If set to true output will be appended to the *.DBG or *.PRT files.	false
25	--enable-opm-rst-file	A Boolean value set to true or false to write OPM specific data sets to the commercial simulators restart file to enable restart of an OPM Flow runs (true) by OPM Flow, or not to write the data (false).	false

OPM Flow 2021-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
26	--enable-storage-cache	A Boolean value set to true or false that turns on (true) or off (false) storing previous storage terms and avoid re-calculating them.	true
27	--enable-terminal-output	A Boolean value set to true or false that turns on (true) or off (false) high-level information about the simulation's progress to the terminal	true
28	--enable-tracer-model	A Boolean value set to true or false that turns on (true) or off (false) transport tracer calculations for when tracers have been declared in the input deck. **Experimental - use with caution.**	false
29	--enable-tuning	A Boolean value set to true or false that instructs OPM Flow to read the time stepping parameters from the *TUNING - Numerical Tuning Control* keyword in the input deck, if set to true. Note that only the first record of the TUNING keyword is processed.	false
30	--enable-vtk-output	A Boolean value set to true or false that turns on (true) or off (false) a global switch for writing VTK files.	false
31	--enable-well-operability-check	A Boolean value set to true or false that enables (true) checking of a well's operating status, or disables (false) the checking.	true
32	--enable-write-all-solutions	A Boolean value set to true or false that turns on (true) or off (false) the writing of all solutions to disk instead of only the ones for the report steps.	false
	OPM Flow Specific Command Line Parameters		
33	--flow-linear-solver-verbosity	A positive integer value that defines the output from linear solver: 0 : No extra output, 1 : output per solution iteration, or 2 : output per iteration.	0
34	--flow-newton-max-iterations	A positive integer that defines the maximum number of Newton iterations per time step used by the simulator.	20
35	--flow-newton-min-iterations	A real positive value that sets the minimum number of Newton iterations per time step used by the simulator.	1
	General eWoms/ebos Command Line Parameters		

OPM Flow 2021-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
36	--force-disable-fluid-in-place-output	A Boolean value set to true or false that instructs OPM Flow to not to print the Fluid In-Place report after each report time step (true) or not (false).	false
37	--full-time-step-initially	A Boolean value set to true or false that instructs OPM Flow to always attempt to finish a report step using a single time step (true) or not (false).	false
38	--gpu-mode	A character string that defines which GPU option to use for the linear solver cusparseSolver or openclSolver. The parameter should be set to: 1) "none|" for no use of the GPU solver, 2) "cusparse" to use the cusparseSolver solver, or 3) "opencl' for the openclSolver.	"none"
39	--ignore-keywords	A character string that defines a list of keywords which should be ignored by OPM Flow. The keyword string should be enclosed in quotes and each keyword separated by a colon, that is ':' .	""
40	--ilu-fillin-level	A positive integer value that sets the fill in level for the ILU preconditioner.	0
41	--ilu-redblack	A Boolean value set to true or false that instructs OPM Flow to use red-black partitioning for the ILU preconditioner. (true) or not (false).	false
42	--ilu-relaxation	A real positive double precision value that sets the relaxation factor of the linear solver's ILU preconditioner	0.9
43	--ilu-reorder-spheres	A Boolean value set to true or false that specifies OPM Flow to reorder the entries of the matrix in the red-black ILU preconditioner in spheres starting at an edge (true) or not (false) If false the original ordering is preserved in each color. Otherwise try to ensure D4 ordering (in a 2D structured grid, the diagonal elements are consecutive).	false
44	--initial-time-step-in-days	A real double precision value that sets the size of initial time step in days.	1.0
45	--linear-solver-ignore-convergence-failure	A Boolean value set to true or false that if set to true convergences failures in the linear solver are ignored. This option should be used with care, as the results may be unreliable.	false

OPM Flow 2021-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
46	--linear-solver-max-iter	A positive integer value that defines the maximum number of linear iterations.	200
47	--linear-solver-reduction	A real positive double precision value that sets the tolerance for the linear solver. The linear solver convergences when the residual is reduced sufficiently.	0.01
48	--linear-solver-require-full-sparsity-pattern	A Boolean value set to true or false that if set to true the simulator will produce the full sparsity pattern for the linear solver.	false
49	--linear-solver-restart	A positive integer value that sets the number of iterations after which GMRES is restarted.	40
50	--linear-solver	A defined quoted character string that sets the configuration of the solver, valid values are: 1) ilu0 (default), 2) cpr_quasiimpes, 3) cpr_trueimpes, or 4) a file specified by the LinearSolverConfigurationJsonFile parameter. The default is "ilu0". Alternatively one can enter a character string enclosed in quotes that defines the filename of a JSON configuration file for the flexible linear solver system. In this case the file extension should be '.json'.	"ilu0"
51	--matrix-add-well-contributions	A Boolean value set to true or false that if set to true explicitly specifies the influences of wells between cells in the Jacobian and preconditioner matrices.	false
52	--max-inner-iter-ms-wells	A positive integer value that defines the maximum number of inner iterations for multi-segment wells.	100
53	--max-inner-iter-wells	A positive integer value that defines the maximum number of inner iterations for standard wells.	50
54	--max-pressure-change-ms-wells	A real positive value that defines the maximum relative pressure change for a single iteration of the multi-segment well model.	1.0×10^{6}
55	--max-residual-allowed	A real positive value that sets the absolute maximum tolerance for residuals without cutting the time step size.	1.0×10^{7}

OPM Flow 2021-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
56	--max-single-precision-days	A real positive value that set the maximum time step size where single precision floating point arithmetic can be used solving for the linear systems of equations.	20.0
57	--max-strict-iter	A positive integer that sets the maximum number of Newton iterations before relaxed tolerances are used for the CNV convergence criterion.	0
58	--max-welleq-iter	A positive integer that defines the maximum number of iterations to determine the solution to the well equations.	30
59	--milu-variant	A defined character string that specifies which variant of the modified-ILU preconditioner ought to be used. Possible variants are: 1) ILU (default, plain ILU), 2) MILU_1 (lump diagonal with dropped row entries), 3) MILU_2 (lump diagonal with the sum of the absolute values of the dropped row entries), 4) MILU_3 (if diagonal is positive add sum of dropped row entries. Otherwise subtract them), 5) MILU_4 (if diagonal is positive add sum of dropped row entries. Otherwise do nothing. The default is "ILU"	"ILU"
60	--min-time-step-based-on-newton-iterations	The minimum time step size (in days for field and metric unit and hours for lab unit) can be reduced to based on newton iteration counts.	0
61	--min-time-step-before-shutting-problematic-wells-in-days	A real positive value that sets the minimum time step size in days for which problematic wells are not shut. Time steps below this value will result in problematic wells being shut-in by the simulator.	0.01
62	--newton-max-relax	A real positive value that sets the maximum relaxation factor of a Newton iteration used by the simulator.	0.5
63	--newton-relaxation-type	A character string that defines the type of relaxation used in Newton's method, default is dampen.	"dampen"

OPM Flow 2021-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
64	--opencl-ilu-reorder	A defined character string that selects the reordering strategy for ILU for openclSolver and should be set to: 1) "level_scheduling", or 2) "graph_coloring" Note that "level_scheduling" behaves like Dune and cusparse, whereas "graph_coloring" is more aggressive and is likely to be faster, but is random-based and generally increases the number of linear solves and linear iterations significantly.	"graph_coloring"
65	--opencl-platform-id	A positive integer that specifies the platform identification ("ID") for the openSolver. Use the "clinfo" command to determine valid IDs.	0
66	--output-dir	A character string that defines the directory to which OPM Flow is to write the commercial simulator compatible output files (restart, summary files etc.). The default value results in the files be written to the same directory as the input file.	""
67	--output-interval	A positive integer that specifies the number of report steps between two consecutive writes of restart data.	1

68	--output-mode	A defined character string that defines the output to the *.PRT and *.DBG files:
		1) none: No output to the files.
		2) log or false: Output logging information only.
		3) all or true: Output everything.
		For example to just output logging information use:
		`--output-mode="log"`
		or
		`--output-mode=false`
			"all"

69	--owner-cells-first	A Boolean value set to true or false that determines if the cells owned by a rank should be ordered before ghost/overlapping the cells (true), or not (false).	true
70	--parameter-file	A character string that defines the name of a parameter file which contains the simulator's set of run-time parameters, as listed in this table.	""
71	--pri-var-oscilation-threshold	A real positive value that defines the threshold value for the primary variable switching conditions after its meaning has switched to hinder oscillations.	1.0×10^{-5}

OPM Flow 2021-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
72	--print-parameters	A positive integer value that request that the run time parameters be printed at the start of the run: 0 : No output to the files. 1 : Output *.DBG file. 2 : Output to *.DBG and *.PRT files (default).	2
73	--print-properties	A positive integer value that request that the compile time parameters be printed at the start of the run: 0 : No output to the files. 1 : Output *.DBG file. 2 : Output to *.DBG and *.PRT files (default).	2
74	--project-saturations	A Boolean value set to true or false that determines if all the fluid saturations should be scaled to ensure the values are in the interval (0, 1), including runs that use solvents (true), or not (false).	false
75	--regularization-factor-msw	A real positive value that defines the "regularization factor" for multi-segment wells.	1.0
76	--relaxed-flow-tol-inner-iter-msw	A real positive value that sets the relaxation tolerance for the inner iteration for the multi-segment well flow solution	1.0
77	--relaxed-max-pv-fraction	A real positive value that defines The fraction of the pore volume of the reservoir where the volumetric error (CNV) may be violated during strict Newton iterations.	0.03
78	--relaxed-pressure-tol-inner-iter-msw	A real positive value that sets the relaxation tolerance for the inner iteration for the multi-segment well pressure solution.	50000
79	--scale-linear-system	A Boolean value set to true or false that switches on (true) or off (false) the scaling of linear system of equations according to the equation scale and primary variable types.	false
80	--sched-restart	A Boolean value that determines for a restart run if the case should initialize wells and groups from the historical SCHEDULE section (true), or from the well and group data on the restart file (false). Note that the commercial simulator always uses data from the restart file (false).	true

OPM Flow 2021-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
81	--serial-partitioning	A Boolean value that determines if partitioning for parallel runs on a single process (true), or not (false).	false
82	--solve-welleq-initially	A Boolean value set to true or false that switches on (true) or off (false) the solving of the well equations as a pre-processing step. Note that the well equations are always added to the full system and solved until converged.	true
83	--solver-growth-factor	A real positive value that specifies the allowed value a time step can be increased by, subject to the maximum allowable time step size set by the --flow-solver-max-time-step-in-days parameter. For example, if the current time step has converged at 10 days and --flow-solver-growth-factor is set to the default value, then the next time step will be 2.0 * 10 days, that is at 20 days.	2.0
84	--solver-max-growth	A real positive value that specifies the maximum allowed value a time step can be increased by after a report time step. subject to the maximum allowable time step size set by the --flow-solver-max-time-step-in-days parameter.	3.0
85	--solver-max-restarts	A positive integer that defines the number of allowed consecutive restarts (or time step chops) before the simulation is terminated.	10
86	--solver-max-time-step-in-days	A real positive double precision value that specifies the maximum allowed time step size in days.	365
87	--solver-min-time-step	A real positive double precision value that specifies the minimum size of a time step in seconds. If a step cannot converge without getting cut below this step size the simulator will stop.	1.0×10^{-12}
88	--solver-restart-factor	A real positive double precision value that sets the time step chop factor of the time step after a convergence failure. For example, if the current non-convergent time step is 30 days and --flow-solver-restart-factor is set to the default value, then the time step will be repeated using 0.33 * 30 days as the time step, that is 9.9 days.	0.33
89	--solver-verbosity	A positive integer that specifies the "chattiness" of the non-linear solver.	1

OPM Flow 2021-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
90	--strict-inner-iter-ms-wells	A positive integer that specifies the number of inner iterations for multi-segment wells with strict tolerance.	40
91	--threads-per-process	A positive integer value that stipulates the maximum number of threads to be instantiated per process ('-1' means 'automatic').	-1
92	--time-step-after-event-in-days	A real double precision value that sets maximum allowed time step after an event; for example, when a well is open or closed etc. The default value of -1 means that events to do effect the time stepping.	-1
93	--time-step-control	A defined character string that defines the time stepping control algorithm and is set to one of the following: 1) pid: PID controller based adaptive time step control as suggested by Turek and Kuzmin[387]. 2) pid+iteration: Use PID and linear iteration numbers to guide the time step. 3) pid+newtoniteration: Use PID and non-linear iterations numbers to guide the time step. 4) Hardcoded: Use time steps supplied by user via the time-step-control-filename parameter.	"pid+newtoniteration"
94	--time-step-control-decay-damping-factor	A real positive value that specifies the decay rate a time step can be decreased by after the number target iterations has been exceeded	1.0
95	--time-step-control-decay-rate	A real positive value that specifies the decay rate a time step size can be decreased by after the number of target iterations has been exceeded	0.75

[387] *Algebraic Flux Correction III. Incompressible Flow Problems. Uni Dortmund, Turek and Kuzmin, January, 2006 (DOI: 10.1007/3-540-27206-2_8)*

OPM Flow 2021-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
96	--time-step-control-file-name	A character string enclosed in quotes that specifies a filename where time steps are specified. The default is the character string timesteps For instance time steps can be generated by the ecl_summary application in libecl as per the following UNIX command line: `path_to_libecl_applications/ecl_summary DECK TIME > filename` Where: DECK is the name of the data deck you want to get the time steps from, TIME tells the application to return the timing for the run, and "filename" is the name of the file the times are piped to.	"timesteps"
97	--time-step-control-growth-damping-factor	A real positive value that specifies the growth rate of the time step increase when the number of target iterations are undercut.	3.2
98	--time-step-control-growth-rate	A real positive value that specifies the growth rate of the time step size when the number of target iterations are undercut.	1.25
99	--time-step-control-target-iterations	A positive integer that defines the number of linear iterations which the time step control scheme should aim for (if applicable).	30
100	--time-step-control-target-newton-iterations	A positive integer that specifies the number of Newton iterations which the time step control scheme should aim for (if applicable).	8
101	--time-step-control-tolerance	A real double precision value that sets the tolerance for PID (only used with the pid and pid+ options defined by the --time-step-control option).	0.1
102	--time-step-verbosity	A positive integer that specifies the "chattiness" during the time integration.	1
103	--tolerance-cnv	A real positive double precision value that specifies the maximum non-linear tolerance error. This is the local convergence tolerance (maximum of local saturation errors).	0.01
104	--tolerance-cnv-relaxed	A real positive value that defines the relaxed local convergence tolerance that applies for iterations after the iterations with the strict tolerance.	1.0

OPM Flow 2021-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
105	--tolerance-mb	A real positive double precision value that sets the maximum mass balance error, that is the tolerated mass balance error relative to total mass present.	1.0 x 10-6
106	--tolerance-pressure-ms-wells	A real positive double precision value that specifies the tolerance for the pressure equations for multi-segment wells.	1000
107	--tolerance-well-control	A real positive double precision value that sets the maximum tolerance for the well control equations.	1.0×10^{-7}
108	--tolerance-wells	A real positive double precision value that defines the maximum non-linear error for the well equations.	1.0×10^{-4}
109	-update-equations-scaling	A Boolean value that switches on (true) or off (false) the updating of the scaling factors for mass balance equations during the simulation.	false
110	--use-gmres	A Boolean value that when set to true OPM Flow will use Generalized Minimal Residual ("GMRES") solver instead of Biconjugate Gradient Stabilized ("BiCGSTAB") as the linear solver within the Newton iterations.	false
111	--use-inner-iterations-ms-wells	A Boolean value that when set to true the simulator will use nested iterations for multi-segment wells.	true
112	--use-inner-iterations-wells	A Boolean value that when set to true the simulator will use nested iterations for standard wells.	false
113	--use-multisegment-well	A Boolean value that when set to true the simulator will use the well model for multi-segment wells instead of the one for single-segment wells.	true
114	--use-update-stabilization	A Boolean value set to true or false that switches on (true) or off (false) the stabilized Newton option. This option may improve convergence for some cases.	true
115	--vtk-write-diffusion-coefficients	A Boolean value that switches on (true) or off (false) the output of the molecular diffusion coefficients to the VTK[388] output files.	false

[388] *Virtualization ToolKit ("VTK") files used to visualize the 3D results from the model using ParaView, an open-source, multi-platform general purpose data analysis and visualization application (https://www.paraview.org/overview/). For the commercial simulator's compatible output files OPM's ResInsight 3D visualization package can be used instead and is specifically tailored for OPM Flow and the commercial simulator.*

OPM Flow 2021-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
116	--vtk-write-effective-diffusion-coefficients	A Boolean value that switches on (true) or off (false) the output of the effective molecular diffusion coefficients for the medium to the VTK output files.	false
117	--vtk-write-tortuosities	A Boolean value that switches on (true) or off (false) the output of the tortuosity for each phase to the VTK output files.	false
118	--zoltan-imbalance-tol	A real positive that defines the tolerable imbalance of the load balancing provided by Zoltan package.	1.1

Notes:

1) Cells colored green in the No. column are new command line parameters for this release. Similarly for the Default column, cells colored green indicate the default value has changed from the previous release.
2) Items shaded in gray are considered to be developer options that should be used with caution as the values associated with these command line parameters are subject to change.
3) VTK Graphics Command Line Parameters[389] set of commands are no longer listed via the basic help command line command:

   ```
   flow --help
   ```

   instead use:

   ```
   flow --help-all
   ```

   to get a list of supported command line Parameters. The –enable-vtk-output option above, if set to "true" will write out the data as stipulated by the compile options. One may need to compile the source code to obtain the desired output.
4) As per all UNIX and LINUX based system the input is case dependent.
5) If a parameter file is used to set one of the above parameters, the leading "--" should not be incorporated in the parameter file.

Table E.5: OPM Flow 2021-04 Command Line Options

[389] *Virtualization ToolKit ("VTK") files used to visualize the 3D results from the model using ParaView, an open-source, multi-platform general purpose data analysis and visualization application (https://www.paraview.org/overview/). For the commercial simulator's compatible output files OPM's ResInsight 3D visualization package can be used instead and is specifically tailored for OPM Flow and the commercial simulator.*

E.6 RUNNING OPM FLOW 2020-10

The command line syntax for running OPM Flow is:

```
flow [OPTIONS] CASENAME
```

and typing the following command from your terminal:

```
flow CASENAME
```

will start the simulator and run the case specified by CASENAME.DATA.

OPM Flow accepts command line options to control various aspects of the simulator's run time behavior, as described in the table below. To give a command line option use "key=value" syntax, with no spaces around the equals sign. It is also possible to put multiple options together in a parameter file. To do so, put one option on each line of the file with ".param" as the extension and pass that filename as a command line parameter to OPM Flow using:

```
flow --parameter-file=CASENAME.PARAM
```

to instruct OPM Flow to read the parameter file.

The available command line options for this release are listed in Table E.6.

OPM Flow 2020-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
	General eWoms/ebos Command Line Parameters		
1	-h or --help	A character string that causes OPM Flow to print a help message that gives a brief description of the available command line parameters.	N/A
2	--bda-device-id	Choose device ID for cusparseSolver or openclSolver, use 'nvidia-smi' or 'clinfo' commands to determine valid IDs.	0
3	--continue-on-convergence-error	A Boolean value set to true or false that that instructs the simulator to continue (true) with a non-converged solution instead of giving up (false) if it encounter a time step size smaller than the minimum time step size.	false
4	--cpr-max-ell-iter	A positive integer that sets the maximum number of Iterations for the elliptic pressure part of the Constrained Pressure Residual ("CPR") solver.	20

OPM Flow 2020-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
5	--cpr-reuse-setup	A positive integer that defines if the CPR solver should re-use the Amg setup. Valid options are: 0 : Recreate the preconditioner for every linear solve. 1 : Recreate once every time step. 2 : Recreate if last linear solve took more than 10 iterations. 3 : Never recreate.	0
6	--dbph-max-rel	A real positive value that sets maximum relative change of the bottom-hole pressure in a single iteration	1.0
7	--dp-max-rel	A real positive double precision value that sets the maximum allowed relative change of pressure per iteration.	0.3
8	--ds-max	A real positive double precision value that sets the maximum allowed change in saturation per iteration.	0.2
9	--dwell-fraction-max	A real positive double precision value that sets the maximum allowed change in well's volume fraction per iteration.	0.2
10	--ecl-deck-file-name	A character string that defines the name of the OPM Flow input file which contains the simulator's ECLIPSE formatted input deck to be simulated.	""
11	--ecl-enable-drift-compensation	A Boolean value set to true or false that enables (true) or disables (false) partial compensation of systematic mass losses via the source term of the next time step.	false
12	--ecl-output-double-precision	A Boolean value set to true or false that switches on (true) or off (false) double precision in restart files. Useful for 'perfect' restarts.	false
13	--ecl-output-interval	An integer value defining the number of report steps that ought to be skipped between two writes of restart files results.	-1
14	--ecl-strict-parsing	A Boolean value set to true or false that switches on (true) or off (false) strict parsing mode for parsing - all errors are collected before the application exists. Useful for debugging initial runs.	false

OPM Flow 2020-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
15	--edge-weights-method	A defined positive integer value that defines the edge-weighing strategy: 0 : For uniform, 1 : for trans, or 2 : for log(trans).	1
16	--enable-adaptive-time-stepping	A Boolean value set to true or false that turns on (true) or off (false) adaptive time stepping. If set to false the report time steps provided by the input deck are used.	true
17	--enable-async-ecl-output	A Boolean value set to true or false that sets the output mode to be asynchronously (true), letting the simulator continue computing the next time step while writing results to restart and summary files in the commercial simulators format.	true
18	--enable-async-vtk-output	A Boolean value set to true or false that sets the output mode to be asynchronously (true), letting the simulator continue computing the next time step while writing results to the VTK output files.	true
19	--enable-dry-run	A Boolean value set to true or false that specifies if the simulation should actually run, (true) or just check the input deck (false). The option is equivalent to activating the NOSIM keyword in the RUNSPEC section of the input deck (see section *5.3.97 NOSIM – Activate the No Simulation Mode for Data File Checking*). The default value of "auto' will use whatever is stipulated in the input deck via the NOSIM keyword.	"auto"
20	--enable-ecl-output	A Boolean value set to true or false that specifies if the binary output files (restart, summary files, etc.) should be written in the commercial simulator's format (true), or OPM Flow's format (false).	true
21	--enable-logging-fallout-warning	A Boolean value set to true or false that sets the developer option to see whether logging was on non-root processors (true). If set to true output will be appended to the *.DBG or *.PRT files.	false
22	--enable-opm-rst-file	A Boolean value set to true or false to write OPM specific data sets to the commercial simulators restart file to enable restart of an OPM Flow runs (true), or not to write the data (false).	true

OPM Flow 2020-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
23	--enable-storage-cache	A Boolean value set to true or false that turns on (true) or off (false) storing previous storage terms and avoid re-calculating them.	true
24	--enable-terminal-output	A Boolean value set to true or false that turns on (true) or off (false) high-level information about the simulation's progress to the terminal	true
25	--enable-tracer-model	A Boolean value set to true or false that turns on (true) or off (false) transport tracer calculations for when tracers have been declared in the input deck. **Experimental - use with caution.**	false
26	--enable-tuning	A Boolean value set to true or false that instructs OPM Flow to read the time stepping parameters from the *TUNING - Numerical Tuning Control* keyword in the input deck, if set to true. Note that only the first record of the TUNING keyword is processed.	false
27	--enable-vtk-output	A Boolean value set to true or false that turns on (true) or off (false) a global switch for writing VTK files.	false
28	--enable-well-operability-check	A Boolean value set to true or false that enables (true) checking of a well's operating status, or disables (false) the checking.	true
29	--enable-write-all-solutions	A Boolean value set to true or false that turns on (true) or off (false) the writing of all solutions to disk instead of only the ones for the report steps.	false
	OPM Flow Specific Command Line Parameters		
30	--flow-linear-solver-verbosity	A positive integer value that defines the output from linear solver: 0 : No extra output, 1 : output per solution iteration, or 2 : output per iteration.	0
31	--flow-newton-max-iterations	A positive integer that defines the maximum number of Newton iterations per time step used by the simulator.	20
32	--flow-newton-min-iterations	A real positive value that sets the minimum number of Newton iterations per time step used by the simulator.	1
	General eWoms/ebos Command Line Parameters		

OPM Flow 2020-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
33	--force-disable-fluid-in-place-output	A Boolean value set to true or false that instructs OPM Flow to not to print the Fluid In-Place report after each report time step (true) or not (false).	false
34	--full-time-step-initially	A Boolean value set to true or false that instructs OPM Flow to always attempt to finish a report step using a single time step (true) or not (false).	false
35	--gpu-mode	A character string that defines which GPU option to use for the linear solver cusparseSolver or openclSolver. The parameter should be set to: 1) "none|" for no use of the GPU solver, 2) "cusparse" to use the cusparseSolver solver, or 3) "opencl' for the openclSolver.	"none"
36	--ignore-keywords	A character string that defines a list of keywords which should be ignored by OPM Flow. The keyword string should be enclosed in quotes and each keyword separated by a colon, that is ':' .	""
37	--ilu-fillin-level	A positive integer value that sets the fill in level for the ILU preconditioner.	0
38	--ilu-redblack	A Boolean value set to true or false that instructs OPM Flow to use red-black partitioning for the ILU preconditioner. (true) or not (false).	false
39	--ilu-relaxation	A real positive double precision value that sets the relaxation factor of the linear solver's ILU preconditioner	0.9
40	--ilu-reorder-spheres	A Boolean value set to true or false specifies OPM Flow to reorder the entries of the matrix in the red-black ILU preconditioner in spheres starting at an edge (true) or not (false) If false the original ordering is preserved in each color. Otherwise try to ensure D4 ordering (in a 2D structured grid, the diagonal elements are consecutive).	false
41	--initial-time-step-in-days	A real double precision value that sets the size of initial time step in days.	1.0
42	--linear-solver-ignore-convergence-failure	A Boolean value set to true or false that if set to true convergences failures in the linear solver are ignored. This option should be used with care, as the results may be unreliable.	false

OPM Flow 2020-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
43	--linear-solver-max-iter	A positive integer value that defines the maximum number of linear iterations.	200
44	--linear-solver-reduction	A real positive double precision value that sets the tolerance for the linear solver. The linear solver convergences when the residual is reduced sufficiently.	0.01
45	--linear-solver-require-full-sparsity-pattern	A Boolean value set to true or false that if set to true the simulator will produce the full sparsity pattern for the linear solver.	false
46	--linear-solver-restart	A positive integer value that sets the number of iterations after which GMRES is restarted.	40
47	--linear-solver	A defined quoted character string that sets the configuration of the solver, valid values are: 1) ilu0 (default), 2) cpr_quasiimpes, 3) cpr_trueimpes, or 4) a file specified by the LinearSolverConfigurationJsonFile parameter. The default is "ilu0". Alternatively one can enter a character string enclosed in quotes that defines the filename of a JSON configuration file for the flexible linear solver system. In this case the file extension should be '.json'.	"ilu0"
48	--matrix-add-well-contributions	A Boolean value set to true or false that if set to true explicitly specifies the influences of wells between cells in the Jacobian and preconditioner matrices.	false
49	--max-inner-iter-ms-wells	A positive integer value that defines the maximum number of inner iterations for multi-segment wells.	100
50	--max-inner-iter-wells	A positive integer value that defines the maximum number of inner iterations for standard wells.	50
51	--max-pressure-change-ms-wells	A real positive value that defines the maximum relative pressure change for a single iteration of the multi-segment well model.	1.0×10^{6}
52	--max-residual-allowed	A real positive value that sets the absolute maximum tolerance for residuals without cutting the time step size.	1.0×10^{7}

OPM Flow 2020-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
53	--max-single-precision-days	A real positive value that set the maximum time step size where single precision floating point arithmetic can be used solving for the linear systems of equations.	20.0
54	--max-strict-iter	A positive integer that sets the maximum number of Newton iterations before relaxed tolerances are used for the CNV convergence criterion.	0
55	--max-welleq-iter	A positive integer that defines the maximum number of iterations to determine the solution to the well equations.	30
56	--milu-variant	A defined character string that specifies which variant of the modified-ILU preconditioner ought to be used. Possible variants are: 1) ILU (default, plain ILU), 2) MILU_1 (lump diagonal with dropped row entries), 3) MILU_2 (lump diagonal with the sum of the absolute values of the dropped row entries), 4) MILU_3 (if diagonal is positive add sum of dropped row entries. Otherwise subtract them), 5) MILU_4 (if diagonal is positive add sum of dropped row entries. Otherwise do nothing. The default is ILU	"ILU"
57	--min-time-step-before-shutting-problematic-wells-in-days	A real positive value that sets the minimum time step size in days for which problematic wells are not shut. Time steps below this value will result in problematic wells being shut-in by the simulator.	0.001
58	--newton-max-relax	A real positive value that sets the maximum relaxation factor of a Newton iteration used by the simulator.	0.5
59	--newton-relaxation-type	A character string that defines the type of relaxation used in Newton's method, default is dampen.	"dampen"
60	--opencl-platform-id	A positive integer that specifies the platform identification ("ID") for the openSolver. Use the "clinfo" command to determine valid IDs.	0

OPM Flow 2020-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
61	--output-dir	A character string that defines the directory to which OPM Flow is to write the commercial simulator compatible output files (restart, summary files etc.). The default value results in the files be written to the same directory as the input file.	""
62	--output-interval	A positive integer that specifies the number of report steps between two consecutive writes of restart data.	1

63	--output-mode	A character string that defines the output to the *.PRT and *.DBG files:
		1) none: No output to the files.
		2) log or false: Output logging information only.
		3) all or true: Output everything.
		For example to just output logging information use:
		`--output-mode=log`
		or
		`--output-mode=false`
			"all"

64	--owner-cells-first	A Boolean value set to true or false that determines if the cells owned by a rank should be ordered before ghost/overlapping the cells (true), or not (false).	true
65	--parameter-file	A character string that defines the name of a parameter file which contains the simulator's set of run-time parameters, as listed in this table.	""
66	--pri-var-oscilation-threshold	A real positive value that defines the threshold value for the primary variable switching conditions after its meaning has switched to hinder oscillations.	1×10^{-5}
67	--print-parameters	A positive integer value that request that the run time parameters be printed at the start of the run: 0 : No output to the files. 1 : Output *.DBG file. 2 : Output to *.DBG and *.PRT files (default).	2
68	--print-properties	A positive integer value that request that the compile time parameters be printed at the start of the run: 0 : No output to the files. 1 : Output *.DBG file. 2 : Output to *.DBG and *.PRT files (default).	2

OPM Flow 2020-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
69	--project-saturations	A Boolean value set to true or false that determines if all the fluid saturations should be scaled to ensure the values are in the interval (0, 1), including runs that use solvents (true), or not (false).	false
70	--regularization-factor-msw	A real positive value that defines the "regularization factor" for multi-segment wells.	1.0
71	--relaxed-flow-tol-inner-iter-msw	A real positive value that sets the relaxation tolerance for the inner iteration for the multi-segment well flow solution	1.0
72	--relaxed-max-pv-fraction	A real positive value that defines The fraction of the pore volume of the reservoir where the volumetric error (CNV) may be violated during strict Newton iterations.	0.03
73	--relaxed-pressure-tol-inner-iter-msw	A real positive value that sets the relaxation tolerance for the inner iteration for the multi-segment well pressure solution.	50000
74	--scale-linear-system	A Boolean value set to true or false that switches on (true) or off (false) the scaling of linear system of equations according to the equation scale and primary variable types.	false
75	--sched-restart	A Boolean value that determines for a restart run if the case should initialize wells and groups from the historical SCHEDULE section (true), or from the well and group data on the restart file (false). Note that the commercial simulator always uses data from the restart file (false).	true
76	--serial-partitioning	A Boolean value that determines if partitioning for parallel runs on a single process (true), or not (false).	false
77	--solve-welleq-initially	A Boolean value set to true or false that switches on (true) or off (false) the solving of the well equations as a pre-processing step. Note that the well equations are always added to the full system and solved until converged.	true

OPM Flow 2020-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
78	--solver-growth-factor	A real positive value that specifies the allowed value a time step can be increased by, subject to the maximum allowable time step size set by the --flow-solver-max-time-step-in-days parameter. For example, if the current time step has converged at 10 days and --flow-solver-growth-factor is set to the default value, then the next time step will be 2.0 * 10 days, that is at 20 days.	2.0
79	--solver-max-growth	A real positive value that specifies the maximum allowed value a time step can be increased by after a report time step. subject to the maximum allowable time step size set by the --flow-solver-max-time-step-in-days parameter.	3.0
80	--solver-max-restarts	A positive integer that defines the number of allowed consecutive restarts (or time step chops) before the simulation is terminated.	10
81	--solver-max-time-step-in-days	A real positive double precision value that specifies the maximum allowed time step size in days.	365
82	--solver-min-time-step	A real positive double precision value that specifies the minimum size of a time step in seconds. If a step cannot converge without getting cut below this step size the simulator will stop.	0.0
83	--solver-restart-factor	A real positive double precision value that sets the time step chop factor of the time step after a convergence failure. For example, if the current non-convergent time step is 30 days and --flow-solver-restart-factor is set to the default value, then the time step will be repeated using 0.33 * 30 days as the time step, that is 9.9 days.	0.33
84	--solver-verbosity	A positive integer that specifies the "chattiness" of the non-linear solver.	1
85	--strict-inner-iter-ms-wells	A positive integer that specifies the number of inner iterations for multi-segment wells with strict tolerance.	40
86	--threads-per-process	A positive integer value that stipulates the maximum number of threads to be instantiated per process ('-1' means 'automatic').	-1

OPM Flow 2020-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
87	--time-step-after-event-in-days	A real double precision value that sets maximum allowed time step after an event; for example, when a well is open or closed etc. The default value of -1 means that events to do effect the time stepping.	-1
88	--time-step-control	A defined character string that defines the time stepping control algorithm and is set to one of the following: 1) pid: PID controller based adaptive time step control as suggested by Turek and Kuzmin[390]. 2) pid+iteration: Use PID and linear iteration numbers to guide the time step. 3) pid+newtoniteration: Use PID and non-linear iterations numbers to guide the time step. 4) Hardcoded: Use time steps supplied by user via the time-step-control-filename parameter.	pid
89	--time-step-control-decay-damping-factor	A real positive value that specifies the decay rate a time step can be decreased by after the number target iterations has been exceeded	1.0
90	--time-step-control-decay-rate	A real positive value that specifies the decay rate a time step size can be decreased by after the number of target iterations has been exceeded	0.75
91	--time-step-control-file-name	A character string enclosed in quotes that specifies a filename where time steps are specified. The default is the character string timesteps For instance time steps can be generated by the ecl_summary application in libecl as per the following UNIX command line: `path_to_libecl_applications/ecl_summary DECK TIME > filename` Where: DECK is the name of the data deck you want to get the time steps from, TIME tells the application to return the timing for the run, and "filename" is the name of the file the times are piped to.	"timesteps"

[390] *Algebraic Flux Correction III. Incompressible Flow Problems. Uni Dortmund, Turek and Kuzmin, January, 2006 (DOI: 10.1007/3-540-27206-2_8)*

OPM Flow 2020-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
92	--time-step-control-growth-damping-factor	A real positive value that specifies the growth rate of the time step increase when the number of target iterations are undercut.	0.833
93	--time-step-control-growth-rate	A real positive value that specifies the growth rate of the time step size when the number of target iterations are undercut.	1.25
94	--time-step-control-target-iterations	A positive integer that defines the number of linear iterations which the time step control scheme should aim for (if applicable).	30
95	--time-step-control-target-newton-iterations	A positive integer that specifies the number of Newton iterations which the time step control scheme should aim for (if applicable).	8
96	--time-step-control-tolerance	A real double precision value that sets the tolerance for PID (only used with the pid and pid+ options defined by the --time-step-control option).	0.1
97	--time-step-verbosity	A positive integer that specifies the "chattiness" during the time integration.	1
98	--tolerance-cnv	A real positive double precision value that specifies the maximum non-linear tolerance error. This is the local convergence tolerance (maximum of local saturation errors).	0.01
99	--tolerance-cnv-relaxed	A real positive value that defines the relaxed local convergence tolerance that applies for iterations after the iterations with the strict tolerance.	1.0
100	--tolerance-mb	A real positive double precision value that sets the maximum mass balance error, that is the tolerated mass balance error relative to total mass present.	1.0 x 10-6
101	--tolerance-pressure-ms-wells	A real positive double precision value that specifies the tolerance for the pressure equations for multi-segment wells.	1000
102	--tolerance-well-control	A real positive double precision value that sets the maximum tolerance for the well control equations.	1.0×10^{-7}
103	--tolerance-wells	A real positive double precision value that defines the maximum non-linear error for the well equations.	1.0×10^{-4}
104	-update-equations-scaling	A Boolean value set to true or false that switches on (true) or off (false) the updating of the scaling factors for mass balance equations during the simulation.	false

OPM Flow 2020-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
105	--use-gmres	A Boolean value set to true or false that when set to true OPM Flow will use Generalized Minimal Residual ("GMRES") solver instead of Biconjugate Gradient Stabilized ("BiCGSTAB") as the linear solver within the Newton iterations.	false
106	--use-inner-iterations-ms-wells	A Boolean value set to true or false that when set to true the simulator will use nested iterations for multi-segment wells.	true
107	--use-inner-iterations-wells	A Boolean value set to true or false that when set to true the simulator will use nested iterations for standard wells.	false
108	--use-multisegment-well	A Boolean value set to true or false that when set to true the simulator will use the well model for multi-segment wells instead of the one for single-segment wells.	false
109	--use-update-stabilization	A Boolean value set to true or false that switches on (true) or off (false) the stabilized Newton option. This option may improve convergence for some cases.	true
	VTK Graphics Command Line Parameters[391] This set of commands are no longer supported from the command line interface but can be assessed when compiling OPM Flow from source. The –enable-vtk-output option above, if set to "true" will write out the data as stipulated by the compile options.		

Notes:

1) Cells colored green in the No. column are new command line parameters for this release.
2) Items shaded in gray are considered to be developer options that should be used with caution as the values associated with these command line parameters are subject to change.
3) As per all UNIX and LINUX based system the input is case dependent.
4) If a parameter file is used to set one of the above parameters, the leading "--" should not be incorporated in the parameter file.

Table E.6: OPM Flow 2020-10 Command Line Options

[391] *Virtualization ToolKit ("VTK") files used to visualize the 3D results from the model using ParaView, an open-source, multi-platform general purpose data analysis and visualization application (https://www.paraview.org/overview/). For the commercial simulator's compatible output files OPM's ResInsight 3D visualization package can be used instead and is specifically tailored for OPM Flow and the commercial simulator.*

E.7 RUNNING OPM FLOW 2020-04

The command line syntax for running OPM Flow is:

```
flow [OPTIONS] CASENAME
```

and typing the following command from your terminal:

```
flow CASENAME
```

will start the simulator and run the case specified by CASENAME.DATA.

OPM Flow accepts command line options to control various aspects of the simulator's run time behavior, as described in the table below. To give a command line option use "key=value" syntax, with no spaces around the equals sign. It is also possible to put multiple options together in a parameter file. To do so, put one option on each line of the file with ".param" as the extension and pass that filename as a command line parameter to OPM Flow using:

```
flow --parameter-file=CASENAME.PARAM
```

to instruct OPM Flow to read the parameter file.

The available command line options for this release are listed in Table E.7.

OPM Flow 2020-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
	General eWoms/ebos Command Line Parameters		
1	-h or --help	A character string that causes OPM Flow to print a help message that gives a brief description of the available command line parameters.	N/A
2	--continue-on-convergence-error	A Boolean value set to true or false that that instructs the simulator to continue (true) with a non-converged solution instead of giving up (false) if it encounter a time step size smaller than the minimum time step size.	false
3	--cpr-ell-solvetype	A positive integer that defines the solver type of the elliptic pressure solver: 0 : bicgstab, 1 : cg, 2 : only amg preconditioner.	0
4	--cpr-max-ell-iter	A positive integer that sets the maximum number of Iterations for the elliptic pressure part of the Constrained Pressure Residual ("CPR") solver.	20
5	--cpr-reuse-setup	A positive integer that defines if the CPR solver should re-use the Amg setup.	0

OPM Flow 2020-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
6	--cpr-solver-verbose	A defined positive integer value that defines the output from CPR solver: 0 : no output 1 : output summary of inner linear solver 2 : output extensive information about inner linear solve, including setup information	0
7	--cpr-use-drs	A Boolean value set to true or false that switches on (true) or off (false) the use of dynamic row sum weights in the CPR solver.	false
8	--dbph-max-rel	A real positive value that sets maximum relative change of the bottom-hole pressure in a single iteration	1.0
9	--dp-max-rel	A real positive double precision value that sets the maximum allowed relative change of pressure per iteration.	0.3
10	--ds-max	A real positive double precision value that sets the maximum allowed change in saturation per iteration.	0.2
11	--dwell-fraction-max	A real positive double precision value that sets the maximum allowed change in well's volume fraction per iteration.	0.2
12	--ecl-deck-file-name	A character string that defines the name of the OPM Flow input file which contains the simulator's ECLIPSE formatted input deck to be simulated.	""
13	--ecl-enable-drift-compensation	A Boolean value set to true or false that enables (true) or disables (false) partial compensation of systematic mass losses via the source term of the next time step.	false
14	--ecl-output-double-precision	A Boolean value set to true or false that switches on (true) or off (false) double precision in restart files. Useful for 'perfect' restarts.	false
15	--ecl-output-interval	An integer value defining the number of report steps that ought to be skipped between two writes of restart files results.	-1
16	--ecl-strict-parsing	A Boolean value set to true or false that switches on (true) or off (false) strict parsing mode for parsing - all errors are collected before the application exists. Useful for debugging initial runs.	false

OPM Flow 2020-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
17	--edge-weights-method	A defined positive integer value that defines the edge-weighing strategy: 0 : for uniform, 1 : for trans, or 2 : for log(trans).	1
18	--enable-adaptive-time-stepping	A Boolean value set to true or false that turns on (true) or off (false) adaptive time stepping. If set to false the report time steps provided by the input deck are used.	true
19	--enable-async-ecl-output	A Boolean value set to true or false that sets the output mode to be asynchronously (true), letting the simulator continue computing the next time step while writing results to restart and summary files in the commercial simulators format.	true
20	--enable-async-vtk-output	A Boolean value set to true or false that sets the output mode to be asynchronously (true), letting the simulator continue computing the next time step while writing results to the VTK output files.	true
21	--enable-dry-run	A Boolean value set to true or false that specifies if the simulation should actually run, (true) or just check the input deck (false). This option is equivalent to activating the NOSIM keyword in the RUNSPEC section of the input deck (see section *5.3.97 NOSIM – Activate the No Simulation Mode for Data File Checking*).	true
22	--enable-ecl-output	A Boolean value set to true or false that specifies if the binary output files (restart, summary files, etc.) should be written in the commercial simulator's format (true), or OPM Flow's format (false).	true
23	--enable-logging-fallout-warning	A Boolean value set to true or false that sets the developer option to see whether logging was on non-root processors (true). If set to true output will be appended to the *.DBG or *.PRT files.	false
24	--enable-opm-rst-file	A Boolean value set to true or false to write OPM specific data sets to the commercial simulators restart file to enable restart of an OPM Flow runs (true), or not to write the data (false).	true
25	--enable-storage-cache	A Boolean value set to true or false that turns on (true) or off (false) storing previous storage terms and avoid re-calculating them.	true

OPM Flow 2020-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
26	--enable-terminal-output	A Boolean value set to true or false that turns on (true) or off (false) high-level information about the simulation's progress to the terminal	true
27	--enable-tracer-mode	A Boolean value set to true or false that turns on (true) or off (false) transport tracer calculations for when tracers have been declared in the input deck. **Experimental - use with caution.**	false
28	--enable-tuning	A Boolean value set to true or false that instructs OPM Flow to read the time stepping parameters from the *TUNING - Numerical Tuning Control* keyword in the input deck, if set to true. Note that only the first record of the TUNING keyword is processed.	false
29	--enable-vtk-output	A Boolean value set to true or false that turns on (true) or off (false) a global switch for writing VTK files.	false
30	--enable-well-operability-check	A Boolean value set to true or false that enables (true) checking of a well's operating status, or disables (false) the checking.	true
31	--enable-write-all-solutions	A Boolean value set to true or false that turns on (true) or off (false) the writing of all solutions to disk instead of only the ones for the report steps.	false
	OPM Flow Specific Command Line Parameters		
32	--flow-linear-solver-verbosity	A positive integer value that defines the output from linear solver: 0 : no extra output 1 : output per solution iteration 2 : output per iteration	0
33	--flow-newton-max-iterations	A positive integer that defines the maximum number of Newton iterations per time step used by the simulator.	20
34	--flow-newton-min-iterations	A real positive value that sets the minimum number of Newton iterations per time step used by the simulator.	1
	General eWoms/ebos Command Line Parameters		
35	--force-disable-fluid-in-place-output	A Boolean value set to true or false that instructs OPM Flow to not to print the Fluid In-Place report after each report time step (true) or not (false).	false

OPM Flow 2020-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
36	--full-time-step-initially	A Boolean value set to true or false that instructs OPM Flow to always attempt to finish a report step using a single time step (true) or not (false).	false
37	--ignore-keywords	A character string that defines a list of keywords which should be ignored by OPM Flow. The keyword string should be enclosed in quotes and each keyword separated by a colon, that is ':' .	""
38	--ilu-fillin-level	A positive integer value that sets the fill in level for the ILU preconditioner.	0
39	--ilu-redblack	A Boolean value set to true or false that instructs OPM Flow to use red-black partitioning for the ILU preconditioner. (true) or not (false).	false
40	--ilu-relaxation	A real positive double precision value that sets the relaxation factor of the linear solver's ILU preconditioner	0.9
41	--ilu-reorder-spheres	A Boolean value set to true or false specifies OPM Flow to reorder the entries of the matrix in the red-black ILU preconditioner in spheres starting at an edge (true) or not (false) If false the original ordering is preserved in each color. Otherwise try to ensure D4 ordering (in a 2D structured grid, the diagonal elements are consecutive).	false
42	--initial-time-step-in-days	A real double precision value that sets the size of initial time step in days.	1.0
43	--linear-solver-configuration	A defined quoted character string that sets the configuration of the solver, valid values are: 1) ilu0 (default), 2) cpr_quasiimpes, 3) cpr_trueimpes, or 4) a file specified by the LinearSolverConfigurationJsonFile parameter. The default is "ilu0".	ilu0
44	--linear-solver-configuration-json-file	A character string enclosed in quotes that defines the filename of a JSON configuration file for the flexible linear solver system.	none

OPM Flow 2020-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
45	--linear-solver-ignore-convergence-failure	A Boolean value set to true or false that if set to true convergences failures in the linear solver are ignored. This option should be used with care, as the results may be unreliable.	false
46	--linear-solver-max-iter	A positive integer value that defines the maximum number of linear iterations.	200
47	--linear-solver-reduction	A real positive double precision value that sets the tolerance for the linear solver. The linear solver convergences when the residual is reduced sufficiently.	0.01
48	--linear-solver-require-full-sparsity-pattern	A Boolean value set to true or false that if set to true the simulator will produce the full sparsity pattern for the linear solver.	false
49	--linear-solver-restart	A positive integer value that sets the number of iterations after which GMRES is restarted.	40
50	--matrix-add-well-contributions	A Boolean value set to true or false that if set to true explicitly specifies the influences of wells between cells in the Jacobian and preconditioner matrices.	false
51	--max-inner-iter-ms-wells	A positive integer value that defines the maximum number of inner iterations for multi-segment wells.	100
52	--max-pressure-change-ms-wells	A real positive value that defines the maximum relative pressure change for a single iteration of the multi-segment well model.	1.0×10^{6}
53	--max-residual-allowed	A real positive value that sets the absolute maximum tolerance for residuals without cutting the time step size.	1.0×10^{7}
54	--max-single-precision-days	A real positive value that set the maximum time step size where single precision floating point arithmetic can be used solving for the linear systems of equations.	20.0
55	--max-strict-iter	A positive integer that sets the maximum number of Newton iterations before relaxed tolerances are used for the CNV convergence criterion.	8
56	--max-welleq-iter	A positive integer that defines the maximum number of iterations to determine the solution to the well equations.	30

OPM Flow 2020-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
57	--milu-variant	A defined character string that specifies which variant of the modified-ILU preconditioner ought to be used. Possible variants are: 1) ILU (default, plain ILU), 2) MILU_1 (lump diagonal with dropped row entries), 3) MILU_2 (lump diagonal with the sum of the absolute values of the dropped row entries), 4) MILU_3 (if diagonal is positive add sum of dropped row entries. Otherwise subtract them), 5) MILU_4 (if diagonal is positive add sum of dropped row entries. Otherwise do nothing. The default is ILU	ILU
58	--min-time-step-before-shutting-problematic-wells-in-days	A real positive value that sets the minimum time step size in days for which problematic wells are not shut. Time steps below this value will result in problematic wells being shut-in by the simulator.	0.25
59	--newton-max-relax	A real positive value that sets the maximum relaxation factor of a Newton iteration used by the simulator.	0.5
60	--newton-relaxation-type	A character string that defines the type of relaxation used in Newton's method, default is dampen.	dampen
61	--output-dir	A character string that defines the directory to which OPM Flow is to write the commercial simulator compatible output files (restart, summary files etc.). The default value results in the files be written to the same directory as the input file.	""
62	--output-interval	A positive integer that specifies the number of report steps between two consecutive writes of restart data.	1

OPM Flow 2020-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**

63	--output-mode	A character string that defines the output to the *.PRT and *.DBG files:
		1) none: No output to the files.
		2) log or false: Output logging information only.
		3) all or true: Output everything.
		For example to just output logging information use:
		`--output-mode=log`
		or
		`--output-mode=false`
			all

64	--owner-cells-first	A Boolean value set to true or false that determines if the cells owned by a rank should be ordered before ghost/overlapping the cells (true), or not (false).	true
65	--parameter-file	A character string that defines the name of a parameter file which contains the simulator's set of run-time parameters, as listed in this table.	""
66	--pri-var-oscilation-threshold	A real positive value that defines the threshold value for the primary variable switching conditions after its meaning has switched to hinder oscillations.	1×10^{-5}
67	--print-parameters	A positive integer value that request that the run time parameters be printed at the start of the run: 0 : No output to the files. 1 : Output *.DBG file 2 : Output to *.DBG and *.PRT files (default)	2
68	--print-properties	A positive integer value that request that the compile time parameters be printed at the start of the run: 0 : No output to the files. 1 : Output *.DBG file 2 : Output to *.DBG and *.PRT files (default)	2
69	--scale-linear-system	A Boolean value set to true or false that switches on (true) or off (false) the scaling of linear system of equations according to the equation scale and primary variable types.	false

OPM Flow 2020-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
70	--sched-restart	A Boolean value that determines for a restart run if the case should initialize wells and groups from the historical SCHEDULE section (true), or from the well and group data on the restart file (false). Note that the commercial simulator always uses data from the restart file (false).	true
71	--solve-welleq-initially	A Boolean value set to true or false that switches on (true) or off (false) the solving of the well equations as a pre-processing step. Note that the well equations are always added to the full system and solved until converged.	true
72	--solver-growth-factor	A real positive value that specifies the allowed value a time step can be increased by, subject to the maximum allowable time step size set by the --flow-solver-max-time-step-in-days parameter. For example, if the current time step has converged at 10 days and --flow-solver-growth-factor is set to the default value, then the next time step will be 2.0 * 10 days, that is at 20 days.	2.0
73	--solver-max-growth	A real positive value that specifies the maximum allowed value a time step can be increased by after a report time step. subject to the maximum allowable time step size set by the --flow-solver-max-time-step-in-days parameter.	3.0
74	--solver-max-restarts	A positive integer that defines the number of allowed consecutive restarts (or time step chops) before the simulation is terminated.	10
75	--solver-max-time-step-in-days	A real positive double precision value that specifies the maximum allowed time step size in days.	365
76	--solver-min-time-step	A real positive double precision value that specifies the minimum size of a time step in seconds. If a step cannot converge without getting cut below this step size the simulator will stop.	0.0

OPM Flow 2020-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
77	--solver-restart-factor	A real positive double precision value that sets the time step chop factor of the time step after a convergence failure. For example, if the current non-convergent time step is 30 days and --flow-solver-restart-factor is set to the default value, then the time step will be repeated using 0.33 * 30 days as the time step, that is 9.9 days.	0.33
78	--solver-verbosity	A positive integer that specifies the "chattiness" of the non-linear solver.	1
79	--system-strategy	A character string that defines the strategy for reformulating and scaling linear system of equations and is set to one of the following: 1) none: No scaling - should not be used with the CPR solver. 2) original: Use weights that are equivalent to no scaling - should not be used with the CPR solver. 3) simple: Form pressure equation as simple sum of conservation equations. 4) quasiimpes: Form pressure equation based on diagonal block. 5) trueimpes: Form pressure equation based on linearization of the accumulation term.	none
80	--threads-per-process	A positive integer value that stipulates the maximum number of threads to be instantiated per process ('-1' means 'automatic').	-1
81	--time-step-after-event-in-days	A real double precision value that sets maximum allowed time step after an event; for example, when a well is open or closed etc. The default value of -1 means that events to do effect the time stepping.	-1

OPM Flow 2020-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
82	--time-step-control	A character string that defines the time stepping control algorithm and is set to one of the following: 1) pid: PID controller based adaptive time step control as suggested by Turek and Kuzmin[392]. 2) pid+iteration: Use PID and linear iteration numbers to guide the time step. 3) pid+newtoniteration: Use PID and non-linear iterations numbers to guide the time step. 4) Hardcoded: Use time steps supplied by user via the time-step-control-filename parameter.	pid
83	--time-step-control-decay-rate	A real positive value that specifies the decay rate a time step can be decreased by after the number of target iterations has been exceeded	0.75
84	--time-step-control-file-name	A character string enclosed in quotes that specifies a filename where time steps are specified. The default is the character string timesteps For instance time steps can be generated by the ecl_summary application in libecl as per the following UNIX command line: `path_to_libecl_applications/ecl_summary DECK TIME > filename` Where: DECK is the name of the data deck you want to get the time steps from, TIME tells the application to return the timing for the run, and "filename" is the name of the file the times are piped to.	timesteps
85	--time-step-control-growth-rate	A real positive value that specifies the growth rate of the time step size of the number of target iterations is undercut.	1.25
86	--time-step-control-target-iterations	A positive integer that defines the number of linear iterations which the time step control scheme should aim for (if applicable).	30
87	--time-step-control-target-newton-iterations	A positive integer that specifies the number of Newton iterations which the time step control scheme should aim for (if applicable).	8

[392] *Algebraic Flux Correction III. Incompressible Flow Problems. Uni Dortmund, Turek and Kuzmin, January, 2006 (DOI: 10.1007/3-540-27206-2_8)*

OPM Flow 2020-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
88	--time-step-control-tolerance	A real double precision value that sets the tolerance for PID (only used with the pid and pid+ options defined by the --time-step-control option).	0.1
89	--time-step-verbosity	A positive integer that specifies the "chattiness" during the time integration.	1
90	--tolerance-cnv	A real positive double precision value that specifies the maximum non-linear tolerance error. This is the local convergence tolerance (maximum of local saturation errors).	0.01
91	--tolerance-cnv-relaxed	A real positive value that defines the relaxed local convergence tolerance that applies for iterations after the iterations with the strict tolerance.	1.0×10^{9}
92	--tolerance-mb	A real positive double precision value that sets the maximum mass balance error, that is the tolerated mass balance error relative to total mass present.	1.0 x 106
93	--tolerance-pressure-ms-wells	A real positive double precision value that specifies the tolerance for the pressure equations for multi-segment wells.	1000
94	--tolerance-well-control	A real positive double precision value that sets the maximum tolerance for the well control equations.	1.0×10^{-7}
95	--tolerance-wells	A real positive double precision value that defines the maximum non-linear error for the well equations.	0.0001
96	-update-equations-scaling	A Boolean value set to true or false that switches on (true) or off (false) the updating of the scaling factors for mass balance equations during the simulation.	false
97	--use-amg	A Boolean value set to true or false that if set to true OPM Flow will use AMG as the linear solver's preconditioner	false
98	--use-cpr	A Boolean value set to true or false that when set to true OPM Flow will use CPR as the linear solver's preconditioner	false
99	--use-gmres	A Boolean value set to true or false that when set to true OPM Flow will use Generalized Minimal Residual (“GMRES”) solver instead of Biconjugate Gradient Stabilized (“BiCGSTAB”) as the linear solver within the Newton iterations.	false

OPM Flow 2020-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
100	--use-gpu	A Boolean value that when set to true will result in OPM Flow using the Graphics Processing Unit ("GPU") cusparseSolver as the linear solver.	false
101	--use-inner-iterations-ms-wells	A Boolean value set to true or false that when set to true the simulator will use nested iterations for multi-segment wells.	true
102	--use-multisegment-well	A Boolean value set to true or false that when set to true the simulator will use the well model for multi-segment wells instead of the one for single-segment wells.	false
103	--use-update-stabilization	A Boolean value set to true or false that switches on (true) or off (false) the stabilized Newton option. This option may improve convergence for some cases.	true
	VTK Graphics Command Line Parameters[393]		
104	--vtk-write-average-molar-masses	A Boolean value set to true or false that turns on (true) or off (false) the output of the average phase mass in the VTK output files.	false
105	--vtk-write-densities	A Boolean value set to true or false that turns on (true) or off (false) the output of the phase densities to the VTK output files	true
106	--vtk-write-dof-index	A Boolean value set to true or false that turns on (true) or off (false) the output of the tracer concentrations to the VTK output files.	false
107	--vtk-write-ecl-tracer-concentration	A Boolean value set to true or false that turns on (true) or off (false) the output of the index of the degrees of freedom to the VTK output files.	false
108	--vtk-write-extrusion-factor	A Boolean value set to true or false that turns on (true) or off (false) the output of the of the extrusion factor of the degrees of freedom to the VTK output files.	false
109	--vtk-write-filter-velocities	A Boolean value set to true or false that turns on (true) or off (false) the output of the filter velocities of the phases in the VTK output files.	false

[393] *Virtualization ToolKit ("VTK") files used to visualize the 3D results from the model using ParaView, an open-source, multi-platform general purpose data analysis and visualization application (https://www.paraview.org/overview/). For the commercial simulator's compatible output files OPM's ResInsight 3D visualization package can be used instead and is specifically tailored for OPM Flow and the commercial simulator.*

OPM Flow 2020-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
110	--vtk-write-fugacities	A Boolean value set to true or false that turns on (true) or off (false) the output of the component fugacities to the VTK output files.	false
111	--vtk-write-fugacity-coeffs	A Boolean value set to true or false that turns on (true) or off (false) the output of the component fugacity coefficients to the VTK output files.	false
112	--vtk-write-gas-dissolution-factor	A Boolean value set to true or false that turns on (true) or off (false) the output of the gas dissolution factor (Rs) of the observed oil to the VTK output files.	false
113	--vtk-write-gas-formation-volume-factor	A Boolean value set to true or false that turns on (true) or off (false) the output of the gas formation volume factor (Bg) to the VTK output files.	false
114	--vtk-write-gas-saturation-pressure	A Boolean value set to true or false that turns on (true) or off (false) the output of the saturation pressure of gas (Pg,sat) to the VTK output files'	false
115	--vtk-write-intrinsic-permeabilities	A Boolean value set to true or false that turns on (true) or off (false) the output of the intrinsic permeability to the VTK output files.	false
116	--vtk-write-mass-fractions	A Boolean value set to true or false that turns on (true) or off (false) the output of the mass fractions to the VTK output files.	false
117	--vtk-write-mobilities	A Boolean value set to true or false that turns on (true) or off (false) the output of the phase mobilities to the VTK output files.	false
118	--vtk-write-molarities	A Boolean value set to true or false that turns on (true) or off (false) the output of the component molarities to the VTK output files.	false
119	--vtk-write-mole-fractions	A Boolean value set to true or false that turns on (true) or off (false) the output of the mole fractions to the VTK output files.	true
120	--vtk-write-oil-formation-volume-factor	A Boolean value set to true or false that turns on (true) or off (false) the output of the oil formation volume factor (Bo) to the VTK output files.	false
121	--vtk-write-oil-saturation-pressure	A Boolean value set to true or false that turns on (true) or off (false) the output of the saturation pressure of oil (Po,sat) to the VTK output files.	false

OPM Flow 2020-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
122	--vtk-write-oil-vaporization-factor	A Boolean value set to true or false that turns on (true) or off (false) the output of the oil vaporization factor (Rv) of the observed gas to the VTK output files.	false
123	--vtk-write-porosity	A Boolean value set to true or false that turns on (true) or off (false) the output of the porosity array to the VTK output files.	true
124	--vtk-write-potential-gradients	A Boolean value set to true or false that turns on (true) or off (false) the output of the phase pressure potential gradients to the VTK output files.	false
125	--vtk-write-pressures	A Boolean value set to true or false that turns on (true) or off (false) the output of the phase pressures to the VTK output files.	true
126	--vtk-write-primary-vars	A Boolean value set to true or false that turns on (true) or off (false) the output of the primary variables to the VTK output files.	false
127	--vtk-write-primary-vars-meaning	A Boolean value set to true or false that turns on (true) or off (false) the output of the how the primary variables should be interpreted to the VTK output files.	false
128	--vtk-write-process-rank	A Boolean value set to true or false that turns on (true) or off (false) the output of the MPI process rank to the VTK output files.	false
129	--vtk-write-relative-permeabilities	A Boolean value set to true or false that turns on (true) or off (false) the output of the phase relative permeabilities to the VTK output files.	true
130	--vtk-write-saturated-gas-oil-vaporization-factor	A Boolean value set to true or false that turns on (true) or off (false) the output of the oil vaporization factor (Rv,sat) of oil saturated gas to the VTK output files.	false
131	--vtk-write-saturated-oil-gas-dissolution-factor	A Boolean value set to true or false that turns on (true) or off (false) the output of the gas dissolution factor (Rs,sat) of gas saturated oil to the VTK output files.	false
132	--vtk-write-saturation-ratios	A Boolean value set to true or false that turns on (true) or off (false) the output of the ratio of the actually and maximum dissolved component of the mixture to the VTK output files	false
133	--vtk-write-saturations	A Boolean value set to true or false that turns on (true) or off (false) the output of the phase saturations to the VTK output files.	true

OPM Flow 2020-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
134	--vtk-write-temperature	A Boolean value set to true or false that turns on (true) or off (false) the output of the temperature array to the VTK output files.	true
135	--vtk-write-total-mass-fractions	A Boolean value set to true or false that turns on (true) or off (false) the output of the total mass fractions to the VTK output files.	false
136	--vtk-write-total-mole-fractions	A Boolean value set to true or false that turns on (true) or off (false) the output of the total mole fractions to the VTK output files.	false
137	--vtk-write-viscosities	A Boolean value set to true or false that turns on (true) or off (false) the output of the component phase viscosities to the VTK output files.	false
138	--vtk-write-water-formation-volume-factor	A Boolean value set to true or false that turns on (true) or off (false) the output of the oil formation volume factor (Bo) to the VTK output files.	false

Notes:

1) Cells colored green in the No. column are new command line parameters for this release.
2) Items shaded in gray are considered to be developer options that should be used with caution as the values associated with these command line parameters are subject to change.
3) As per all UNIX and LINUX based system the input is case dependent.
4) If a parameter file is used to set one of the above parameters, the leading "--" should not be incorporated in the parameter file.

Table E.7: OPM Flow 2020-04 Command Line Options

E.8 RUNNING OPM FLOW 2019-10

The command line syntax for running OPM Flow is:

```
flow [OPTIONS] CASENAME
```

and typing the following command from your terminal:

```
flow CASENAME
```

will start the simulator and run the case specified by CASENAME.DATA.

OPM Flow accepts command line options to control various aspects of the simulator's run time behavior, as described in the table below. To give a command line option use "key=value" syntax, with no spaces around the equals sign. It is also possible to put multiple options together in a parameter file. To do so, put one option on each line of the file with ".param" as the extension and pass that filename as a command line parameter to OPM Flow using:

```
flow --parameter-file=CASENAME.PARAM
```

to instruct OPM Flow to read the parameter file.

The available command line options for this release are listed in Table E.8.

OPM Flow 2019-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
	General eWoms/ebos Command Line Parameters		
1	-h or --help	A character string that causes OPM Flow to print a help message that gives a brief description of the available command line parameters.	N/A
2	--continue-on-convergence-error	A Boolean value set to true or false that that instructs the simulator to continue (true) with a non-converged solution instead of giving up (false) if it encounter a time step size smaller than the minimum time step size.	false
3	--cpr-ell-solvetype	A positive integer that defines the solver type of the elliptic pressure solver: 0 : bicgstab, 1 : cg, 2 : only amg preconditioner	0
4	--cpr-max-ell-iter	A positive integer that sets the maximum number of Iterations for the elliptic pressure part of the Constrained Pressure Residual ("CPR") solver.	20
5	--cpr-reuse-setup	A positive integer that defines if the CPR solver should re-use the Amg setup.	0
6	--cpr-solver-verbose	A defined positive integer value that defines the output from CPR solver: 0 : no output 1 : output summary of inner linear solver 2 : output extensive information about inner linear solve, including setup information	0

OPM Flow 2019-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
7	--cpr-use-drs	A Boolean value set to true or false that switches on (true) or off (false) the use of dynamic row sum weights in the CPR solver.	false
8	--dbph-max-rel	A real positive value that sets maximum relative change of the bottom-hole pressure in a single iteration	1.0
9	--dp-max-rel	A real positive double precision value that sets the maximum allowed relative change of pressure per iteration.	0.3
10	--ds-max	A real positive double precision value that sets the maximum allowed change in saturation per iteration.	0.2
11	--dwell-fraction-max	A real positive double precision value that sets the maximum allowed change in well's volume fraction per iteration.	0.2
12	--ecl-deck-file-name	A character string that defines the name of the OPM Flow input file which contains the simulator's ECLIPSE formatted input deck to be simulated.	""
13	--ecl-enable-drift-compensation	A Boolean value set to true or false that enables (true) or disables (false) partial compensation of systematic mass losses via the source term of the next time step.	false
14	--ecl-output-double-precision	A Boolean value set to true or false that switches on (true) or off (false) double precision in restart files. Useful for 'perfect' restarts.	false
15	--ecl-output-interval	An integer value defining the number of report steps that ought to be skipped between two writes of restart files results.	-1
16	--ecl-strict-parsing	A Boolean value set to true or false that switches on (true) or off (false) strict parsing mode for parsing - all errors are collected before the application exists. Useful for debugging initial runs.	false
17	--edge-weights-method	A defined positive integer value that defines the edge-weighing strategy: 0 : for uniform, 1 : for trans, or 2 : for log(trans).	1
18	--enable-adaptive-time-stepping	A Boolean value set to true or false that turns on (true) or off (false) adaptive time stepping. If set to false the report time steps provided by the input deck are used.	true
19	--enable-async-ecl-output	A Boolean value set to true or false that sets the output mode to be asynchronously (true), letting the simulator continue computing the next time step while writing results to restart and summary files in the commercial simulators format.	true

OPM Flow 2019-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
20	--enable-async-vtk-output	A Boolean value set to true or false that sets the output mode to be asynchronously (true), letting the simulator continue computing the next time step while writing results to the VTK output files.	true
21	--enable-dry-run	A Boolean value set to true or false that specifies if the simulation should actually run, (true) or just check the input deck (false). This option is equivalent to activating the NOSIM keyword in the RUNSPEC section of the input deck (see section *5.3.97 NOSIM – Activate the No Simulation Mode for Data File Checking*).	true
22	--enable-ecl-output	A Boolean value set to true or false that specifies if the binary output files (restart, summary files, etc.) should be written in the commercial simulator's format (true), or OPM Flow's format (false).	true
23	--enable-logging-fallout-warning	A Boolean value set to true or false that sets the developer option to see whether logging was on non-root processors (true). If set to true output will be appended to the *.DBG or *.PRT files.	false
24	--enable-opm-rst-file	A Boolean value set to true or false to write OPM specific data sets to the commercial simulators restart file to enable restart of an OPM Flow runs (true), or not to write the data (false).	true
25	--enable-storage-cache	A Boolean value set to true or false that turns on (true) or off (false) storing previous storage terms and avoid re-calculating them.	true
26	--enable-terminal-output	A Boolean value set to true or false that turns on (true) or off (false) high-level information about the simulation's progress to the terminal	true
27	--enable-tracer-mode	A Boolean value set to true or false that turns on (true) or off (false) transport tracer calculations for when tracers have been declared in the input deck. **Experimental - use with caution.**	false
28	--enable-tuning	A Boolean value set to true or false that instructs OPM Flow to read the time stepping parameters from the *TUNING - Numerical Tuning Control* keyword in the input deck, if set to true. Note that only the first record of the TUNING keyword is processed.	false
29	--enable-vtk-output	A Boolean value set to true or false that turns on (true) or off (false) a global switch for writing VTK files.	false
30	--enable-well-operability-check	A Boolean value set to true or false that enables (true) checking of a well's operating status, or disables (false) the checking.	true

OPM Flow 2019-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
31	--enable-write-all-solutions	A Boolean value set to true or false that turns on (true) or off (false) the writing of all solutions to disk instead of only the ones for the report steps.	false
	OPM Flow Specific Command Line Parameters		
32	--flow-linear-solver-verbosity	A positive integer value that defines the output from linear solver: 0 : no extra output 1 : output per solution iteration 2 : output per iteration	0
33	--flow-newton-max-iterations	A positive integer that defines the maximum number of Newton iterations per time step used by the simulator.	20
34	--flow-newton-min-iterations	A real positive value that sets the minimum number of Newton iterations per time step used by the simulator.	1
	General eWoms/ebos Command Line Parameters		
35	--force-disable-fluid-in-place-output	A Boolean value set to true or false that instructs OPM Flow to not to print the Fluid In-Place report after each report time step (true) or not (false).	false
36	--full-time-step-initially	A Boolean value set to true or false that instructs OPM Flow to always attempt to finish a report step using a single time step (true) or not (false).	false
37	--ignore-keywords	A character string that defines a list of keywords which should be ignored by OPM Flow. The keyword string should be enclosed in quotes and each keyword separated by a colon, that is ':' .	“”
38	--ilu-fillin-level	A positive integer value that sets the fill in level for the ILU preconditioner.	0
39	--ilu-redblack	A Boolean value set to true or false that instructs OPM Flow to use red-black partitioning for the ILU preconditioner. (true) or not (false).	false
40	--ilu-relaxation	A real positive double precision value that sets the relaxation factor of the linear solver's ILU preconditioner	0.9
41	--ilu-reorder-spheres	A Boolean value set to true or false specifies OPM Flow to reorder the entries of the matrix in the red-black ILU preconditioner in spheres starting at an edge (true) or not (false) If false the original ordering is preserved in each color. Otherwise try to ensure D4 ordering (in a 2D structured grid, the diagonal elements are consecutive).	false
42	--initial-time-step-in-days	A real double precision value that sets the size of initial time step in days.	1.0

OPM Flow 2019-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
43	--linear-solver-configuration-json-file	A character string enclosed in quotes that defines the filename of a JSON configuration file for the flexible linear solver system.	none
44	--linear-solver-ignore-convergence-failure	A Boolean value set to true or false that if set to true convergences failures in the linear solver are ignored. This option should be used with care, as the results may be unreliable.	false
45	--linear-solver-max-iter	A positive integer value that defines the maximum number of linear iterations.	200
46	--linear-solver-reduction	A real positive double precision value that sets the tolerance for the linear solver. The linear solver convergences when the residual is reduced sufficiently.	0.01
47	--linear-solver-require-full-sparsity-pattern	A Boolean value set to true or false that if set to true the simulator will produce the full sparsity pattern for the linear solver.	false
48	--linear-solver-restart	A positive integer value that sets the number of iterations after which GMRES is restarted.	40
49	--matrix-add-well-contributions	A Boolean value set to true or false that if set to true explicitly specifies the influences of wells between cells in the Jacobian and preconditioner matrices.	false
50	--max-inner-iter-ms-wells	A positive integer value that defines the maximum number of inner iterations for multi-segment wells.	100
51	--max-pressure-change-ms-wells	A real positive value that defines the maximum relative pressure change for a single iteration of the multi-segment well model.	1.0×10^{6}
52	--max-residual-allowed	A real positive value that sets the absolute maximum tolerance for residuals without cutting the time step size.	1.0×10^{7}
53	--max-single-precision-days	A real positive value that set the maximum time step size where single precision floating point arithmetic can be used solving for the linear systems of equations.	20.0
54	--max-strict-iter	A positive integer that sets the maximum number of Newton iterations before relaxed tolerances are used for the CNV convergence criterion.	8
55	--max-welleq-iter	A positive integer that defines the maximum number of iterations to determine the solution to the well equations.	30

OPM Flow 2019-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
56	--milu-variant	A defined character string that specifies which variant of the modified-ILU preconditioner ought to be used. Possible variants are: 1) ILU (default, plain ILU), 2) MILU_1 (lump diagonal with dropped row entries), 3) MILU_2 (lump diagonal with the sum of the absolute values of the dropped row entries), 4) MILU_3 (if diagonal is positive add sum of dropped row entries. Otherwise subtract them), 5) MILU_4 (if diagonal is positive add sum of dropped row entries. Otherwise do nothing. The default is ILU	ILU
57	--newton-max-relax	A real positive value that sets the maximum relaxation factor of a Newton iteration used by the simulator.	0.5
58	--newton-relaxation-type	A character string that defines the type of relaxation used in Newton's method, default is dampen.	dampen
59	--output-dir	A character string that defines the directory to which OPM Flow is to write the commercial simulator compatible output files (restart, summary files etc.). The default value results in the files be written to the same directory as the input file.	""
60	--output-interval	A positive integer that specifies the number of report steps between two consecutive writes of restart data.	1

61	--output-mode	A character string that defines the output to the *.PRT and *.DBG files:
		1) none: No output to the files.
		2) log or false: Output logging information only.
		3) all or true: Output everything.
		For example to just output logging information use:
		`--output-mode=log`
		or
		`--output-mode=false`
			all

62	--parameter-file	A character string that defines the name of a parameter file which contains the simulator's set of run-time parameters, as listed in this table.	""
63	--pri-var-oscilation-threshold	A real positive value that defines the threshold value for the primary variable switching conditions after its meaning has switched to hinder oscillations.	1×10^{-5}
64	--print-parameters	A positive integer value that request that the run time parameters be printed at the start of the run: 0 : No output to the files. 1 : Output *.DBG file 2 : Output to *.DBG and *.PRT files (default)	2

OPM Flow 2019-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
65	--print-properties	A positive integer value that request that the compile time parameters be printed at the start of the run: 0 : No output to the files. 1 : Output *.DBG file 2 : Output to *.DBG and *.PRT files (default)	2
66	--scale-linear-system	A Boolean value set to true or false that switches on (true) or off (false) the scaling of linear system of equations according to the equation scale and primary variable types.	false
67	--solve-welleq-initially	A Boolean value set to true or false that switches on (true) or off (false) the solving of the well equations as a pre-processing step. Note that the well equations are always added to the full system and solved until converged.	true
68	--solver-growth-factor	A real positive value that specifies the allowed value a time step can be increased by, subject to the maximum allowable time step size set by the --flow-solver-max-time-step-in-days parameter. For example, if the current time step has converged at 10 days and --flow-solver-growth-factor is set to the default value, then the next time step will be 2.0 * 10 days, that is at 20 days.	2.0
69	--solver-max-growth	A real positive value that specifies the maximum allowed value a time step can be increased by after a report time step. subject to the maximum allowable time step size set by the --flow-solver-max-time-step-in-days parameter.	3.0
70	--solver-max-restarts	A positive integer that defines the number of allowed consecutive restarts (or time step chops) before the simulation is terminated.	10
71	--solver-max-time-step-in-days	A real positive double precision value that specifies the maximum allowed time step size in days.	365
72	--solver-restart-factor	A real positive double precision value that sets the time step chop factor of the time step after a convergence failure. For example, if the current non-convergent time step is 30 days and --flow-solver-restart-factor is set to the default value, then the time step will be repeated using 0.33 * 30 days as the time step, that is 9.9 days.	0.33
73	--solver-verbosity	A positive integer that specifies the "chattiness" of the non-linear solver.	1

OPM Flow 2019-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
74	--system-strategy	A character string that defines the strategy for reformulating and scaling linear system of equations and is set to one of the following: 1) none: No scaling - should not be used with the CPR solver. 2) original: Use weights that are equivalent to no scaling - should not be used with the CPR solver. 3) simple: Form pressure equation as simple sum of conservation equations. 4) quasiimpes: Form pressure equation based on diagonal block. 5) trueimpes: Form pressure equation based on linearization of the accumulation term.	none
75	--threads-per-process	A positive integer value that stipulates the maximum number of threads to be instantiated per process ('-1' means 'automatic').	-1
76	--time-step-after-event-in-days	A real double precision value that sets maximum allowed time step after an event; for example, when a well is open or closed etc. The default value of -1 means that events to do effect the time stepping.	-1
77	--time-step-control	A character string that defines the time stepping control algorithm and is set to one of the following: 1) pid: PID controller based adaptive time step control as suggested by Turek and Kuzmin[394]. 2) pid+iteration: Use PID and linear iteration numbers to guide the time step. 3) pid+newtoniteration: Use PID and non-linear iterations numbers to guide the time step. 4) Hardcoded: Use time steps supplied by user via the time-step-control-filename parameter.	pid
78	--time-step-control-decay-rate	A real positive value that specifies the decay rate a time step can be decreased by after the number of target iterations has been exceeded	0.75

[394] *Algebraic Flux Correction III. Incompressible Flow Problems. Uni Dortmund, Turek and Kuzmin, January, 2006 (DOI: 10.1007/3-540-27206-2_8)*

OPM Flow 2019-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
79	--time-step-control-file-name	A character string enclosed in quotes that specifies a filename where time steps are specified. The default is the character string timesteps For instance time steps can be generated by the ecl_summary application in libecl as per the following UNIX command line: `path_to_libecl_applications/ ecl_summary DECK TIME > filename` Where: DECK is the name of the data deck you want to get the time steps from, TIME tells the application to return the timing for the run, and "filename" is the name of the file the times are piped to.	timesteps
80	--time-step-control-growth-rate	A real positive value that specifies the growth rate of the time step size of the number of target iterations is undercut.	1.25
81	--time-step-control-target-iterations	A positive integer that defines the number of linear iterations which the time step control scheme should aim for (if applicable).	30
82	--time-step-control-target-newton-iterations	A positive integer that specifies the number of Newton iterations which the time step control scheme should aim for (if applicable).	8
83	--time-step-control-tolerance	A real double precision value that sets the tolerance for PID (only used with the pid and pid+ options defined by the --time-step-control option).	0.1
84	--time-step-verbosity	A positive integer that specifies the "chattiness" during the time integration.	1
85	--tolerance-cnv	A real positive double precision value that specifies the maximum non-linear tolerance error. This is the local convergence tolerance (maximum of local saturation errors).	0.01
86	--tolerance-cnv-relaxed	A real positive value that defines the relaxed local convergence tolerance that applies for iterations after the iterations with the strict tolerance.	1.0×10^{9}
87	--tolerance-mb	A real positive double precision value that sets the maximum mass balance error, that is the tolerated mass balance error relative to total mass present.	1.0 x 106
88	--tolerance-pressure-ms-wells	A real positive double precision value that specifies the tolerance for the pressure equations for multi-segment wells.	1000
89	--tolerance-well-control	A real positive double precision value that sets the maximum tolerance for the well control equations.	1.0×10^{-7}
90	--tolerance-wells	A real positive double precision value that defines the maximum non-linear error for the well equations.	0.0001

OPM Flow 2019-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
91	-update-equations-scaling	A Boolean value set to true or false that switches on (true) or off (false) the updating of the scaling factors for mass balance equations during the simulation.	false
92	--use-amg	A Boolean value set to true or false that if set to true OPM Flow will use AMG as the linear solver's preconditioner	false
93	--use-cpr	A Boolean value set to true or false that when set to true OPM Flow will use CPR as the linear solver's preconditioner	false
94	--use-gmres	A Boolean value set to true or false that when set to true OPM Flow will use Generalized Minimal Residual ("GMRES") solver instead of Biconjugate Gradient Stabilized ("BiCGSTAB") as the linear solver within the Newton iterations.	false
95	--use-inner-iterations-ms-wells	A Boolean value set to true or false that when set to true the simulator will use nested iterations for multi-segment wells.	true
96	--use-multisegment-well	A Boolean value set to true or false that when set to true the simulator will use the well model for multi-segment wells instead of the one for single-segment wells.	false
97	--use-update-stabilization	A Boolean value set to true or false that switches on (true) or off (false) the stabilized Newton option. This option may improve convergence for some cases.	true
	VTK Graphics Command Line Parameters[395]		
98	--vtk-write-average-molar-masses	A Boolean value set to true or false that turns on (true) or off (false) the output of the average phase mass in the VTK output files.	false
99	--vtk-write-densities	A Boolean value set to true or false that turns on (true) or off (false) the output of the phase densities to the VTK output files	true
100	--vtk-write-dof-index	A Boolean value set to true or false that turns on (true) or off (false) the output of the tracer concentrations to the VTK output files.	false
101	--vtk-write-ecl-tracer-concentration	A Boolean value set to true or false that turns on (true) or off (false) the output of the index of the degrees of freedom to the VTK output files.	false
102	--vtk-write-extrusion-factor	A Boolean value set to true or false that turns on (true) or off (false) the output of the of the extrusion factor of the degrees of freedom to the VTK output files.	false

[395] *Virtualization ToolKit ("VTK") files used to visualize the 3D results from the model using ParaView, an open-source, multi-platform general purpose data analysis and visualization application (https://www.paraview.org/overview/). For the commercial simulator's compatible output files OPM's ResInsight 3D visualization package can be used instead and is specifically tailored for OPM Flow and the commercial simulator.*

OPM Flow 2019-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
103	--vtk-write-filter-velocities	A Boolean value set to true or false that turns on (true) or off (false) the output of the filter velocities of the phases in the VTK output files.	false
104	--vtk-write-fugacities	A Boolean value set to true or false that turns on (true) or off (false) the output of the component fugacities to the VTK output files.	false
105	--vtk-write-fugacity-coeffs	A Boolean value set to true or false that turns on (true) or off (false) the output of the component fugacity coefficients to the VTK output files.	false
106	--vtk-write-gas-dissolution-factor	A Boolean value set to true or false that turns on (true) or off (false) the output of the gas dissolution factor (Rs) of the observed oil to the VTK output files.	false
107	--vtk-write-gas-formation-volume-factor	A Boolean value set to true or false that turns on (true) or off (false) the output of the gas formation volume factor (Bg) to the VTK output files.	false
108	--vtk-write-gas-saturation-pressure	A Boolean value set to true or false that turns on (true) or off (false) the output of the saturation pressure of gas (Pg,sat) to the VTK output files'	false
109	--vtk-write-intrinsic-permeabilities	A Boolean value set to true or false that turns on (true) or off (false) the output of the intrinsic permeability to the VTK output files.	false
110	--vtk-write-mass-fractions	A Boolean value set to true or false that turns on (true) or off (false) the output of the mass fractions to the VTK output files.	false
111	--vtk-write-mobilities	A Boolean value set to true or false that turns on (true) or off (false) the output of the phase mobilities to the VTK output files.	false
112	--vtk-write-molarities	A Boolean value set to true or false that turns on (true) or off (false) the output of the component molarities to the VTK output files.	false
113	--vtk-write-mole-fractions	A Boolean value set to true or false that turns on (true) or off (false) the output of the mole fractions to the VTK output files.	true
114	--vtk-write-oil-formation-volume-factor	A Boolean value set to true or false that turns on (true) or off (false) the output of the oil formation volume factor (Bo) to the VTK output files.	false
115	--vtk-write-oil-saturation-pressure	A Boolean value set to true or false that turns on (true) or off (false) the output of the saturation pressure of oil (Po,sat) to the VTK output files.	false
116	--vtk-write-oil-vaporization-factor	A Boolean value set to true or false that turns on (true) or off (false) the output of the oil vaporization factor (Rv) of the observed gas to the VTK output files.	false

OPM Flow 2019-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
117	--vtk-write-porosity	A Boolean value set to true or false that turns on (true) or off (false) the output of the porosity array to the VTK output files.	true
118	--vtk-write-potential-gradients	A Boolean value set to true or false that turns on (true) or off (false) the output of the phase pressure potential gradients to the VTK output files.	false
119	--vtk-write-pressures	A Boolean value set to true or false that turns on (true) or off (false) the output of the phase pressures to the VTK output files.	true
120	--vtk-write-primary-vars	A Boolean value set to true or false that turns on (true) or off (false) the output of the primary variables to the VTK output files.	false
121	--vtk-write-primary-vars-meaning	A Boolean value set to true or false that turns on (true) or off (false) the output of the how the primary variables should be interpreted to the VTK output files.	false
122	--vtk-write-process-rank	A Boolean value set to true or false that turns on (true) or off (false) the output of the MPI process rank to the VTK output files.	false
123	--vtk-write-relative-permeabilities	A Boolean value set to true or false that turns on (true) or off (false) the output of the phase relative permeabilities to the VTK output files.	true
124	--vtk-write-saturated-gas-oil-vaporization-factor	A Boolean value set to true or false that turns on (true) or off (false) the output of the oil vaporization factor (Rv,sat) of oil saturated gas to the VTK output files.	false
125	--vtk-write-saturated-oil-gas-dissolution-factor	A Boolean value set to true or false that turns on (true) or off (false) the output of the gas dissolution factor (Rs,sat) of gas saturated oil to the VTK output files.	false
126	--vtk-write-saturation-ratios	A Boolean value set to true or false that turns on (true) or off (false) the output of the ratio of the actually and maximum dissolved component of the mixture to the VTK output files	false
127	--vtk-write-saturations	A Boolean value set to true or false that turns on (true) or off (false) the output of the phase saturations to the VTK output files.	true
128	--vtk-write-temperature	A Boolean value set to true or false that turns on (true) or off (false) the output of the temperature array to the VTK output files.	true
129	--vtk-write-total-mass-fractions	A Boolean value set to true or false that turns on (true) or off (false) the output of the total mass fractions to the VTK output files.	false
130	--vtk-write-total-mole-fractions	A Boolean value set to true or false that turns on (true) or off (false) the output of the total mole fractions to the VTK output files.	false

OPM Flow 2019-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
131	--vtk-write-viscosities	A Boolean value set to true or false that turns on (true) or off (false) the output of the component phase viscosities to the VTK output files.	false
132	--vtk-write-water-formation-volume-factor	A Boolean value set to true or false that turns on (true) or off (false) the output of the oil formation volume factor (Bo) to the VTK output files.	false

Notes:

1) Cells colored green in the No. column are new command line parameters for this release.
2) Items shaded in gray are considered to be developer options that should be used with caution as the values associated with these command line parameters are subject to change.
3) The --preconditioner-add-well-contributions parameter option that switches on (true) or off (false) the influences of wells between cells for the preconditioner matrix only, has been retired in the 2019-10 release.
4) As per all UNIX and LINUX based system the input is case dependent.
5) If a parameter file is used to set one of the above parameters, the leading "--" should not be incorporated in the parameter file.

Table E.8: OPM Flow 2019-10 Command Line Options

E.9 RUNNING OPM FLOW 2019-04

The command line syntax for running OPM Flow is:

```
flow [OPTIONS] CASENAME
```

and typing the following command from your terminal:

```
flow CASENAME
```

will start the simulator and run the case specified by CASENAME.DATA.

OPM Flow accepts command line options to control various aspects of the simulator's run time behavior, as described in the table below. To give a command line option use "key=value" syntax, with no spaces around the equals sign. It is also possible to put multiple options together in a parameter file. To do so, put one option on each line of the file with ".param" as the extension and pass that filename as a command line parameter to OPM Flow using:

```
flow --parameter-file=CASENAME.PARAM
```

to instruct OPM Flow to read the parameter file.

The available command line options for this release are listed in Table E.9.

OPM Flow 2019-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
	General eWoms/ebos Command Line Parameters		
1	-h or --help	A character string that causes OPM Flow to print a help message that gives a brief description of the available command line parameters.	N/A
2	--cpr-ell-solvetype	A positive integer that defines the solver type of the elliptic pressure solver: 0 : bicgstab, 1 : cg, or 2 : only amg preconditioner.	0
3	--cpr-max-ell-iter	A positive integer that sets the maximum number of Iterations for the elliptic pressure part of the Constrained Pressure Residual ("CPR") solver.	20
4	--cpr-reuse-setup	A positive integer that that defines if the CPR solver should re-use the Amg setup.	0
5	--cpr-solver-verbose	A positive integer value that defines the output from CPR solver: 0 : no output 1 : output summary of inner linear solver 2 : output extensive information about inner linear solve, including setup information	0
6	--cpr-use-drs	A Boolean value set to true or false that switches on (true) or off (false) the use of dynamic row sum weights in the CPR solver.	false

OPM Flow 2019-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
7	--dbph-max-rel	A real positive value that sets maximum relative change of the bottom-hole pressure in a single iteration	1.0
8	--dp-max-rel	A real positive double precision value that sets the maximum allowed relative change of pressure per iteration.	0.3
9	--ds-max	A real positive double precision value that sets the maximum allowed change in saturation per iteration.	0.2
10	--dwell-fraction-max	A real positive double precision value that sets the maximum allowed change in well's volume fraction per iteration.	0.2
11	--ecl-deck-file-name	A character string that defines the name of the OPM Flow input file which contains the simulator's ECLIPSE formatted input deck to be simulated.	""
12	--ecl-output-double-precision	A Boolean value set to true or false that switches on (true) or off (false) double precision in restart files. Useful for 'perfect' restarts.	false
13	--ecl-output-interval	An integer value defining the number of report steps that ought to be skipped between two writes of restart files results.	-1
14	--ecl-strict-parsing	A Boolean value set to true or false that switches on (true) or off (false) strict parsing mode for parsing - all errors are collected before the application exists. Useful for debugging initial runs.	false
15	--enable-adaptive-time-stepping	A Boolean value set to true or false that that turns on (true) or off (false) adaptive time stepping. If set to false the report time steps provided by the input deck are used.	true
16	--enable-async-ecl-output	A Boolean value set to true or false that sets the output mode to be asynchronously (true), letting the simulator continue computing the next time step while writing results to restart and summary files in the commercial simulators format.	true
17	--enable-async-vtk-output	A Boolean value set to true or false that sets the output mode to be asynchronously (true), letting the simulator continue computing the next time step while writing results to the VTK output files.	true
18	--enable-dry-run	A Boolean value set to true or false that specifies if the simulation should actually run, (true) or just check the input deck (false). This option is equivalent to activating the NOSIM keyword in the RUNSPEC section of the input deck (see section *5.3.97 NOSIM – Activate the No Simulation Mode for Data File Checking*).	true

OPM Flow 2019-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
19	--enable-ecl-output	A Boolean value set to true or false that specifies if the binary output files (restart, summary files, etc.) should be written in the commercial simulator's format (true), or OPM Flow's format (false).	true
20	--enable-logging-fallout-warning	A Boolean value set to true or false that sets the developer option to see whether logging was on non-root processors (true). If set to true output will be appended to the *.DBG or *.PRT files.	false
21	--enable-opm-rst-file	A Boolean value set to true or false to write OPM specific data sets to the commercial simulators restart file to enable restart of an OPM Flow runs (true), or not to write the data (false).	true
22	--enable-storage-cache	A Boolean value set to true or false that turns on (true) or off (false) storing previous storage terms and avoid re-calculating them.	true
23	--enable-terminal-output	A Boolean value set to true or false that turns on (true) or off (false) high-level information about the simulation's progress to the terminal	true
24	--enable-tracer-mode	A Boolean value set to true or false that turns on (true) or off (false) transport tracer calculations for when tracers have been declared in the input deck. **Experimental - use with caution.**	false
25	--enable-tuning	A Boolean value set to true or false that that instructs OPM Flow to read the time stepping parameters from the *TUNING - Numerical Tuning Control* keyword in the input deck, if set to true. Note that only the first record of the TUNING keyword is processed.	false
26	--enable-vtk-output	A Boolean value set to true or false that turns on (true) or off (false) a global switch for writing VTK files.	false
27	--enable-write-all-solutions	A Boolean value set to true or false that turns on (true) or off (false) the writing of all solutions to disk instead of only the ones for the report steps.	false
	OPM Flow Specific Command Line Parameters		
28	--flow-linear-solver-verbosity	A positive integer value that defines the output from linear solver: 0 : no extra output 1 : output per solution iteration 2 : output per iteration	0
29	--flow-newton-max-iterations	A positive integer that defines the maximum number of Newton iterations per time step used by the simulator.	20

OPM Flow 2019-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
30	--flow-newton-min-iterations	A real positive value that sets the minimum number of Newton iterations per time step used by the simulator.	1
	General eWoms/ebos Command Line Parameters		
31	--force-disable-fluid-in-place-output	A Boolean value set to true or false that instructs OPM Flow to not to print the Fluid In-Place report after each report time step (true) or not (false).	false
32	--full-time-step-initially	A Boolean value set to true or false that instructs OPM Flow to always attempt to finish a report step using a single time step (true) or not (false).	false
33	--ignore-keywords	A character string that defines a list of keywords which should be ignored by OPM Flow. The keyword string should be enclosed in quotes and each keyword separated by a colon, that is ':' .	""
34	--ilu-fillin-level	A positive integer value that sets the fill in level for the ILU preconditioner.	0
35	--ilu-redblack	A Boolean value set to true or false that instructs OPM Flow to use red-black partitioning for the ILU preconditioner. (true) or not (false).	false
36	--ilu-relaxation	A real positive double precision value that sets the relaxation factor of the linear solver's ILU preconditioner	0.9
37	--ilu-reorder-spheres	A Boolean value set to true or false specifies OPM Flow to reorder the entries of the matrix in the red-black ILU preconditioner in spheres starting at an edge (true) or not (false) If false the original ordering is preserved in each color. Otherwise try to ensure D4 ordering (in a 2D structured grid, the diagonal elements are consecutive).	false
38	--initial-time-step-in-days	A real double precision value that sets the size of initial time step in days.	1.0
39	--linear-solver-ignore-convergence-failure	A Boolean value set to true or false that if set to true convergences failures in the linear solver are ignored. This option should be used with care, as the results may be unreliable.	false
40	--linear-solver-max-iter	A positive integer value that defines the maximum number of linear iterations.	200
41	--linear-solver-reduction	A real positive double precision value that sets the tolerance for the linear solver. The linear solver convergences when the residual is reduced sufficiently.	0.01
42	--linear-solver-require-full-sparsity-pattern	A Boolean value set to true or false that if set to true the simulator will produce the full sparsity pattern for the linear solver.	false

OPM Flow 2019-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
43	--linear-solver-restart	A positive integer value that sets the number of iterations after which GMRES is restarted.	40
44	--matrix-add-well-contributions	A Boolean value set to true or false that if set to true explicitly specifies the influences of wells between cells in the Jacobian and preconditioner matrices.	false
45	--max-inner-iter-ms-wells	A positive integer value that defines the maximum number of inner iterations for multi-segment wells.	10
46	--max-pressure-change-ms-wells	A real positive value that defines the maximum relative pressure change for a single iteration of the multi-segment well model.	200000
47	--max-residual-allowed	A real positive value that sets the absolute maximum tolerance for residuals without cutting the time step size.	1.0×10^7
48	--max-single-precision-days	A real positive value that set the maximum time step size where single precision floating point arithmetic can be used solving for the linear systems of equations.	20.0
49	--max-strict-iter	A positive integer that sets the maximum number of Newton iterations before relaxed tolerances are used for the CNV convergence criterion.	8
50	--max-welleq-iter	A positive integer that defines the maximum number of iterations to determine the solution to the well equations.	30
51	--milu-variant	A defined character string that specifies which variant of the modified-ILU preconditioner ought to be used. Possible variants are: 1) ILU (default, plain ILU), 2) MILU_1 (lump diagonal with dropped row entries), 3) MILU_2 (lump diagonal with the sum of the absolute values of the dropped row entries), 4) MILU_3 (if diagonal is positive add sum of dropped row entrires. Otherwise substract them), 5) MILU_4 (if diagonal is positive add sum of dropped row entrires. Otherwise do nothing. The default is ILU	ILU
52	--newton-max-relax	A real positive value that sets the maximum relaxation factor of a Newton iteration used by the simulator.	0.5
53	--newton-relaxation-type	A character string that defines the type of relaxation used in Newton's method., default is dampen.	dampen
54	--output-dir	A character string that defines the directory to which OPM Flow is to write the commercial simulator compatible output files (restart, summary files etc.). The default value results in the files be written to the same directory as the input file.	""

OPM Flow 2019-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
55	--output-interval	A positive integer that specifies the number of report steps between two consecutive writes of restart data.	1
56	--output-mode	A character string that defines the output to *.PRT and *.DBG files: 1) none: No output to the files. 2) log or false: Output logging information only. 3) all or true: Output everything. For example to just output logging information use: `--output-mode=log or --output-mode=false`	all
57	--parameter-file	A character string that defines the name of a parameter file which contains the simulator's set of run-time parameters, as listed in this table.	""
58	--preconditioner-add-well-contributions	A Boolean value set to true or false that switches on (true) or off (false) the influences of wells between cells for the preconditioner matrix only. **Note this parameter is missing from the help when executing:** `flow -h`	false
59	--pri-var-oscilation-threshold	A real positive value that defines the threshold value for the primary variable switching conditions after its meaning has switched to hinder oscillations.	1×10^{-5}
60	--print-parameters	A positive integer value that request that the run time parameters be printed at the start of the run: 0 : No output to the files. 1 : Output *.DBG file 2 : Output to *.DBG and *.PRT files (default)	2
61	--print-properties	A positive integer value that request that the compile time parameters be printed at the start of the run: 0 : No output to the files. 1 : Output *.DBG file 2 : Output to *.DBG and *.PRT files (default)	2
62	--scale-linear-system	A Boolean value set to true or false that switches on (true) or off (false) the scaling of linear system of equations according to the equation scale and primary variable types.	false
63	--solve-welleq-initially	A Boolean value set to true or false that switches on (true) or off (false) the solving of the well equations as a pre-processing step. Note that the well equations are always added to the full system and solved until converged.	true

OPM Flow 2019-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
64	--solver-growth-factor	A real positive value that specifies the allowed value a time step can be increased by, subject to the maximum allowable time step size set by the --flow-solver-max-time-step-in-days parameter. For example, if the current time step has converged at 10 days and --flow-solver-growth-factor is set to the default value, then the next time step will be 2.0 * 10 days, that is at 20 days.	2.0
65	--solver-max-growth	A real positive value that specifies the maximum allowed value a time step can be increased by after a report time step. subject to the maximum allowable time step size set by the --flow-solver-max-time-step-in-days parameter.	3.0
66	--solver-max-restarts	A positive integer that defines the number of allowed consecutive restarts (or time step chops) before the simulation is terminated.	10
67	--solver-max-time-step-in-days	A real positive double precision value that specifies the maximum allowed time step size in days.	365
68	--solver-restart-factor	A real positive double precision value that sets the time step chop factor of the time step after a convergence failure. For example, if the current non-convergent time step is 30 days and --flow-solver-restart-factor is set to the default value, then the time step will be repeated using 0.33 * 30 days as the time step, that is 9.9 days.	0.33
69	--solver-verbosity	A positive integer that specifies the "chattiness" of the non-linear solver.	1
70	--system-strategy	A character string that defines the strategy for reformulating and scaling linear system of equations and is set to one of the following: 1) none: No scaling - should not be used with the CPR solver. 2) original: Use weights that are equivalent to no scaling - should not be used with the CPR solver. 3) simple: Form pressure equation as simple sum of conservation equations. 4) quasiimpes: Form pressure equation based on diagonal block. 5) trueimpes: Form pressure equation based on linearization of the accumulation term.	none
71	--threads-per-process	A positive integer value that stipulates the maximum number of threads to be instantiated per process ('-1' means 'automatic').	1

OPM Flow 2019-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
72	--time-step-after-event-in-days	A real double precision value that sets maximum allowed time step after an event; for example, when a well is open or closed etc. The default value of -1 means that events to do effect the time stepping.	-1
73	--time-step-control	A character string that defines the time stepping control algorithm and is set to one of the following: 1) pid: PID controller based adaptive time step control as suggested by Turek and Kuzmin[396]. 2) pid+iteration: Use PID and linear iteration numbers to guide the time step. 3) pid+newtoniteration: Use PID and non-linear iterations numbers to guide the time step. 4) Hardcoded: Use time steps supplied by user. Via timestep.control.filename	pid
74	--time-step-control-decay-rate	A real positive value that specifies the decay rate a time step can be decreased by after the number of target iterations has been exceeded	0.75
75	--time-step-control-file-name	A character string that specifies a filename where time steps are specified. The default is the character string timesteps For instance time steps can be generated by the ecl_summary application in libecl as per the following UNIX command line: `path_to_libecl_applications/ ecl_summary DECK TIME > filename` Where: DECK is the name of the data deck you want to get the time steps from, TIME tells the application to return the timing for the run, and "filename" is the name of the file the times are piped to.	timesteps
76	--time-step-control-growth-rate	A real positive value that specifies the growth rate of the time step size of the number of target iterations is undercut.	1.25
77	--time-step-control-target-iterations	A positive integer that defines the number of linear iterations which the time step control scheme should aim for (if applicable).	30
78	--time-step-control-target-newton-iterations	A positive integer that specifies the number of Newton iterations which the time step control scheme should aim for (if applicable).	8
79	--time-step-control-tolerance	A real double precision value that sets the tolerance for PID (only used with the pid and pid+ options defined by the --time-step-control option).	0.1

[396] *Algebraic Flux Correction III. Incompressible Flow Problems. Uni Dortmund, Turek and Kuzmin, January, 2006 (DOI: 10.1007/3-540-27206-2_8)*

OPM Flow 2019-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
80	--time-step-verbosity	A positive integer that specifies the "chattiness" during the time integration.	1
81	--tolerance-cnv	A real positive double precision value that specifies the maximum non-linear tolerance error. This is the local convergence tolerance (maximum of local saturation errors).	0.01
82	--tolerance-cnv-relaxed	A real positive value that defines the relaxed local convergence tolerance that applies for iterations after the iterations with the strict tolerance.	1.0×10^{9}
83	--tolerance-mb	A real positive double precision value that sets the maximum mass balance error, that is the tolerated mass balance error relative to total mass present.	1.0 x 106
84	--tolerance-pressure-ms-wells	A real positive double precision value that specifies the tolerance for the pressure equations for multi-segment wells.	1000
85	--tolerance-well-control	A real positive double precision value that sets the maximum tolerance for the well control equations.	1.0×10^{-7}
86	--tolerance-wells	A real positive double precision value that defines the maximum non-linear error for the well equations.	0.0001
87	-update-equations-scaling	A Boolean value set to true or false that switches on (true) or off (false) the updating of the scaling factors for mass balance equations during the simulation.	false
88	--use-amg	A Boolean value set to true or false that if set to true OPM Flow will use AMG as the linear solver's preconditioner	false
89	--use-cpr	A Boolean value set to true or false that when set to true OPM Flow will use CPR as the linear solver's preconditioner	false
90	--use-gmres	A Boolean value set to true or false that when set to true OPM Flow will use Generalized Minimal Residual ("GMRES") solver instead of Biconjugate Gradient Stabilized ("BiCGSTAB") as the linear solver within the Newton iterations.	false
91	--use-inner-iterations-ms-wells	A Boolean value set to true or false that when set to true the simulator will use nested iterations for multi-segment wells.	true
92	--use-multisegment-well	A Boolean value set to true or false that when set to true the simulator will use the well model for multi-segment wells instead of the one for single-segment wells.	false
93	--use-update-stabilization	A Boolean value set to true or false that switches on (true) or off (false) the stabilized Newton option. This option may improve convergence for some cases.	true
	VTK Graphics Command Line Parameters[397]		

OPM Flow 2019-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
94	--vtk-write-average-molar-masses	A Boolean value set to true or false that turns on (true) or off (false) the output of the average phase mass in the VTK output files.	false
95	--vtk-write-densities	A Boolean value set to true or false that turns on (true) or off (false) the output of the phase densities to the VTK output files	true
96	--vtk-write-dof-index	A Boolean value set to true or false that turns on (true) or off (false) the output of the tracer concentrations to the VTK output files.	false
97	--vtk-write-ecl-tracer-concentration	A Boolean value set to true or false that turns on (true) or off (false) the output of the index of the degrees of freedom to the VTK output files.	false
98	--vtk-write-extrusion-factor	A Boolean value set to true or false that turns on (true) or off (false) the output of the of the extrusion factor of the degrees of freedom to the VTK output files.	false
99	--vtk-write-filter-velocities	A Boolean value set to true or false that turns on (true) or off (false) the output of the filter velocities of the phases in the VTK output files.	false
100	--vtk-write-fugacities	A Boolean value set to true or false that turns on (true) or off (false) the output of the component fugacities to the VTK output files.	false
101	--vtk-write-fugacity-coeffs	A Boolean value set to true or false that turns on (true) or off (false) the output of the component fugacity coefficients to the VTK output files.	false
102	--vtk-write-gas-dissolution-factor	A Boolean value set to true or false that turns on (true) or off (false) the output of the gas dissolution factor (Rs) of the observed oil to the VTK output files.	false
103	--vtk-write-gas-formation-volume-factor	A Boolean value set to true or false that turns on (true) or off (false) the output of the gas formation volume factor (Bg) to the VTK output files.	false
104	--vtk-write-gas-saturation-pressure	A Boolean value set to true or false that turns on (true) or off (false) the output of the saturation pressure of gas (Pg,sat) to the VTK output files'	false
105	--vtk-write-intrinsic-permeabilities	A Boolean value set to true or false that turns on (true) or off (false) the output of the intrinsic permeability to the VTK output files.	false
106	--vtk-write-mass-fractions	A Boolean value set to true or false that turns on (true) or off (false) the output of the mass fractions to the VTK output files.	false

[397] *Virtualization ToolKit ("VTK") files used to visualize the 3D results from the model using ParaView, an open-source, multi-platform general purpose data analysis and visualization application (https://www.paraview.org/overview/). For the commercial simulator's compatible output files OPM's ResInsight 3D visualization package can be used instead and is specifically tailored for OPM Flow and the commercial simulator.*

OPM Flow 2019-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
107	--vtk-write-mobilities	A Boolean value set to true or false that turns on (true) or off (false) the output of the phase mobilities to the VTK output files.	false
108	--vtk-write-molarities	A Boolean value set to true or false that turns on (true) or off (false) the output of the component molarities to the VTK output files.	false
109	--vtk-write-mole-fractions	A Boolean value set to true or false that turns on (true) or off (false) the output of the mole fractions to the VTK output files.	true
110	--vtk-write-oil-formation-volume-factor	A Boolean value set to true or false that turns on (true) or off (false) the output of the oil formation volume factor (Bo) to the VTK output files.	false
111	--vtk-write-oil-saturation-pressure	A Boolean value set to true or false that turns on (true) or off (false) the output of the saturation pressure of oil (Po,sat) to the VTK output files.	false
112	--vtk-write-oil-vaporization-factor	A Boolean value set to true or false that turns on (true) or off (false) the output of the oil vaporization factor (Rv) of the observed gas to the VTK output files.	false
113	--vtk-write-porosity	A Boolean value set to true or false that turns on (true) or off (false) the output of the porosity array to the VTK output files.	true
114	--vtk-write-potential-gradients	A Boolean value set to true or false that turns on (true) or off (false) the output of the phase pressure potential gradients to the VTK output files.	false
115	--vtk-write-pressures	A Boolean value set to true or false that turns on (true) or off (false) the output of the phase pressures to the VTK output files.	true
116	--vtk-write-primary-vars	A Boolean value set to true or false that turns on (true) or off (false) the output of the primary variables to the VTK output files.	false
117	--vtk-write-primary-vars-meaning	A Boolean value set to true or false that turns on (true) or off (false) the output of the how the primary variables should be interpreted to the VTK output files.	false
118	--vtk-write-process-rank	A Boolean value set to true or false that turns on (true) or off (false) the output of the MPI process rank to the VTK output files.	false
119	--vtk-write-relative-permeabilities	A Boolean value set to true or false that turns on (true) or off (false) the output of the phase relative permeabilities to the VTK output files.	true
120	--vtk-write-saturated-gas-oil-vaporization-factor	A Boolean value set to true or false that turns on (true) or off (false) the output of the oil vaporization factor (Rv,sat) of oil saturated gas to the VTK output files.	false

OPM Flow 2019-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
121	--vtk-write-saturated-oil-gas-dissolution-factor	A Boolean value set to true or false that turns on (true) or off (false) the output of the gas dissolution factor (Rs,sat) of gas saturated oil to the VTK output files.	false
122	--vtk-write-saturation-ratios	A Boolean value set to true or false that turns on (true) or off (false) the output of the ratio of the actually and maximum dissolved component of the mixture to the VTK output files	false
123	--vtk-write-saturations	A Boolean value set to true or false that turns on (true) or off (false) the output of the phase saturations to the VTK output files.	true
124	--vtk-write-temperature	A Boolean value set to true or false that turns on (true) or off (false) the output of the temperature array to the VTK output files.	true
125	--vtk-write-total-mass-fractions	A Boolean value set to true or false that turns on (true) or off (false) the output of the total mass fractions to the VTK output files.	false
126	--vtk-write-total-mole-fractions	A Boolean value set to true or false that turns on (true) or off (false) the output of the total mole fractions to the VTK output files.	false
127	--vtk-write-viscosities	A Boolean value set to true or false that turns on (true) or off (false) the output of the component phase viscosities to the VTK output files.	false
128	--vtk-write-water-formation-volume-factor	A Boolean value set to true or false that turns on (true) or off (false) the output of the oil formation volume factor (Bo) to the VTK output files.	false

Notes:

1) Cells colored green in the No. column are new command line parameters for this release.
2) Items shaded in gray are considered to be developer options that should be used with caution as the values associated with these command line parameters are subject to change.
3) As per all UNIX and LINUX based system the input is case dependent.
4) If a parameter file is used to set one of the above parameters, the leading "--" should not be incorporated in the parameter file.

Table E.9: OPM Flow 2019-04 Command Line Options

E.10 RUNNING OPM FLOW 2018-10

OPM Flow release 2018-10 and beyond have switched to the eWoms/ebos[398] command line interface. Previously the simulator used only OPM Flow specific command line parameters which were internally translated to the equivalent eWoms/ebos command line parameters. The pre 2018-10 set of command line parameters are now deprecated but are still documented in this section for backward compatibility with previous releases of the simulator.

The command line syntax for running OPM Flow is:

```
flow [OPTIONS] CASENAME
```

and typing the following command from your terminal:

```
flow CASENAME
```

will start the simulator and run the case specified by CASENAME.DATA.

OPM Flow accepts command line options to control various aspects of the simulator's run time behavior, as described in the table below. To give a command line option use "key=value" syntax, with no spaces around the equals sign.

```
flow --ecl-deck-file-name=path_to_data/CASENAME
```

It is also possible to put multiple options together in a parameter file. To do so, put one option on each line of the file with ".param" as the extension and pass that filename as a command line parameter to OPM Flow using:

```
flow --parameter-file=CASENAME.PARAM
```

to instruct OPM Flow to read the parameter file.

The available command line options for this release are depicted in Table E.10.

OPM Flow 2018-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
	General eWoms/ebos Command Line Parameters		
1	-h or --help	A character string that causes OPM Flow to print a help message that gives a brief description of the available command line parameters.	N/A
2	--dbph-max-rel	A real positive value that sets maximum relative change of the bottom-hole pressure in a single iteration	1.0
3	--dp-max-rel	A real positive double precision value that sets the maximum allowed relative change of pressure per iteration.	0.3
4	--ds-max	A real positive double precision value that sets the maximum allowed change in saturation per iteration.	0.2
5	--dwell-fraction-max	A real positive double precision value that sets the maximum allowed change in well's volume fraction per iteration.	0.2

[398] *eWorms is a C++ software framework concerned with fully-implicit numerical models for flow and transport in porous media. It it is developed as an integral part of the Open Porous Media (OPM) initiative.*

OPM Flow 2018-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
6	--ecl-deck-file-name	A character string that defines the name of the OPM Flow input file which contains the simulator's ECLIPSE formatted input deck to be simulated.	N/A
7	--ecl-output-double-precision	A Boolean value set to true or false that switches on (true) or off (false) double precision in restart files. Useful for 'perfect' restarts.	false
8	--ecl-output-interval	An integer value defining the number of report steps that ought to be skipped between two writes of restart files results.	-1
9	--enable-adaptive-time-stepping	A Boolean value set to true or false that that turns on (true) or off (false) adaptive time stepping. If set to false the report time steps provided by the input deck are used.	true
10	--enable-async-ecl-output	A Boolean value set to true or false that sets the output mode to be asynchronously (true), letting the simulator continue computing the next time step while writing results to restart and summary files.	true
11	--enable-async-vtk-output	A Boolean value set to true or false that sets the output mode to be asynchronously (true), letting the simulator continue computing the next time step while writing results to the VTK output files.	true
12	--enable-dry-run	A Boolean value set to true or false that specifies if the simulation should actually run, (true) or just check the input deck (false). This option is equivalent to activating the NOSIM keyword in the RUNSPEC section of the input deck (see section *5.3.97 NOSIM – Activate the No Simulation Mode for Data File Checking*).	true
13	--enable-ecl-output	A Boolean value set to true or false to write the binary output which is compatible with the ECLIPSE commercial simulator (restart and summary files).	true
14	--enable-opm-rst-file	A Boolean value set to true or false to write OPM specific data sets to the commercial simulators restart file to enable restart of an OPM Flow run (true), or not to write the data (false).	true
15	--enable-storage-cache	A Boolean value set to true or false that turns on (true) or off (false) storing previous storage terms and avoid re-calculating them.	true
16	--enable-terminal-output	A Boolean value set to true or false that turns on (true) or off (false) high-level information about the simulation's progress to the terminal	true

OPM Flow 2018-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
17	--enable-tuning	A Boolean value set to true or false that that instructs OPM Flow to read the time stepping parameters from the *TUNING - Numerical Tuning Control* keyword in the input deck, if set to true. Note that only the first record of the TUNING keyword is processed.	false
18	--enable-vtk-output	A Boolean value set to true or false that turns on (true) or off (false) a global switch for writing VTK files.	false
19	--enable-write-all-solutions	A Boolean value set to true or false that turns on (true) or off (false) the writing of all solutions to disk instead of only the ones for the report steps.	false
	OPM Flow Specific Command Line Parameters		
20	--flow-linear-solver-verbosity	A positive integer value that defines the output from linear solver: 0 : no extra output 1 : output per solution iteration 2 : output per iteration	0
21	--flow-newton-max-iterations	A positive integer that defines the maximum number of Newton iterations per time step used by the simulator.	20
22	--flow-newton-min-iterations	A real positive value that sets the minimum number of Newton iterations per time step used by the simulator.	1
	General eWoms/ebos Command Line Parameters		
23	--force-disable-fluid-in-place-output	A Boolean value set to true or false that instructs OPM Flow to not to print the Fluid In-Place report after each report time step (true) or not (false).	false
24	--full-time-step-initially	A Boolean value set to true or false that instructs OPM Flow to always attempt to finish a report step using a single time step (true) or not (false).	false
25	--ilu-fillin-level	A positive integer value that sets the fill in level for the ILU preconditioner.	0
26	--ilu-redblack	A Boolean value set to true or false that instructs OPM Flow to use red-black partitioning for the ILU preconditioner. (true) or not (false).	false
27	--ilu-relaxation	A real positive double precision value that sets the relaxation factor of the linear solver's ILU preconditioner	0.9

OPM Flow 2018-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
28	--ilu-reorder-spheres	A Boolean value set to true or false specifies OPM Flow to reorder the entries of the matrix in the red-black ILU preconditioner in spheres starting at an edge (true) or not (false) If false the original ordering is preserved in each color. Otherwise try to ensure D4 ordering (in a 2D structured grid, the diagonal elements are consecutive).	false
29	--initial-time-step-in-days	A real double precision value that sets the size of initial time step in days.	1.0
30	--linear-solver-ignore-convergence-failure	A Boolean value set to true or false that if set to true convergences failures in the linear solver are ignored. This option should be used with care, as the results may be unreliable.	false
31	--linear-solver-max-iter	A positive integer value that defines the maximum number of linear iterations.	200
32	--linear-solver-reduction	A real positive double precision value that sets the tolerance for the linear solver. The linear solver convergences when the residual is reduced sufficiently.	0.01
33	--linear-solver-require-full-sparsity-pattern	A Boolean value set to true or false that if set to true the simulator will produce the full sparsity pattern for the linear solver.	false
34	--linear-solver-restart	A positive integer value that sets the number of iterations after which GMRES is restarted.	40
35	--matrix-add-well-contributions	A Boolean value set to true or false that if set to true explicitly specifies the influences of wells between cells in the Jacobian and preconditioner matrices.	false
36	--max-inner-iter-ms-wells	A positive integer value that defines the maximum number of inner iterations for multi-segment wells.	10
37	--max-pressure-change-ms-wells	A real positive value that defines the maximum relative pressure change for a single iteration of the multi-segment well model.	200000
38	--max-residual-allowed	A real positive value that sets the absolute maximum tolerance for residuals without cutting the time step size.	1.0×10^7
39	--max-single-precision-days	A real positive value that set the maximum time step size where single precision floating point arithmetic can be used solving for the linear systems of equations.	20.0
40	--max-strict-iter	A positive integer that sets the maximum number of Newton iterations before relaxed tolerances are used for the CNV convergence criterion.	8
41	--max-welleq-iter	A positive integer that defines the maximum number of iterations to determine the solution to the well equations.	15

OPM Flow 2018-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
42	--milu-variant	A defined character string that specifies which variant of the modified-ILU preconditioner ought to be used. Possible variants are: 1) ILU (default, plain ILU), 2) MILU_1 (lump diagonal with dropped row entries), 3) MILU_2 (lump diagonal with the sum of the absolute values of the dropped row entries), 4) MILU_3 (if diagonal is positive add sum of dropped row entrires. Otherwise substract them), 5) MILU_4 (if diagonal is positive add sum of dropped row entrires. Otherwise do nothing. The default is ILU	ILU
43	--newton-max-relax	A real positive value that sets the maximum relaxation factor of a Newton iteration used by the simulator.	0.5
44	--newton-relaxation-type	A character string that defines the type of relaxation used in Newton's method., default is dampen.	dampen
45	--output-dir	A character string that defines the directory to which OPM Flow to write the ECLIPSE compatible output files (restart and summary files).	N/A
46	--output-interval	A positive integer that specifies the number of report steps between two consecutive writes of restart data.	1
47	--output-mode	A character string that defines the output to *.PRT and *.DBG files: 1) none: No output to the files. 2) log or false: Output logging information only. 3) all or true: Output everything. For example to just output logging information use: `--output-mode=log` or `--output-mode=false`	all
48	--parameter-file	A character string that defines the name of a parameter file which contains the simulator's set of run-time parameters, as listed in this table.	N/A
49	--preconditioner-add-well-contributions	A Boolean value set to true or false that switches on (true) or off (false) the influences of wells between cells for the preconditioner matrix only.	false
50	--pri-var-oscilation-threshold	A real positive value that defines the threshold value for the primary variable switching conditions after its meaning has switched to hinder oscilations.	1×10^{-5}
51	--print-parameters	A positive integer value that request that the run time parameters be printed at the start of the run: 0 : No output to the files. 1 : Output *.DBG file 2 : Output to *.DBG and *.PRT files (default)	2

OPM Flow 2018-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
52	--print-properties	A positive integer value that request that the compile time parameters be printed at the start of the run: 0 : No output to the files. 1 : Output *.DBG file 2 : Output to *.DBG and *.PRT files (default)	2
53	--solve-welleq-initially	A Boolean value set to true or false that switches on (true) or off (false) the solving of the well equations as a pre-processing step. Note that the well equations are always added to the full system and solved until converged.	true
54	--solver-growth-factor	A real positive value that specifies the allowed value a time step can be increased by, subject to the maximum allowable time step size set by the --flow-solver-max-time-step-in-days parameter. For example, if the current time step has converged at 10 days and --flow-solver-growth-factor is set to the default value, then the next time step will be 2.0 * 10 days, that is at 20 days.	2.0
55	--solver-max-growth	A real positive value that specifies the maximum allowed value a time step can be increased by after a report time step. subject to the maximum allowable time step size set by the --flow-solver-max-time-step-in-days parameter.	3.0
56	--solver-max-restarts	A positive integer that defines the number of allowed consecutive restarts (or time step chops) before the simulation is terminated.	10
57	--solver-max-time-step-in-days	A real positive double precision value that specifies the maximum allowed time step size in days.	365
58	--solver-restart-factor	A real positive double precision value that sets the time step chop factor of the time step after a convergence failure. For example, if the current non-convergent time step is 30 days and --flow-solver-restart-factor is set to the default value, then the time step will be repeated using 0.33 * 30 days as the time step, that is 9.9 days.	0.33
59	--solver-verbosity	A positive integer that specifies the "chattiness" of the non-linear solver.	1
60	--threads-per-process	A positive integer value that stipulates the maximum number of threads to be instantiated per process ('-1' means 'automatic').	1
61	--time-step-after-event-in-days	A real double precision value that sets maximum allowed time step after an event; for example, when a well is open or closed etc. The default value of -1 means that events to do effect the time stepping.	-1

OPM Flow 2018-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
62	--time-step-control	A character string that defines the time stepping control algorithm and is set to one of the following: 1) pid: PID controller based adaptive time step control as suggested by Turek and Kuzmin[399]. 2) pid+iteration: Use PID and linear iteration numbers to guide the time step. 3) pid+newtoniteration: Use PID and non-linear iterations numbers to guide the time step. 4) Hardcoded: Use time steps supplied by user. Via timestep.control.filename	pid
63	--time-step-control-decay-rate	A real positive value that specifies the decay rate a time step can be decreased by after the number of target iterations has been exceeded	0.75
64	--time-step-control-file-name	A character string that specifies a filename where time steps are specified. The default is the character string timesteps For instance generated by the ecl_summary application in libecl as per the following UNIX command line: `path_to_libecl_applications/ ecl_summary DECK TIME > filename` Where: DECK is the name of the data deck you want to get the time steps from, TIME tells the application to return the timing for the run, and "filename" is the name of the file the times are piped to.	timesteps
65	--time-step-control-growth-rate	A real positive value that specifies the growth rate of the time step size of the number of target iterations is undercut.	1.25
66	--time-step-control-target-iterations	A positive integer that defines the number of linear iterations which the time step control scheme should aim for (if applicable).	30
67	--time-step-control-target-newton-iterations	A positive integer that specifies the number of Newton iterations which the time step control scheme should aim for (if applicable).	8
68	--time-step-control-tolerance	A real double precision value that sets the tolerance for PID (only used with the pid and pid+ options defined by the --flow-timestep.control option).	0.1
69	--time-step-verbosity	A positive integer that specifies the "chattiness" during the time integration.	1

[399] *Algebraic Flux Correction III. Incompressible Flow Problems. Uni Dortmund, Turek and Kuzmin, January, 2006 (DOI: 10.1007/3-540-27206-2_8)*

OPM Flow 2018-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
70	--tolerance-cnv	A real positive double precision value that specifies the maximum non-linear tolerance error. This is the local convergence tolerance (maximum of local saturation errors).	0.01
71	--tolerance-cnv-relaxed	A real positive value that defines the relaxed local convergence tolerance that applies for iterations after the iterations with the strict tolerance.	1.0×10^{9}
72	--tolerance-mb	A real positive double precision value that sets the maximum mass balance error, that is the tolerated mass balance error relative to total mass present.	1.0 x 105
73	--tolerance-pressure-ms-wells	A real positive double precision value that specifies the tolerance for the pressure equations for multi-segment wells.	1000
74	--tolerance-well-control	A real positive double precision value that sets the maximum tolerance for the well control equations.	1.0×10^{-7}
75	--tolerance-wells	A real positive double precision value that defines the maximum non-linear error for the well equations.	0.0001
76	-update-equations-scaling	A Boolean value set to true or false that switches on (true) or off (false) the updating of the scaling factors for mass balance equations during the simulation.	false
77	--use-amg	A Boolean value set to true or false that if set to true OPM Flow will use AMG as the linear solver's preconditioner	false
78	--use-cpr	A Boolean value set to true or false that when set to true OPM Flow will use CPR as the linear solver's preconditioner	false
79	--use-gmres	A Boolean value set to true or false that when set to true OPM Flow will use GMRes instead of BiCGStab as linear solver within the Newton iterations.	false
80	--use-inner-iterations-ms-wells	A Boolean value set to true or false that when set to true the simulator will use nested iterations for multi-segment wells.	true
81	--use-multisegment-well	A Boolean value set to true or false that when set to true the simulator will use the well model for multi-segment wells instead of the one for single-segment wells.	false
82	--use-update-stabilization	A Boolean value set to true or false that switches on (true) or off (false) the stabilized Newton option. This option may improve convergence for some cases.	true
	VTK Graphics Command Line Parameters[400]		

[400] *Virtualization ToolKit ("VTK") files used to visualize the 3D results from the model using ParaView, an open-source, multi-platform general purpose data analysis and visualization application (https://www.paraview.org/overview/). For ECLIPSE compatible output files OPMS's ResInsight 3D visualization package can be used which is specifically tailored for OPM Flow and the commercial simulator.*

OPM Flow 2018-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
83	--vtk-write-average-molar-masses	A Boolean value set to true or false that turns on (true) or off (false) the output of the average phase mass in the VTK output files.	false
84	--vtk-write-densities	A Boolean value set to true or false that turns on (true) or off (false) the output of the phase densities to the VTK output files	true
85	--vtk-write-dof-index	A Boolean value set to true or false that turns on (true) or off (false) the output of the index of the degrees of freedom to the VTK output files.	false
86	--vtk-write-extrusion-factor	A Boolean value set to true or false that turns on (true) or off (false) the output of the of the extrusion factor of the degrees of freedom to the VTK output files.	false
87	--vtk-write-filter-velocities	A Boolean value set to true or false that turns on (true) or off (false) the output of the filter velocities of the phases in the VTK output files.	false
88	--vtk-write-fugacities	A Boolean value set to true or false that turns on (true) or off (false) the output of the component fugacities to the VTK output files.	false
89	--vtk-write-fugacity-coeffs	A Boolean value set to true or false that turns on (true) or off (false) the output of the component fugacity coefficients to the VTK output files.	false
90	--vtk-write-gas-dissolution-factor	A Boolean value set to true or false that turns on (true) or off (false) the output of the gas dissolution factor (Rs) of the observed oil to the VTK output files.	false
91	--vtk-write-gas-formation-volume-factor	A Boolean value set to true or false that turns on (true) or off (false) the output of the gas formation volume factor (Bg) to the VTK output files.	false
92	--vtk-write-gas-saturation-pressure	A Boolean value set to true or false that turns on (true) or off (false) the output of the saturation pressure of gas (Pg,sat) to the VTK output files'	false
93	--vtk-write-intrinsic-permeabilities	A Boolean value set to true or false that turns on (true) or off (false) the output of the intrinsic permeability to the VTK output files.	false
94	--vtk-write-mass-fractions	A Boolean value set to true or false that turns on (true) or off (false) the output of the mass fractions to the VTK output files.	false
95	--vtk-write-mobilities	A Boolean value set to true or false that turns on (true) or off (false) the output of the phase mobilities to the VTK output files.	false
96	--vtk-write-molarities	A Boolean value set to true or false that turns on (true) or off (false) the output of the component molarities to the VTK output files.	false

OPM Flow 2018-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
97	--vtk-write-mole-fractions	A Boolean value set to true or false that turns on (true) or off (false) the output of the mole fractions to the VTK output files.	true
98	--vtk-write-oil-formation-volume-factor	A Boolean value set to true or false that turns on (true) or off (false) the output of the oil formation volume factor (Bo) to the VTK output files.	false
99	--vtk-write-oil-saturation-pressure	A Boolean value set to true or false that turns on (true) or off (false) the output of the saturation pressure of oil (Po,sat) to the VTK output files.	false
100	--vtk-write-oil-vaporization-factor	A Boolean value set to true or false that turns on (true) or off (false) the output of the oil vaporization factor (Rv) of the observed gas to the VTK output files.	false
101	--vtk-write-porosity	A Boolean value set to true or false that turns on (true) or off (false) the output of the porosity array to the VTK output files.	true
102	--vtk-write-potential-gradients	A Boolean value set to true or false that turns on (true) or off (false) the output of the phase pressure potential gradients to the VTK output files.	false
103	--vtk-write-pressures	A Boolean value set to true or false that turns on (true) or off (false) the output of the phase pressures to the VTK output files.	true
104	--vtk-write-primary-vars	A Boolean value set to true or false that turns on (true) or off (false) the output of the primary variables to the VTK output files.	false
105	--vtk-write-primary-vars-meaning	A Boolean value set to true or false that turns on (true) or off (false) the output of the how the primary variables should be interpreted to the VTK output files.	false
106	--vtk-write-process-rank	A Boolean value set to true or false that turns on (true) or off (false) the output of the MPI process rank to the VTK output files.	false
107	--vtk-write-relative-permeabilities	A Boolean value set to true or false that turns on (true) or off (false) the output of the phase relative permeabilities to the VTK output files.	true
108	--vtk-write-saturated-gas-oil-vaporization-factor	A Boolean value set to true or false that turns on (true) or off (false) the output of the oil vaporization factor (Rv,sat) of oil saturated gas to the VTK output files.	false
109	--vtk-write-saturated-oil-gas-dissolution-factor	A Boolean value set to true or false that turns on (true) or off (false) the output of the gas dissolution factor (Rs,sat) of gas saturated oil to the VTK output files.	false

OPM Flow 2018-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
110	--vtk-write-saturation-ratios	A Boolean value set to true or false that turns on (true) or off (false) the output of the ratio of the actually and maximum dissolved component of the mixture to the VTK output files	false
111	--vtk-write-saturations	A Boolean value set to true or false that turns on (true) or off (false) the output of the phase saturations to the VTK output files.	true
112	--vtk-write-temperature	A Boolean value set to true or false that turns on (true) or off (false) the output of the temperature array to the VTK output files.	true
113	--vtk-write-total-mass-fractions	A Boolean value set to true or false that turns on (true) or off (false) the output of the total mass fractions to the VTK output files.	false
114	--vtk-write-total-mole-fractions	A Boolean value set to true or false that turns on (true) or off (false) the output of the total mole fractions to the VTK output files.	false
115	--vtk-write-viscosities	A Boolean value set to true or false that turns on (true) or off (false) the output of the component phase viscosities to the VTK output files.	false
116	--vtk-write-water-formation-volume-factor	A Boolean value set to true or false that turns on (true) or off (false) the output of the oil formation volume factor (Bo) to the VTK output files.	false

Notes:

1) Items shaded in gray are considered to be developer options that should be used with caution as the values associated with these command line parameters are subject to change.
2) As per all UNIX and LINUX based system the input is case dependent.
3) If a parameter file is used to set one of the above parameters, the leading "--" should not be incorporated in the parameter file.

Table E.10: OPM Flow 2018-10 Command Line Options

E.11 RUNNING OPM FLOW 2018-04

This section describes the command line options up to the 2018-04 release of OPM Flow, post this release the command line options were changed to be the same as eWoms/ebos[401] command line parameters. It is anticipated that this section will be removed from the manual once the 2018-10 and later versions are firmly established.

The command line syntax for running OPM Flow is:

```
flow [OPTIONS] CASENAME
```

and typing the following command from your terminal:

```
flow CASENAME
```

will start the simulator and run the case specified by CASENAME.DATA.

OPM Flow accepts command line options to control various aspects of the simulator's run time behavior, as described in the table below. To give a command line option use "key=value" syntax, with no spaces around the equals sign. It is also possible to put multiple options together in a parameter file. To do so, put one option on each line of the file with ".param" as the extension and pass that filename as a command line parameter to OPM Flow using:

```
flow --parameter-file=CASENAME.PARAM
```

to instruct OPM Flow to read the parameter file.

The available command line options for this release are listed in Table E.11.

OPM Flow 2018-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
1	deck_filename	A character string that defines the name of the OPM Flow input file which contains the simulator's ECLIPSE formatted input deck to be simulated.	N/A
2	output_ecl	A Boolean value set to true or false that turns on (true) or off (false) output to the .*RST and .*SMRY files.	true
3	output	A character string that defines the output to *.PRT and *.DBG files: 1) none: No output to the files. 2) log or false: Output logging information only. 3) all or true: Output everything. For example to just output logging information use: `output=log` or `output=false`	all
4	output_dir	Set the directory to which output files are written.	deck location

[401] *eWorms is a C++ software framework concerned with fully-implicit numerical models for flow and transport in porous media. It it is developed as an integral part of the Open Porous Media (OPM) initiative.*

OPM Flow 2018-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
5	restart_double_si	A Boolean value set to true or false that switches on (true) or off (false) double precision in restart files and stores all restart data in SI units rather than using the unit family (METRIC, FIELD etc.) used in the input deck. The option improves the quality of the restart.	false
6	async_output	A Boolean value set to true or false that sets the output mode to be asynchronously (true), letting the simulator continue computing the next time step while writing results to restart and summary files.	true
7	newton_use_gmres	A Boolean value set to true or false that when set to true OPM Flow will use GMRes instead of BiCGStab as linear solver within the Newton iterations.	false
8	linear_solver_reduction	A real positive double precision value that sets the tolerance for the linear solver. The linear solver convergences when the residual is reduced sufficiently.	0.01
9	linear_solver_maxiter	A positive integer value that defines the maximum number of linear iterations.	150
10	linear_solver_verbosity	A positive integer value that defines the output from linear solver: 0 : no extra output 1 : output per solution iteration 2 : output per iteration	0
11	linear_solver_ignoreconvergencefailure	A Boolean value set to true or false that if set to true convergences failures in the linear solver are ignored. This option should be used with care, as the results may be unreliable.	false
12	linear_solver_use_amg	A Boolean value set to true or false that if set to true OPM Flow will use amg as the preconditioner instead of ilu.	false
13	ilu_relaxation	A real positive double precision value that sets the relaxation parameter for the ILU preconditioner.	0.9
14	ilu_fillin_level	A positive integer value that sets the fill in level for the ILU preconditioner.	0
15	dp_max_rel	A real positive double precision value that sets the maximum allowed relative change of pressure per iteration.	0.3
16	ds_max	A real positive double precision value that sets the maximum allowed change in saturation per iteration.	0.2

OPM Flow 2018-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
17	dr_max_rel	A real positive double precision value that sets the maximum allowed relative change in dissolved gas and vaporized oil per iteration	1e+09
18	dbhp_max_rel	A real positive double precision value that sets the maximum allowed relative change in BHP per iteration.	1
19	dwell_fraction_max	A real positive double precision value that sets the maximum allowed change in well's volume per iteration.	1e+07
20	tolerance_mb	A real positive double precision value that sets the maximum mass balance error.	1e-05
21	tolerance_cnv	A real positive double precision value that specifies the maximum non-linear tolerance error.	0.01
22	tolerance_wells	A real positive double precision value that defines the maximum non-linear error for the well equations.	0.0001
23	max_strict_iter	A positive integer value that sets the maximum number of non-linear iterations. After this maximum value has been exceeded only the mass balance error is checked.	8
24	solve_welleq_initially	A Boolean value set to true or false that switches on (true) or off (false) the solving of the well equations as a pre-processing step. Note that the well equations are always added to the full system and solved until converged.	true
25	use_update_stabilization	A Boolean value set to true or false that switches on (true) or off (false) the stabilized Newton option. This option may improve convergence for some cases.	true
26	relax_max	A real positive double precision value that is used to tune the stabilized Newton option.	0.5
27	relax_type	A character string that sets relaxation type of the stabilized Newton option.	dampen
28	max_iter	A positive integer that sets the maximum number of non-linear iterations.	10
29	min_iter	A positive integer that sets the minimum number of non-linear iterations.	1
30	output_terminal	A Boolean value set to true or false that turns on (true) or off (false) output to terminal.	true

OPM Flow 2018-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
31	use_TUNING	A Boolean value set to true or false that that instructs OPM Flow to read the time stepping parameters from the *TUNING - Numerical Tuning Control* keyword in the input deck, if set to true. Note that only the first record of the TUNING keyword is processed.	false
32	timestep.adaptive	A Boolean value set to true or false that that turns on (true) or off (false) adaptive time stepping. If set to false the report time steps provided by the input deck are used.	true
33	solver.restartfactor	A real positive double precision value that sets the time step chop factor of the time step after convergence failure. For example, if the current non-convergent time step is 30 days and solver.restartfactor is set to the default value, then the time step will be repeated using 0.33 * 30 days as the time step, that is 9.9 days.	0.33
34	timestep.control.maxgrowth	A real positive double precision value that sets the maximum allowed value a time step can be increased by, subject to the maximum allowable time step size set by the timestep.max_timestep_in_days parameter. For example, if the current time step has converged at 10 days and solver.restartfactor is set to the default value, then the next time step will be 3.0 * 10 days, that is at 30 days.	3
35	timestep.max_timestep_in_days	A real positive double precision value that sets the maximum allowed time step size in days.	365
36	solver.restart	A positive integer that defines the number of allowed consecutive restarts (or time step chops) before the simulation is terminated.	10
37	solver.verbose	A Boolean value set to true or false that switches on (true) or off (false) solver specific output.	true
38	timestep.verbose	A Boolean value set to true or false that switches on (true) or off (false) time step specific output.	true
39	timestep.initial_timestep_in_days	A real double precision value that sets the size of initial time step in days. The default value of -1 sets the initial time step to be *solver.restartfactor* * the length of the first report step.	-1
40	full_timestep_initially	Try to use the report steps as time steps.	false
41	timestep.timestep_in_days_after_event	A real double precision value that sets maximum allowed time step after an event; for example, when a well is open or closed etc. The default value of -1 means that events to do effect the time stepping.	-1

OPM Flow 2018-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
42	timestep.control	A character string that defines the time stepping control algorithm and is set to one of the following: 1) pid: PID controller based adaptive time step control as suggested by Turek and Kuzmin[402]. 2) pid+iteration: Use PID and linear iteration numbers to guide the time step. 3) pid+newtoniteration: Use PID and non-linear iterations numbers to guide the time step. 4) Hardcoded: Use time steps supplied by user. Via timestep.control.filename	pid
43	timestep.control.tol	A real double precision value that sets the tolerance for PID (only used with the pid and pid+ options defined by the timestep.control option).	0.1
44	timestep.control.targetiteration	A positive integer that sets the target number of linear / non-linear iterations. This option can only be only used by pid+iterations and pid+newtoniteration defined by the timestep.control option.	8
45	timestep.control.filename	A character string that specifies a file name where time steps are specified. For instance generated by the ecl_summary application in libecl as per the following UNIX command line: `path_to_libecl_applications/ecl_summary DECK TIME > filename` Where: DECK is the name of the data deck you want to get the time steps from, TIME tells the application to return the timing for the run, and "filename" is the name of the file the times are piped to.	
Notes: 1) As per all UNIX and LINUX based system the input is case dependent.			

Table E.11: OPM Flow 2018-04 Command Line Options

[402] *Algebraic Flux Correction III. Incompressible Flow Problems. Uni Dortmund, Turek and Kuzmin, January, 2006 (DOI: 10.1007/3-540-27206-2_8)*

APPENDIX F: OPM FLOW OUTPUT FILE FORMATS

APPENDIX TABLE OF CONTENTS

APPENDIX INDEX OF TABLES

F.1 OVERVIEW

This section of the manual describes the various output files generated by OPM Flow and attempts to outline the format of the various files, in order for third party software to read and write these type of files. From the 2019-04 release a substantial effort has been undertaken to make OPM Flow's output files compatible with the commercial simulator's output files. This compatibility enables OMP Flow to "restart" from the commercial simulator's generated RESTART files as well as the commercial simulator to "restart" from the OPM Flow's RESTART files. Although, this might not be the case for all models, as restarting a simulation model from another simulators restart file is complicated and may not be possible in all cases.

Where applicable, files written by OPM Flow can be loaded into OPM ResInsight post processing software for further analysis and for displaying the results. Please see section *2.2 Running OPM Flow 2023-10 From The Command Line* for the various command line options for setting the output format type.

F.1.1 FILE TYPES

OPM Flow, similar to the commercial simulator, writes out various files, some of which are used by post processing software (OPM ResInsight) and some that are directly used by the user, for example the *.PRT file that contains various reports. Table F.1 summaries the various file formats and the status of the file formats currently supported by OPM Flow.

File Type	Data Type	Description	OPM Flow Status
DATA	Input Data	DATA files contain the input data in ASCII format used to run OPM Flow.	Fully Supported, as outlined in the manual
DBG	Debug Data	This file contains ASCII developer debug output specific to OPM Flow, that is there no compatibility with the commercial simulator's DBG file.	OPM Flow Specific
EGRID	Structure Data	EGRID files contain the structural information for the model via the COORD and ZCORN etc., keywords, and employs the extensible GRID format, which is more compact than the GRID formatted files and is the only format supported by OPM Flow. The output for this file type must be requested via the *GRIDFILE – Set the Grid File Output Options* in the GRID section. The data is used in post-processing software to view the grid and is not in itself used by OPM Flow.	Fully Supported
GRID	Structure Data	This file type contains the structural information for the model via the COORD and ZCORN etc., keywords, and consists of two data formats: 1) The standard GRID file format that contains only the active global cells. 2) The extended GRID file format that contains (1) plus local grid refinements and local coarsening grid specifications. Neither of these two formats are currently supported by OPM Flow.	Not Supported
INIT Index	Static Property Index	The INIT index file type specifies and defines the format and data type written to the INIT Data file. This file is not required by OPM Flow or OPM ResInsight to be able to read and write the INIT data file.	Not Supported and Not Required

File Type	Data Type	Description	OPM Flow Status
INIT Data	Static Property Data	This file type contains static model properties, that is, 1) Grid property data: porosity (PORO array), permeability (PERMX, PERMY, and PERMZ), net-to-gross (NTG), etc. 2) Region allocation array data: EQLUM, FIPNUM, PVTNUM, SATNUM, etc. 3) Fluid and rock property static data, including end-point scaling arrays: SGC, SGL, SOWCR, PCW etc. 4) Fluid and rock property tables: PVDG, PVTO, PCW, SGFN, SWFN, etc. In order to request this type of output see the *INIT – Activate the INIT File Option* in the GRID section. Activating this keyword results in both the INIT static property index and data files being written out by the commercial simulator. OPM Flow only writes out this file as the index file is not required by OPM ResInsight. The data is used in post-processing software to view the grid property data and is not in itself used by OPM Flow.	Fully Supported
LOG	Output File	The LOG file for the commercial simulator contains an ASCII copy of the output from the terminal. The file itself is not generated by the commercial simulator but by an auxiliary program, ECLRUN, that is used to call and execute all the commercial simulators. OPM Flow does not generate a LOG file; however, the file can be generated by using the Linux `tee` command, for example: `flow CASENAME | tee CASENAME.LOG` Which will copy the terminal output to the CASENAME.LOG file. Alternatively, one can use the OPMRUN, the Graphical User Interface ("GUI") program for OPM Flow, that provides similar functionality to the commercial simulator's auxiliary program.	OPM Flow Specific
PRT	Output File	This file contains ASCII user output reports specific to OPM Flow, although there is a strong desire to make the OPM Flow reports similar to those that are produced by the commercial simulator.	OPM Flow Specific
RESTART Index	Dynamic Solution Index	The RESTART index file type specifies and defines the format and data type written to the RESTART Data file. This file is not required by OPM Flow or OPM ResInsight to be able to read and write the RESTART data file.	Not Supported and Not Required

File Type	Data Type	Description	OPM Flow Status
RESTART Data	Dynamic Solution Data	This file type contains dynamic solution data for each active cell in the model at the user requested time steps, that is, 1) Grid solution pressure and saturation data (PRESSURE, SGAS, SOIL, and SWAT). Note that Ternary saturations are normally computed by the post-processing software. 2) Grid solution fluid property data (GAS_DEN, GAS_VISC, OIL-DENS, etc.). The *RPTRST – Define Data to be Written to the RESTART File* and the *RPTSCHED – Define SCHEDULE Section Reporting* keywords in the SCHEDULE defines the data and frequency of the data to be written to the RESTART file at each requested restart point. Activating this keyword results in both the RESTART index and data files being written out by the commercial simulator; OPM Flow only writes this file. In addition to the solution data, this file type also includes group and well configuration data (number of connections, connections open and closed etc.) This information is required to ensure that the correct group and well configuration is available at a given time in order to be able to restart the simulation at a restart point. The data is used to visualize the simulation results of the model in two and three dimensional space using post-processing software. The file is also used by OPM Flow to "restart" from a previous simulation case.	Fully Supported
RFT	Dynamic Wellbore Vector Data	Data written to the RFT file consists of wellbore vector data, for example, pressure and saturation versus wellbore depth at various time steps. The data written out is not restricted to just Repeat Formation Tester ("RFT") data, but can contain any Production Logging Tool ("PLT") data made available in the simulator. Note that only the RFT data set is currently supported by OPM Flow. The keywords *WRFT – Activate Well RFT Reporting to the RFT File* and *WRFTPLT – Activate Well RFT and PLT Reporting to the RFT File* in the SCHEDULE section are used to define the wells, data and time step at which the data should be written to the RFT file. The data is used in post-processing software to compare the actual wire line logging data with the simulation derived results. The file is not in itself used by OPM Flow.	Fully Supported (RFT)
RSM	Output File	The RSM file contains the variables requested to be written to the SUMMARY file via the keywords described in the *SUMMARY SECTION*, in a tabulated ASCII output format. The format of the file enables the data to be easily loaded into the LibreOffice Calc program for further processing, as each vector represents one column. The report is written at the end of the simulation run by parsing the SUMMARY Index and SUMMARY Data files.	Fully Supported
SAVE	Staic and Dynamic Data	SAVE files are basically a combination of both the INIT and RESTART files, except for the static initialization data that also includes the processed data, that is the PORV and TRANX, TRANY and TRANZ arrays etc. This enables "fast restarts" as the pore volumes and transmissibility arrays do not have to be re-calculated in a "restart" run. This file type is currently not supported by OPM Flow.	Not Supported

File Type	Data Type	Description	OPM Flow Status
SUMMARY ESMRY	Dynamic Vector Data	This is an OPM FLOW specific Enhanced SUMMARY (*.ESMRY) output file format that is optimized for fast loading of selected vectors by post-processing applications. The *.ESMRY file is re-written for every time step and does not have a performance impact on the simulation. Load times are in the order of 30 times faster compared with fetching selected vectors using the *.UNSMRY file format in post processing software.	Fully Supported
SUMMARY Index	Dynamic Vector Index	The SUMMARY index file type specifies and defines the format and data type written to the SUMMARY Data file.	Fully Supported
SUMMARY Data	Dynamic Vector Data	The SUMMARY data file contains the variables requested to be written to the file via the keywords described in the *SUMMARY SECTION*. The data consists of vectors that are used to generate line graphs of properties such as oil flow rate versus time, grid plot pressure versus time, etc. The properties to be stored on the SUMMARY file are written to the summary file at the end of each successful time step. The data can be used to compare actual production data with the simulation derived results in post processing software. Note that although the file format is fully supported, not all of the SUMMARY variables are are available.	Fully Supported

Notes:

1) All files can be written out in either ASCII or binary formats, except for DBG , LOG. PRT and RSM files that are always written in ASCII format.
2) In addition, SUMMARY and RESTART files can be loaded or written out in either unified or non-unified formats.
3) File Type cells colored in gray indicate the output may be different to the commercial simulators equivalent file type. File Type cells colored in orange represent file types that are not currently supported by OPM Flow.

Table F.1: OPM Flow Output File Types Summary

As mentioned in Table F.1 all files can be written out in either ASCII or binary formats and in addition the SUMMARY and RESTART files can be loaded or written out in either unified or non-unified formats. The file type (ASCII or binary) and file structure format (unified or non-unified formats) are set via various keywords in the RUNSPEC section, as outlined in Table F.2.

Process	RUNSPEC Keyword	Description	Files
Input	FMTIN	The keyword defines the input files to be formatted as ASCII i.e. text files, as oppose to binary files. The input deck file is always of this type. The option relates to the OPM Flow derived files that used as input, for for example when restarting from another case. If the keyword is omitted then the default is for binary file input.	*.FEGRID *.FINSPEC *.FINIT *.FRSSPEC *.FUNRST *.FSMSPEC *.FUNSMRY

Process	RUNSPEC Keyword	Description	Files
	MULTIN	A character string that defines the input files to be non-unified multiple files, as opposed to unified files. In this case, one file is read in per reporting time step, as opposed to all time steps reports being read from one file. If the keyword is omitted then the default is for one file per reporting time step.	*.RSSPEC *.X0001 *.SMSPEC *.S0001
	UNIFIN	UNIFIN defines the input files to be unified files, as opposed to non-unified multiple files. A unified file is a single file containing output for each reporting time step. For this option a single summary file and a single restart file will be read. If the keyword is omitted then the default is for one file per report time step.	*.RSSPEC *.UNRST *.SMSPEC *.UNSMRY
Output	FMTOUT	The keyword sets all output files to be formatted as ASCII i.e. text files, as opposed to binary files. The *.DBG, *.LOG, *.PRT and *.RSM files are always of this type. The option relates to the OPM Flow output files only. In this case the files will be portable across operating systems, but will also be very large in terms of hard disk space. For this reason it is recommend that the default option is used so that only binary files are written out. If the keyword is omitted then the default is for binary file input.	*.FEGRID *.FINSPEC *.FINIT *.FRSSPEC *.FUNRST *.FSMSPEC *.FUNSMRY
	MULTOUT	A character string that defines the output files to be non-unified multiple files, as opposed to unified files. In this case, one file is written for each reporting time step, as opposed to all time steps reports being written in one file. If the keyword is omitted then the default is for one file per reporting time step.	*.RSSPEC *.X0001 *.SMSPEC *.S0001
	UNIFOUT	UNIFOUT defines the output files to be unified files, as opposed to non-unified multiple files. A unified file is a single file containing output for each reporting time step. Here a single summary file and a single restart file will be generated, as opposed to one file per report time step. If the keyword is omitted then the default is for one file per report time step input.	*.RSSPEC *.UNRST *.SMSPEC *.UNSMRY

Notes:

1) A binary file is computer-readable but not human-readable. All executable programs are stored in binary files, as are most numeric data files. In contrast, text files are stored in a form (usually ASCII) that is human-readable.
2) For unified files if the run terminates unexpectedly, or there is sufficient disk space, then the last report output is not stored. The main advantage of unified files is that if a number of simulation cases reside in one directory, the number of output files per case is minimum compared to using non-unified multiple files. There is no limit on the number of reporting steps that a unified file can store.

Table F.2: RUNSPEC Input and Output File Format Keywords

F.1.2 FILE NAMING CONVENTIONS

OPM Flow automatically generates the file names based on the input file name and the output options selected via the keywords in the RUNSPEC summarized in in Table F.2. For example, the command line syntax for running OPM Flow is:

```
flow [OPTIONS] CASENAME
```

and typing the following command from the terminal:

```
flow CASENAME
```

will start the simulator and run the case specified by CASENAME.DATA. Here CASENAME is the "root" of the filename and DATA is the extension of the filename. OPM Flow will generate the output files based on the CASENAME with the extension based on the type (ASCII or binary) and file structure format (unified or non-unified formats), as outlined in Table F.3. Note that if If CASENAME includes a relative or absolute path to the data file, run directory will be set so that all output files are generated in the directory holding the data file.

File Type	Unformatted Extension	Description	Formatted Extension
DATA		Input data file.	*.DATA
DBG		Output debug file.	*.DBG
EGRID	*.EGRID	EGRID files containing the structural grid data and is the only format supported by OPM Flow. This output for this file type must be requested via the *GRIDFILE – Set the Grid File Output Options* in the GRID section.	*.FEGRID
GRID	*.GRID	Grid file in either the standard GRID file format or the extended GRID file format Neither of these two formats are currently supported by OPM Flow.	*.FGRID
INIT Index	*.INSPEC	The INIT index file for both unified and non-unified formats	*.FINSPEC
INIT	*.INIT	This file type contains static model properties.	*.FINIT
LOG		Output log file that contains a copy of the output from the terminal. The file itself is not generated by the commercial simulator but by an auxiliary program, ECLRUN, that is used to call and execute all the commercial simulators.	*.LOG
PRT		Output print file.	*.PRT
RESTART Index	*.RSSPEC	The RESTART index file for both unified and non-unified formats. This file is not required by OPM Flow or OPM ResInsight to be able to read and write the RESTART data file.	*.FRSSPEC

File Type	Unformatted Extension	Description	Formatted Extension
RESTART Data Non-Unified	*.Xnnnn	The RESTART data files containing the solution arrays requested to be written to the RESTART files. For example: CASENAME.X0001 CASENAME.X0002 CASENAME.X0003 etc. CASENAME.RSSPEC	*.Fnnnn
RESTART Data Unified	*.UNRST	The RESTART data file containing the solution arrays requested to be written to the RESTART file. For example: CASENAME.UNRST CASENAME.RSSPEC	*.FUNRST
RFT	*.RFT	The RFT data file containing wellbore vector data requested to be written to the RFT file. For example: CASENAME.RFT	*.FRFT
RSM		Output RSM file.	*.RSM
SAVE	*.SAVE	The SAVE file type is currently not supported by OPM Flow.	*.FSAVE
SUMMARY ESMRY	*.ESMRY	OPM Flow Enhanced SUMMARY (*.ESMRY) output file format that is optimized for fast loading of selected vectors by post-processing applications. Note that the FMTOUT, MULTOUT, and UNIFOUT keywords have no impact on this file type.	
SUMMARY Index	*.SMSPEC	The SUMMARY index file for both unified and non-unified formats.	*.FSMSPEC
SUMMARY Data Non-Unified	*.Snnnn	The SUMMARY data files containing the variables requested to be written to the SUMMARY files. For example: CASENAME.S0001 CASENAME.S0002 CASENAME.S0003 etc. CASENAME.SMSPEC	*.Annnn
SUMMARY Data Unified	*.UNSMRY	The SUMMARY data file containing the variables requested to be written to the SUMMARY file. For example: CASENAME.UNSMRY CASENAME.SMSPEC	*.FUNSMRY

Notes:

1) The above file naming convention is for Linux type operating systems, as OPM Flow is currently only officially supported for Linux distributions.
2) File Type cells colored in gray indicate the output may be different to the commercial simulators equivalent file type. File Type cells colored in orange represent file types that are not supported by OPM Flow.
3) Formatted and Unformatted Extension cells colored in gray indicate the format is not available or not applicable.

Table F.3: OPM Flow File Naming Conventions

F.1.3 UNFORMATTED FILE FORMAT CONSIDERATIONS

Originally the commercial simulator was written in FORTRAN 77 and the current version is still written in a version of FORTRAN. FORTRAN unformatted file output is dependent on the FORTRAN compiler used to generated the executable program. Typically an unformatted record consists of a four byte prefix outlining the length of the record in bytes, then the actual data record, followed by suffix containing the length of the record in bytes, Most but not all compilers use four bytes. This aids in reading records, for example, the four byte record size at the end of the record assists with a backspace operation. If the record size is greater than two Gigabytes (231 bytes), the record is divide into sub-records. In this case the sign bit of the prefix inform that the record is continued by a sub-record or not and the sign bit of the suffix indicate whether or not there is a preceding sub record.

OPM Flow unformatted files are written using the big-endian mode, that is the most significant value in the sequence is stored at the lowest storage address, that is first. This is opposite to the little-endian mode, where the least significant value in the sequence is stored first. For example, consider the number 1025 (2 to the tenth power plus one) stored in a four byte integer:

```
00000000 00000000 00000100 00000001
```

Address	Big-Endian Representation Of 1025	Little-Endian Representation Of 1025
00	00000000	00000001
01	00000000	00000100
02	00000100	00000000
03	00000001	00000000

Table F.4: Big-Endian and Little-Endian Representation

OPM Flow is written in C++ using the open source GNU C++ compiler and therefore developers using C++ to need to ensure that unformatted files are read or written using the correct big-endian mode.

F.1.4 FILE OUTPUT FORMAT

All the files consist of a series of "header" keywords that defined various parameters based on their format type: integer, real, Boolean, double precision and character. For example for the INT and RESTART files the header keywords are:

1) SEQNUM: Report time step keyword (RESTART file only).
2) INTEHEAD: Integer (four byte) keyword that contains constant values for example, the dimensions of the model, as well as the location and length of data contained in the subsequent property keywords.
3) LOGIHEAD: Boolean or logical (four byte) keyword that consists of a set of either T (true) or F (false) character strings, for example, if dissolved gas is present in the model then set to T for true or F for false.
4) DOUBHEAD: Real double precision keyword (eight byte) that contain real values, for example, the current time step SCHEDULE section TUNING keyword parameter real values. Note that the integer parameters on the TUNING keyword are stored in the INTEHEAD keyword data array.

These are then followed by a series of "property" keywords that that outline the various additional data. For the RESTART files the header keywords are repeated for each time step a RESTART record is written. There are also additional header keywords for Local Grid Refinements ("LGR") that are repeated for each LGR and for time step.

All the keywords follow the same format, including the property keywords, as outlined in Table F.5, which shows a portion of the INTEHEAD header keyword definition for the RESTART file.

No.	Description	Output File Keyword Format			Status Or Value
		Keyword	No. of Entries	Data Type	
	INTEHEAD	Global grid header keyword for the RESTART Data file that defines the integer variables for this time step. NIHEAD is the number of entries associated with this keyword, and is dependent on the features being used in the model. Undefined values should be set to zero.			Required
1-1	Format	INTEHEAD	NIHEAD	INTE	
2-1	Data	An encoded integer corresponding to the time the file was created. For files not originating from the commercial simulator, this value may be set to zero.			ISNUM
2-2	Data	Simulator version, should be set to zero.			VERSION
2-3	Data	Unit system used in the simulation, set to 1 for metric, 2 for field, and 3 for laboratory			
2-4	Data	Undefined.			0
Notes: 1) Rows shaded in gray indicate the start of a keyword. 2) Cells colored red in the No. column indicate that this item is either "Not Used" or "Undefined".					

Table F.5: Output File Keyword Format

The row colored gray in the table specifies the start of a keyword definition and shows the format for the keyword (INTEHEAD) together with a description of the keyword. The next row, item 1-1 in the No. column, outlines the format of the keyword, that is the keyword name (INITHEAD), the number of data entries for the keyword (NIHEAD) and the type of entries (INTE for integers). The next set of rows specify the individual data entries, for example, item 2-3 in the No. column is the third integer value that declares the

unit system for the file. Note that Cells colored red in the No. column indicate that this item is either "Not Used" or "Undefined".

Note

Under the No. column in Table F.5 the count base is one (offset one), as per FORTRAN; however, in C++ the base is zero (offset zero) and therefore C++ programmers must subtract one from this column to obtain the correct reference.

This reference adjustment should be applied to all references in this Appendix.

OPM Flow is written in C++ developers should use offset zero in reviewing or modifying the code.

The Data Type variable in row 1-1 in Table F.5 should have one of the following values:

1) CHAR for eight character words enclosed in single quotes (ASCII character decimal 39) for formatted output.
2) DOUB for double precision eight byte real values.
3) INTE for standard four byte integers.
4) LOGI for standard four byte Boolean (logical) values.
5) MESS keyword that has no associated data.
6) REAL for single precision four byte reals values.

All keywords follow the same format including the property array like PORO and PRESSURE. So for example, the INTEHEAD keyword would be like this in a typical formatted RESTART file:

```
'INTEHEAD'         249 'INTE'
 -955283513      200400           2       -2345       -2345       -2345
      -2345       -2345          10          10           1         100
         10       -2345           7       -2345           0           1
          0           2           2           0           0           0
        110         108         109           3          97          93
      -2345       -2345          19          38          53       -2345
         97          93         146           5           0           1
         15          24           8           5           2           4
  ..............................................
```

And the pressure and water saturation arrays would be of the same form in a typical formatted RESTART file:

```
 'PRESSURE'       44431 'REAL'
   0.26889725E+03   0.26838983E+03   0.26826810E+03   0.26820352E+03
   0.26814493E+03   0.26809378E+03   0.26807767E+03   0.26806363E+03
   0.26802625E+03   0.26798474E+03   0.26795001E+03   0.26791434E+03
   0.26787915E+03   0.26783920E+03   0.26777118E+03   0.26769208E+03
..............................................
 'SWAT    '       44431 'REAL'
   0.10500000E+00   0.10500000E+00   0.10500000E+00   0.10500000E+00
   0.10500000E+00   0.10500000E+00   0.10500000E+00   0.14000000E+00
   0.14000000E+00   0.14500000E+00   0.14500000E+00   0.14500000E+00
   0.14500000E+00   0.14500000E+00   0.16000000E+00   0.16000000E+00
..............................................
```

Secondly, the No. Entries is the number of values following the keyword and as such cannot be used to determine the size of an array that is not associated with the grid, say for example the number of active

connections in a well. Instead, the array size is given in another property keyword. In this case, the number of elements for each well in the IWEL property array is given in the INTEHEAD keyword as shown in Table F.6 (see Table F.21 for a full description of the RESTART INTEHEAD keyword).

No.	Keyword Name	RESTART Data - Header Keywords			Status Or Value
		Keyword	No. of Entries	Data Type	
	INTEHEAD	Global grid header keyword for the RESTART Data file.			Required
1-1	Format	INTEHEAD	NIHEAD	INTE	
2-17	Data	Number of wells in the model.			NWELLS
2-25	Data	Number of values per well in the IWEL array (default value is 97)			NIWELZ

Table F.6: INTEHEAD Keyword - IWEL Example

Now if INTEHEAD(17) was equal to three and INTEHEAD(5) was equal to five then the IWEL property array for the three wells would have the data arranged as shown in Figure F.1.

IWEL Property Array Data Layout
INTEHEAD(17) = 3 and INTEHEAD(25)= 5

Well-1					Well-2					Well-3				
Item 1	Item 2	Item 3	Item 4	Item 5	Item 1	Item 2	Item 3	Item 4	Item 5	Item 1	Item 2	Item 3	Item 4	Item 5
Pos 1	Pos 2	Pos 3	Pos 4	Pos 5	Pos 6	Pos 7	Pos 8	Pos 9	Pos 10	Pos 11	Pos 12	Pos 13	Pos 14	Pos 15

Figure F.1: IWEL Property Keyword Example

Here the Items are the values affiliated with a specific well and Pos(ition) is the actual position in the IWEL property array, shown as gray boxes in Figure F.1.

Similarly for well connections and completions. In this case the maximum number of connections for all the wells is given in INTEHEAD(18) element and a well's current number of connections is given via the IWEL(5) element in the IWEL property array (see Table F.27 for a detailed description of the IWEL keyword).

Now if the number of wells equal two (INTEHEAD(17)) and INTEHEAD(18) is equal to four, that is the maximum number of connections is set to four, together with IWEL(5) equal to three for the first well and four for the second well, the ICON property array for the two wells would have the data arranged as shown in Figure F.2.

ICON Property Array Data Layout
INTEHEAD(17) = 2 and INTEHEAD(18) = 4
Well-1 with IWEL(5) = 3 and Well-2 with IWEL(5) = 4

ICON Array																							
Well-1												Well-2											
Conn-1			Conn-2			Conn-3			Conn-4			Conn-1			Conn-2			Conn-3			Conn-4		
I	J	K	I	J	K	I	J	K	0	0	0	I	J	K	I	J	K	I	J	K	I	J	K
Position																							
1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16	17	18	19	20	21	22	23	24

Figure F.2: ICON Property Keyword Example

Here the connection co-ordinates are given by (I, J, K) and the actual position in the ICON array is shown in the gray colored boxes. Notice that Well-1 has the required maximum number of connections but the fourth connection has the default values of zero, as the IWEL(5) value for this well only has three active connections.

Note also that the number of data elements per connection in the ICON vector is given by NICONZ which is given by INTEHEAD(33) and that the size of the ICON vector is NICONZ x NCWMAX x NWELLS, where NCWMAX is set by INTEHEAD (18) and NWELLS by INTEHEAD(17). In the example, NICONZ was assumed to be three for illustrative purposes, while the actual value is a constant 25 in the most recent versions of the commercial simulator. These arbitrary constants, like NICONZ, use to determine vector lengths, appear to be constant for various versions of the commercial simulator.

The following sections outline the format of the various individual files supported by OPM Flow, except for the DBG, LOG, PRT, and RSM ASCII format files.

F.2 EGRID - MODEL STRUCTURAL DATA FOR IRREGULAR CORNER-POINT GRIDS FILE

EGRID files contain the structural information for the model via the COORD and ZCORN etc., keywords, and employs the extensible GRID format, which is more compact than the GRID formatted files and is the only format supported by OPM Flow.

The output for this file type must be requested via the *GRIDFILE – Set the Grid File Output Options* in the GRID section, as shown below:

```
--
--       GRID FILE OUTPUT OPTIONS
--       GRID    EGRID
--       OPTN    OPTN
GRIDFILE
         0       1                                                      /
```

The above example defines that no GRID file will be written out and that the extensible GRID (that is the EGRID geometry format) file will be produced. This is the only configuration that OPM Flow supports.

The FMTIN and FMTOUT keywords in the RUNSPEC section defines the input and output files to be formatted as ASCII i.e. text files, (*.FEGRID), as oppose to unformatted binary files (*.EGRID). If these keywords are omitted then the default is for unformatted binary file input.

The EGRID file structure consists of a series header keywords that define the header keyword data type, the number of data entries, and then the associated header keyword data. There are a total of four major keyword header types for this file type (1) File Header, (2) Global Grid Header, (3) Local Grid Refinement Header[403] (repeated for each local grid refinement in the model), and (4) Non-Neighbor Connection Header.

In addition to the structural data, this file type also contains information describing the geometry relationship between the global grid and any Local Grid Refinements ("LGR") that are present in the model, together with the Non-Neighbor Connections ("NNC") that may be present due to structural discontinuities. The overall structure of this file is is outlined in Table F.7

Reference Section	Global Section Keywords	Global Data Keywords	LGR Section Keywords	LGR Data Keywords	Notes
F.2.1	FILEHEAD MAPUNITS MAPAXES GRIDUNIT GDORIENT				EGRID header keyword that defines various Global Grid grid properties
F.2.2	GRIDHEAD	COORD ZCORN etc.			EGRID global irregular corner point grid keyword that defines the grid geometry
F.2.2	ENDGRID				Marks the end of the EGRID global irregular corner point grid keyword
F.2.3			LGR LGRPARNT LGRSGRID		EGRID LGR irregular corner point grid keyword that defines the LGR name and optionally the nested or parent LGRs associated with this LGR.
F.2.3			GRIDHEAD	COORD ZCORN etc.	EGRID LGR irregular corner point grid keyword that defines the grid geometry
F.2.3			ENDGRID		Marks the end of the EGRID LGR irregular corner point grid keyword
F.2.3			ENDLGR		Sets the end of current LGR section.
F.2.3			LGR LGRPARNT LGRSGRID		EGRID LGR irregular corner point grid keyword that defines the LGR name and optionally the nested or parent LGRs associated with this LGR
F.2.3			GRIDHEAD	COORD ZCORN etc.	EGRID LGR irregular corner point grid keyword that defines the grid geometry
F.2.3			ENDGRID		End of the EGRID LGR grid section.
F.2.3			ENDLGR		Sets the end of current LGR section.
F.2.4	NNCHEAD	NNC1			EGRID Global Non-Neighbor Connections

[403] *Several keywords are used for both global and LGR grids, for example the GRIDHEAD, COORD, ZCORN keywords.*

Reference Section	Global Section Keywords	Global Data Keywords	LGR Section Keywords	LGR Data Keywords	Notes
		NNC2			for Irregular Corner Point Grid keywords.
F.2.4			NNCHEAD	NNC1	First LGR EGRID global and local non-neighbor connections for irregular corner point grid keywords.
				NNC2	
				NNCL	
				NNCG	
F.2.4			NNCHEAD	NNC1	Second LGR EGRID global and local non-neighbor connections for irregular corner point grid keywords.
				NNC2	
				NNCL	
				NNCG	
F.2.4			NNCHEADA	NNA1	EGRID amalgamated and local non-neighbor connections for irregular corner point grid keywords.
				NNA2	

Notes:

1) Cells shaded in gray indicate the combination is not applicable and cells colored orange indicate that this item is not currently supported by OPM Flow.

Table F.7: EGRID - Model Structural Data for Irregular Corner-Point Grids

Each major header keyword is subdivided into a series of sub-header keywords that define a particular data set. The general format for a header keyword is the keyword (enclosed in single quotes), followed by the number of data entries (an integer value), followed by the data type which is set to CHAR, INTE, or REAL (enclosed in quotes). This will then be followed by the data for the keyword.

The following sections describe the format of the various keywords and the associated data sets.

F.2.1 EGRID HEADER KEYWORDS

The FILEHEAD keyword defines the key attributes of the file via a series of integer constants, this is then followed by the MAPUNITS, MAPAXES, GRIDUNIT and GDORIENT keywords and their data sets. Note that some keywords are optional and are not used by OPM Flow per se, but are used by pre- and post processing software. Secondly, some features are not supported by OPM Flow, for example local grid refinements, and thus OPM will not write out the data associated with these features.

Table F.8 outlines the structure of the EGRID Header Keywords and their affiliated data.

No.	Keyword Name	EGRID Header Keyword Table F.8			Status Or Value
		Keyword	No. of Entries	Data Type	
	FILEHEAD	This keyword marks the start of the EGRID file and defines various parameters associated with this file type.			Required
1-1	Format	FILEHEAD	100	INTE	
2-1	Data	Version number for this file type, for example 3.			3
2-2	Data	The year this version of the file format was released, for example 2004.			2004

No.	Keyword Name	EGRID Header Keyword Table F.8			Status Or Value
		Keyword	No. of Entries	Data Type	
2-3	Data	Not used.			
2-4	Data	Version number of earliest this file format is supported, normally set to 0.			0
2-5	Data	Type of grid, set to 0 for Irregular Corner-Point Grids, 1 for Unstructured Grids, or 2 for a mixture of Irregular Corner-Point and Unstructured Grids (Mixed Grids).			0
2-6	Data	Type of rock model, set to 0 for a single porosity/permeability rock model, 1 for dual porosity rock model, or 3 for a dual permeability rock model.			0
2-7	Data	Original grid format, set to 0 for unknown, 1 for Irregular Corner-Point Grids, or 2 for Cartesian Regular Grids.			
2-8 to 2-100	Data	Not used			0
	MAPUNITS	This keyword defines the MAPUNITS.			Optional
1-1	Format	MAPUNITS	1	CHAR	
2-1	Data	The units of the map data, normally set to FEET or METRES.			
	MAPAXES	The MAPAXES keyword defines the grid position relative to the map coordinates. The six values in the data fields define the relative map (x, y) coordinates for three locations to enable conversion from grid to map coordinate in pre- and post processing software.			Optional
1-1	Format	MAPAXES	6	REAL	
2-1	Data	X-coordinate on the y-axis located at the end of the y-axis.			
2-2	Data	Y-coordinate on the y-axis located at the end of the y-axis.			
2-3	Data	X-coordinate at the origin.			
2-4	Data	Y-coordinate at the origin.			
2-5	Data	X-coordinate on the x-axis located at the end of the x-axis.			
2-6	Data	Y-coordinate on the x-axis located at the end of the x-axis.			
	GRIDUNIT	The GRIDUNIT keyword defines units of the COORD and ZCORN arrays and the grid coordinate space			Required
1-2	Format	GRIDUNIT	2	CHAR	
2-1	Data	The first entry defines units of the COORD and ZCORN arrays and should be normally set to FEET or METRES (enclosed in single quotes).			
2-2	Data	The second entry defines the grid coordinate space (MAP or MAPFT if COORD and ZCORN are defined in map coordinates, or blank if in grid coordinates (‘ ’).			

No.	Keyword Name	EGRID Header Keyword Table F.8			Status Or Value
		Keyword	No. of Entries	Data Type	
	GDORIENT	Grid orientation keyword defines the property ordering in the I, J, and K dimensions, the direction of the Z-direction, and the "handedness of the grid.			Optional
1-1	Format	GDORIENT	5	CHAR	
2-1	Data	Define the property ordering in the I dimension (set to either INC for increasing or DEC for decreasing).			
2-2	Data	Define the property ordering in the J dimension (set to either INC for increasing or DEC for decreasing).			
2-3	Data	Define the property ordering in the K dimension (set to either INC for increasing or DEC for decreasing).			
2-4	Data	Declare the direction of the Z-direction, should be set to either UP or DOWN.			
2-5	Data	Set the "handedness of the grid., should be set to either LEFT or RIGHT.			

Notes:

1) Rows shaded in gray indicate the start of a keyword.
2) Cells colored orange in the No. column indicate that this item is not supported by OPM Flow and those colored red are either "Not Used" or "Undefined".
3) For formatted output all character variables, including the keywords should be enclosed in single quotes, for example the FILEHEAD keyword should be written out as 'FILEHEAD'.
4) GDORIENT keyword in the GRID section is not supported by OPM Flow.

Table F.8: EGRID Header Keywords

Example: EGRID Header Keywords

The following example shows a typical formatted file EGRID header keyword data set taken from the Norne model.

```
 'FILEHEAD'         100 'INTE'
          3        2004           0           0           0           0
          0           0           0           0           0           0
          0           0           0           0           0           0
          0           0           0           0           0           0
          0           0           0           0           0           0
          0           0           0           0           0           0
          0           0           0           0           0           0
          0           0           0           0           0           0
          0           0           0           0           0           0
          0           0           0           0           0           0
          0           0           0           0           0           0
          0           0           0           0           0           0
          0           0           0           0           0           0
          0           0           0           0           0           0
          0           0           0           0           0           0
          0           0           0           0           0           0
          0           0           0           0
 'MAPAXES '           6 'REAL'
   0.00000000E+00   0.10000000E+03   0.00000000E+00   0.00000000E+00
   0.10000000E+03   0.00000000E+00
 'GRIDUNIT'           2 'CHAR'
 'METRES  ' '        '
```

Notice that the GDORIENT keyword is not written out.

F.2.2 EGRID GLOBAL IRREGULAR CORNER POINT GRID KEYWORDS

Following the EGRID Header Keyword data set, the EGRID Global Irregular Corner Point Grid Keywords should be read or written to the EGRID file. This data set includes the actual structural data for the model, that is the COORD and ZCORN keyword data, as well as the ACTNUM keyword data that defines if a given cell is active (set to 1), or inactive (set to zero). The structure for this type data is defined in Table F.9.

No.	Keyword Name	EGRID Global Irregular Corner Point Grid Keywords Table F.9			Status Or Value
		Keyword	No. of Entries	Data Type	
	GRIDHEAD	Global grid keyword that defines the start of this keyword type.			Required
1-1	Format	GRIDHEAD	100	INTE	
2-1	Data	Type of grid, set to 0 for a mixture of Irregular Corner-Point and Unstructured Grids, 1 for. Irregular Corner-Point Grids, and 2 for Unstructured Grids. Only the default value of one is supported by OPM Flow.			1
2-2	Data	The number of grid blocks in the x-direction (NX).			
2-3	Data	The number of grid blocks in the y-direction (NY).			
2-4	Data	The number of grid blocks in the z-direction (NZ).			
2-5	Data	The grid reference number set to zero for the global grid or a value greater than zero to represent a LGR. For example if there are five LGR's, then this complete data set is repeated six times, once for the global grid, and five times for the five LGRs., with the first LGR having a reference number of 1 and the last LGR having a reference number of 5.			
2-6 to 2-24	Data	Not used.			0
2-25	Data	NUMRES the number of reservoirs, that is the number COORD data sets to be processed by OPM Flow. This should be set to one as the only value supported by OPM Flow			1
2-26	Data	NSEG the number of coordinate line segments, should be set to 1.			1
2-27	Data	Cartesian/Radial grid indicator set to 0 for Cartesian grids and greater than zero for radial grids. Currently OPM Flow does not support radial geometries so this value should always be set to zero.			0
2-28	Data	The LGR location of the lower I-index LGR in the global grid. This value should be set to zero if there is no LGR grids.			0
2-29	Data	The LGR location of the lower J-index LGR in the global grid. This value should be set to zero if there is no LGR grids.			0
2-30	Data	The LGR location of the lower K-index LGR in the global grid. This value should be set to zero if there is no LGR grids.			0
2-31	Data	The LGR location of the upperI -index LGR in the global grid. This value should be set to zero if there is no LGR grids.			0
2-32	Data	The LGR location of the upper J-index LGR in the global grid. This value should be set to zero if there is no LGR grids.			0

No.	Keyword Name	EGRID Global Irregular Corner Point Grid Keywords Table F.9			Status Or Value
		Keyword	No. of Entries	Data Type	
2-33	Data	The LGR location of the uper K-index LGR in the global grid. This value should be set to zero if there is no LGR grids.			0
2-34 to 2-100	Data	Not used by OPM Flow.			0
	BOXORIG	Not supported.			Optional
1-1	Format	BOXORIG	3	INTE	
2-1 to 2-3	Data	Ignored by OPM Flow.			
	COORD	Property keyword that defines the start of the COORD data set. The COORD data defines a set of coordinate lines or pillars for a reservoir grid via an array. The number of data values for this keyword should be 6 x (NX+1) x (NY+1) x NUMRES. Where NUMRES represents the number of separate coordinate lines (or reservoirs). In OPM Flow NUMRES can only be set to one.			Required
1-1	Format	COORD	6 x (NX+1) x (NY+1)	REAL	
2-1	Data	COORD data in the same format as the COORD keyword described in the GRID section of the manual.			
	COORDSYS	Coordinate definitions for each NUMRES reservoir. The number of data values for this keyword should be 6 x NUMRES. Where NUMRES represents the number of separate coordinate lines (or reservoirs). In OPM Flow NUMRES can only be set to one.			Optional
1-1	Format	COORDSYS	6	INTE	
2-1 to 2-6	Data	Ignored by OPM Flow.			
	ZCORN	Property keyword that defines the start of the ZCORN data set. ZCORN defines the depth of each corner point of a grid block on the pillars defining the reservoir grid. The number of data values for this keyword should be 8 x NX x NY x NZ.			Required
1-2	Format	ZCORN	8 x NX x NY x NZ	REAL	
1-3	Type	Type of data in the Data fields.			
2-1	Data	ZCORN data in the same format as the ZCORN keyword described in the GRID section of the manual.			

No.	Keyword Name	EGRID Global Irregular Corner Point Grid Keywords Table F.9			Status Or Value
		Keyword	No. of Entries	Data Type	
	ACTNUM	Property keyword that defines the start of the ACTNUM data set. ACTNUM specifies which grid blocks are active or inactive. A value of 1 indicates the block is active and a value of 0 indicates the block is inactive. Although this data set status is set to Optional, it is normally always written out by pre-processing software. The number of data values for this keyword should be NX x NY x NZ integer values.			Optional
1-1	Format	ACTNUM	NX x NY x NZ	INTE	
2-1	Data	ACTNUM data in the same format as the ACTNUM keyword described in the GRID section of the manual.			
	CORSNUM	Property keyword that defines the start of the CORSNUM data set for when the grid has been coarsened. The number of data values for this keyword should be NX x NY x NZ integer values. Grid coarsening is currently not supported in OPM Flow.			Optional
1-1	Format	CORSNUM	NX x NY x NZ	INTE	
2-1	Data	CORSNUM data set.			
1-1	ENDGRID	The ENDGRID keyword marks the end of the these set of keywords. Always set to 0 to indicate that there is no data section following this keyword.			Required
1-2	Format	ENDGRID	0	INTE	0

Notes:

1) Rows shaded in gray indicate the start of a keyword.
2) Cells colored orange in the No. column indicate that this item is not supported by OPM Flow and those colored red are either "Not Used" or "Undefined".
3) For formatted output all character variables, including the keywords, should be enclosed in single quotes, for example the GRIDHEAD keyword should be written out as 'GRIDHEAD'.

Table F.9: EGRID Global Irregular Corner Point Grid Keywords

Example: EGRID Global Irregular Corner Point Grid Keywords

The following example shows a typical formatted EGRID Global Irregular Corner Point Grid Keywords data set, taken from the Norne model.

```
'GRIDHEAD'         100 'INTE'
           1          46         112          22           0           0
           0           0           0           0           0           0
           0           0           0           0           0           0
           0           0           0           0           0           0
           1           1           0           0           0           0
           0           0           0           0           0           0
           0           0           0           0           0           0
           0           0           0           0           0           0
           0           0           0           0           0           0
           0           0           0           0           0           0
           0           0           0           0           0           0
           0           0           0           0           0           0
           0           0           0           0           0           0
           0           0           0           0           0           0
           0           0           0           0           0           0
           0           0           0           0           0           0
           0           0           0           0
 'COORD   '       31866 'REAL'
  0.45311400E+06   0.73199210E+07   0.30374729E+04   0.45311400E+06
  0.73199210E+07   0.31328311E+04   0.45315503E+06   0.73198400E+07
  0.29839331E+04   0.45314275E+06   0.73198640E+07   0.31735720E+04
  0.45319609E+06   0.73197590E+07   0.30059690E+04   0.45317150E+06
  0.73198075E+07   0.32158359E+04   0.45323716E+06   0.73196780E+07
  0.30002649E+04   0.45320025E+06   0.73197510E+07   0.32172500E+04
  0.45327819E+06   0.73195970E+07   0.29893479E+04   0.45322903E+06
  0.73196940E+07   0.32139509E+04   0.45331925E+06   0.73195165E+07
.....
 'ZCORN   '      906752 'REAL'
  0.30374729E+04   0.29839331E+04   0.29839331E+04   0.30059690E+04
  0.30059690E+04   0.30002649E+04   0.30002649E+04   0.29893479E+04
  0.29893479E+04   0.29956799E+04   0.29956799E+04   0.30008550E+04
  0.30008550E+04   0.30052520E+04   0.30052520E+04   0.30308621E+04
  0.30308621E+04   0.30368701E+04   0.30368701E+04   0.30380171E+04
  0.30380171E+04   0.30450271E+04   0.30450271E+04   0.30554099E+04
  0.30554099E+04   0.30665410E+04   0.30665410E+04   0.30766240E+04
  0.30766240E+04   0.30869380E+04   0.30869380E+04   0.30961531E+04
.....
 'ACTNUM  '      113344 'INTE'
           0           0           0           0           0           0
           0           0           0           0           0           0
           0           0           0           0           0           0
           0           0           0           0           0           0
           0           0           0           0           0           0
           0           0           0           0           0           0
           0           0           0           0           0           0
           0           0           0           0           0           0
           0           0           0           0           0           0
           0           0           0           0           0           0
.....
 'ENDGRID '           0 'INTE'
```

End of Example

F.2.3 EGRID LGR IRREGULAR CORNER POINT GRID KEYWORDS

The data structure for Local Grid Refinement ("LGR") Irregular Corner Point Grids is similar to the global grid format described in the previous section. Additional data that the defines the LGR properties (LGR Name for example) are included in this definition and the LGR keywords are repeated for each LGR in the model. The keyword description is outlined in Table F.10.

Note that currently OPM Flow does not support LGR grids and therefore this series of keywords cannot be used by OPM Flow.

No.	Keyword Name	EGRID LGR Irregular Corner Point Grid Keywords Table F.10 (Repeated For Each LGR In The Model)			Status Or Value
		Keyword	No. of Entries	Data Type	
	LGR	LGR keyword defines the start of the LGR section of keywords and is required when LGRs are in the model.			Required
1-1	Format	LGR	1	CHAR	
2-1	Data	Name of the LGR			
	LGRPARNT	Start of LGRPARNT keyword that defines the parent LGR for when nested LGR have been used.			Optional
1-1	Format	LGRPARNT	1	CHAR	
2-1	Data	Name of parent LGR			
	LGRSGRID	Start of the LGRSGRID keyword for LGRs parented from a sub-grid.			Optional
1-1	Format	LGRSGRID	1	CHAR	
2-1	Data	Name of parent LGR			
	GRIDHEAD	LGR grid keyword that defines the start of this keyword type for the LGR. The format is the same as the Global grid keyword - see Table F.9.			Required
	BOXORIG	Not supported. The format is the same as the Global grid keyword - see Table F.9.			Optional
	COORD	Property keyword that defines the start of the LGR COORD data set. The COORD data defines a set of coordinate lines or pillars for LGR grid via an array. The format is the same as the Global grid keyword - see Table F.9.			Required
	COORDSYS	Coordinate definitions for the LGR. The format is the same as the Global grid keyword - see Table F.9.			Optional
	ZCORN	Property keyword that defines the start of the LGR ZCORN data set. The format is the same as the Global grid keyword - see Table F.9.			Required
	ACTNUM	Property keyword that defines the start of the LGR ACTNUM data set. The format is the same as the Global grid keyword - see Table F.9.			Optional

No.	Keyword Name	EGRID LGR Irregular Corner Point Grid Keywords Table F.10 (Repeated For Each LGR In The Model)			Status Or Value
		Keyword	No. of Entries	Data Type	
	HOSTNUM	Property keyword that defines the start of the HOSTNUM data set. which is only applicable to LGR grids. Number of data values for this keyword should be NX x NY x NZ integer values.			Required
1-1	Format	HOSTNUM	NX x NY x NZ	INTE	
2-1	Data	HOSTNUM data set.			
	ENDGRID	The ENDGRID keyword marks the end of the LGR Grid Header section. Always set to 0 to indicate that there is no Data section for this keyword.			Required
1-1	Format	ENDGRID	0	INTE	0
	ENDLGR	ENDLGR keyword marks the end of the LGR Header section. Always set to 0 to indicate that there is no Data section for this keyword.			Required
1-1	Format	ENDLGR	0	INTE	0

Notes:

1) Rows shaded in gray indicate the start of a keyword.
2) Cells colored orange in the No. column indicate that this item is not supported by OPM Flow.
3) For formatted output all character variables, including the keywords, should be enclosed in single quotes, for example the LGR keyword should be written out as 'LGR '.

Table F.10: EGRID LGR Irregular Corner Point Grid Keywords

Note that this data set is repeated for each LGR in the model as demonstrated in the following example.

Example: EGRID LGR Irregular Corner Point Grid Keywords

The following example shows a typical formatted data set for this series of keywords with two Cartesian LGR grids, named LGR-1 and LGR-2 define via the following CARFIN keyword statements:

```
--
--       CARFIN LGR GRID COMMANDS
--
--       LGR        ----- FINE GRID ------    -- CARFIN GRID --   MAX     HOST
--       NAME       I1  I2  J1  J2  K1  K2     NX    NY    NZ   WELLS   NAME
CARFIN
       'LGR-1'      2   2   2   2   1   1      2     2     2     1    GLOBAL /
CARFIN LGR GRID PARAMETERS

--
--       CARFIN LGR GRID COMMANDS
--
--       LGR        ----- FINE GRID ------    -- CARFIN GRID --   MAX     HOST
--       NAME       I1  I2  J1  J2  K1  K2     NX    NY    NZ   WELLS   NAME
CARFIN
       'LGR-2'      9   9   9   9   1   1      2     2     2     1    GLOBAL /
CARFIN LGR GRID PARAMETERS

ENDFIN
```

The resulting keywords are as follows:

```
'LGR     '           1 'CHAR'
'LGR-1   '
'LGRPARNT'           1 'CHAR'
'        '
'GRIDHEAD'         100 'INTE'
         1           2           2           2           1           0
         0           0           0           0           0           0
         0           0           0           0           0           0
         0           0           0           0           0           0
         1           1           0           2           2           1
 ..........................................
          0           0           0           0
'COORD   '          54 'REAL'
  0.10000000E+04   0.10000000E+04   0.00000000E+00   0.10000000E+04
  0.10000000E+04   0.10000000E+04   0.15000000E+04   0.10000000E+04
  0.00000000E+00   0.15000000E+04   0.10000000E+04   0.10000000E+04
  0.20000000E+04   0.10000000E+04   0.00000000E+00   0.20000000E+04
  0.10000000E+04   0.10000000E+04   0.10000000E+04   0.15000000E+04
 ..........................................
   0.20000000E+04   0.10000000E+04
'ZCORN   '          64 'REAL'
  0.83250000E+04   0.83250000E+04   0.83250000E+04   0.83250000E+04
  0.83250000E+04   0.83250000E+04   0.83250000E+04   0.83250000E+04
  0.83250000E+04   0.83250000E+04   0.83250000E+04   0.83250000E+04
  0.83250000E+04   0.83250000E+04   0.83250000E+04   0.83250000E+04
  0.83350000E+04   0.83350000E+04   0.83350000E+04   0.83350000E+04
 ..........................................
   0.83450000E+04   0.83450000E+04   0.83450000E+04   0.83450000E+04
'ACTNUM  '           8 'INTE'
         1           1           1           1           1           1
         1           1
'HOSTNUM '           8 'INTE'
        12          12          12          12          12          12
        12          12
'ENDGRID '           0 'INTE'
'ENDLGR  '           0 'INTE'
'LGR     '           1 'CHAR'
'LGR-2   '
'LGRPARNT'           1 'CHAR'
'        '
'GRIDHEAD'         100 'INTE'
         1           2           2           2           2           0
  ..........................................
         0           0           0           0
'COORD   '          54 'REAL'
  0.80000000E+04   0.80000000E+04   0.00000000E+00   0.80000000E+04
  0.80000000E+04   0.10000000E+04   0.85000000E+04   0.80000000E+04
  ..........................................
  0.90000000E+04   0.10000000E+04
'ZCORN   '          64 'REAL'
  0.83250000E+04   0.83250000E+04   0.83250000E+04   0.83250000E+04
  ..........................................
  0.83450000E+04   0.83450000E+04   0.83450000E+04   0.83450000E+04
'ACTNUM  '           8 'INTE'
         1           1           1           1           1           1
         1           1
'HOSTNUM '           8 'INTE'
        89          89          89          89          89          89
        89          89
'ENDGRID '           0 'INTE'
'ENDLGR  '           0 'INTE'
```

End of Example

F.2.4 EGRID NON-NEIGHBOR CONNECTIONS FOR IRREGULAR CORNER POINT GRID KEYWORDS

The keywords in this section are used to define Non-Neighbor Connections ("NNC") for both the global grid and any LGR's included in the model. This data set is only written out if there are NNCs in the model and can be skipped if this is the case; however, apart from very simple models, nearly all full field models will have NNCs as a result of modeling the structural faults in the model. Likewise, the keywords affiliate with LGRs are also skipped if the there are no LGRs in the model.

Note that actual NNC transmissibility data for the NNCs (TRANX, TRANY and TRANZ) are stored in the INIT Data File and not in the EGRID File.

The keyword definitions are tabulated in Table F.11.

No.	Keyword Name	EGRID Non-Neighbor Connections Keywords For Irregular Corner Point Grids Table F.11			Status Or Value
		Keyword	No. of Entries	Data Type	
	NNCHEAD	This keyword marks the start of the non-neighbor keywords and defines the number of non-neighbor connections.			Required
1-1	Format	NNCHEAD	10	INTE	
2-1	Data	The number of non-neighbor connections (NUMNNC)			
2-2	Data	Grid identifier set to zero for the global grid, or a value greater than zero to represent a LGR.			
2-3 to 2-10	Data	Not used.			
	NNC1	NNC1 defines the cell numbers for the UPSTREAM non-neighbor connections, where NUMNNC is the number of non-neighbor connections.			Required
1-1	Format	NNC1	NUMNNC	INTE	
2-1	Data	NUMNNC integer values defining the cell numbers for the UPSTREAM non-neighbor connections.			
	NNC2	NNC2 defines the cell numbers for the DOWNSTREAM non-neighbor connections, where NUMNNC is the number of non-neighbor connections			Required
1-1	Format	NNC2	NUMNNC	INTE	
2-1	Data	NUMNNC integer values defining the cell numbers for the DOWNSTREAM non-neighbor connections.			
	NNCL	This keyword defines the local cells connected to the global grid, where NCONGL is the number of entries.			Optional
1-1	Format	NNCL	NCONCL	INTE	
2-1	Data	NCONCL integer values defining the local cells connected to the global grid.			

No.	Keyword Name	EGRID Non-Neighbor Connections Keywords For Irregular Corner Point Grids Table F.11			Status Or Value
		Keyword	No. of Entries	Data Type	
	NNCG	This keyword defines the global cells connected to the current local grid being defined by this section of keywords, where NCONGL is the number of entries.			Optional
1-1	Format	NNCG	NCONGL	INTE	
2-1	Data	NCONGL integer values defining the global cells connected to the current local grid.			
	NNCHEADA	This keyword defines the connection between two LGRs that have been amalgamated.			Optional
1-1	Format	NNCHEADA	10	INTE	
2-1	Data	LGR index of the first LGR in the amalgamation (ILOC1).			
2-2	Data	LGR index of the second LGR in the amalgamation (ILOC2).			
2-3 to 2-10		Not used			
	NNA1	This keyword defines the ILOC1 cells connected in the amalgamation, where NUMNCA is the number of entries.			Optional
1-1	Format	NNA1	NUMNCA	INTE	
2-1	Data	NUMNCA integer values defining the ILOC1 cells connected in the amalgamation.			
	NNA2	This keyword defines the ILOC2 cells connected in the amalgamation, where NUMNCA is the number of entries.			
1-1	Format	NNA2	NUMNCA	INTE	
2-1	Data	NUMNCA integer values defining the ILOC2 cells connected in the amalgamation.			

Notes:

1) Rows shaded in gray indicate the start of a keyword.
2) Cells colored orange in the No. column indicate that this item is not supported by OPM Flow and those colored red are either "Not Used" or "Undefined".
3) For formatted output all character variables, including the keywords, should be enclosed in single quotes, for example the NNCHEAD keyword should be written out as 'NNCHEAD'.
4) Local Grid Refinements are currently not supported by OPM Flow and neither therefore is LGR amalgamation of LGRs.

Table F.11: EGRID NNC Keywords for Irregular Corner Point Grids

Note that if LGR's are present in the model then the keywords relating to LGRs are repeated for each LGR in the model.

Example: EGRID Grid Keywords for Non-Neighbor Connections for Irregular Corner Point Grids

The example shows the NNCs for the global grid and two LGRs.

```
 'NNCHEAD '          10 'INTE'
      11363           0           0           0           0           0
          0           0           0           0
 'NNC1    '       11363 'INTE'
       5717        5717       21173       26325       31477       36629
      41781       46933       52085       57237       62389       67541
      72693       77845       82997       88149       93301      103605
……..……..
 'NNC2    '       11363 'INTE'
        566       10870       26326       31478       36630       41782
      46934       52086       57238       62390       67542       72694
      77846       82998       88150       93302       98454       98454
     103606         612        5764       26372       21220       31524
……..……..

 'NNCHEAD '          10 'INTE'
          0           1           0           0           0           0
          0           0           0           0
 'NNC1    '           0 'INTE'
 'NNC2    '           0 'INTE'
 'NNCL    '          16 'INTE'
          1           5           3           7           1           5
          2           6           2           6           4           8
          3           7           4           8
 'NNCG    '          16 'INTE'
         11          11          11          11           2           2
          2           2          13          13          13          13
         22          22          22          22
 'NNCHEAD '          10 'INTE'
          0           2           0           0           0           0
          0           0           0           0
 'NNC1    '           0 'INTE'
 'NNC2    '           0 'INTE'
 'NNCL    '          16 'INTE'
          1           5           3           7           1           5
          2           6           2           6           4           8
          3           7           4           8
 'NNCG    '          16 'INTE'
         88          88          88          88          79          79
         79          79          90          90          90          90
         99          99          99          99
```

End of Example

F.3 EGRID - MODEL STRUCTURAL DATA FOR UNSTRUCTURED GRIDS FILE

This file format is currently not supported by OPM Flow.

F.4 EGRID - MODEL STRUCTURAL DATA FOR MIXED GRIDS FILE

This file format is currently not supported by OPM Flow.

F.5 GRID - MODEL STRUCTURAL DATA FILE

This file format is currently not supported by OPM Flow.

F.5.1 RESTART DATA - LGR GRID HEADER KEYWORDS

This set of keywords define an LGR's properties, LGR Name for example, and always proceeds an LGR data set for a given LGR; that is the keywords are repeated for each LGR data set in the model and each time step. Note that a given LGR data set is terminated by the ENDLGR keyword and the SEQNUM keyword terminates the input for a time step, which switches the input back to the global grid properties. The keyword description is outlined in Table F.12.

Note that currently OPM Flow does not support LGR grids and therefore this set of keywords cannot be used by OPM Flow.

No.	Keyword Name	RESTART Data - LGR Grid Header Keywords Table F.12 (Repeated For Each LGR In The Model)			Status Or Value
		Keyword	No. Of Entries	Data Type	
	LGR	LGR grid header keyword that defines the name of the LGR.			Always Required
1-1	Keyword	LGR	1	CHAR	
2-1	Data	Name of the LGR.			
	LGRHEADI	LGR that defines the integer variables for this time step.			Always Required
1-1	Format	LGRHEADI	45	INTE	
2-1 to 2-45	Data	Undefined.			
	LGRHEADQ	LGR grid keyword that defines the logical variables (T for true and F for false) for this time step.			Always Required
1-1	Format	LGRHEADQ	5	LOGI	
2-1 to 2-5	Data	Undefined.			

No.	Keyword Name	RESTART Data - LGR Grid Header Keywords Table F.12 (Repeated For Each LGR In The Model)			Status Or Value
		Keyword	No. Of Entries	Data Type	
	LGRHEADD	LGR grid keyword that defines the double precision REAL variables for this time step.			Always Required
1-1	Format	LGRHEADD	5	DOUB	5
2-1 to 2-5	Data	Undefined.			
	ENDLGR	The ENDLGR keyword marks the end of an LGR section for a given LGR.			Always Required
1-1	Format	ENDLGR	1	MESG	
	Data	LGR number.			

Notes:

1) Rows shaded in gray indicate the start of a keyword.
2) Cells colored orange in the No. column indicate that this item is not supported by OPM Flow and those colored red are either "Not Used" or "Undefined".
3) For formatted output all character variables, including the keyword, should be enclosed in single quotes, for example the LGR keyword should be written out as 'LGR '.

Table F.12: RESTART Data - LGR Grid Header Keywords

This keyword type is repeated for each LGR in the model with the last LGR data set terminated by the LGSGONE keyword to mark the end of the LGR input.

Note

The LGR, LGRHEADI, LGRHEADQ and LGRHEADD always proceed LGR data sets and the ENDLGR keyword terminates the end of a data section of LGR data.

Example: RESTART Data - LGR Grid Header Keywords

The following example shows a typical formatted LGR grid header data set for a single LGR grid named LGR-1.

```
'LGR     '           1 'CHAR'
'LGR-1   '
'LGRHEADI'          45 'INTE'
         1         100       -2345       -2345       -2345       -2345
     -2345       -2345       -2345       -2345       -2345       -2345
     -2345       -2345       -2345       -2345       -2345       -2345
     -2345       -2345       -2345       -2345       -2345       -2345
     -2345       -2345       -2345       -2345       -2345       -2345
     -2345       -2345       -2345       -2345       -2345       -2345
     -2345       -2345       -2345       -2345       -2345       -2345
     -2345       -2345       -2345
'LGRHEADQ'           5 'LOGI'
 F  F  F  F  F
'LGRHEADD'           5 'DOUB'
  0.00000000000000D+00  -0.10000000200409D+21  -0.10000000200409D+21
 -0.10000000200409D+21  -0.10000000200409D+21
```

End of Example

F.6 INIT – MODEL INITIALIZATION AND STATIC DATA FILE

The INIT files contain the static data specified in the GRID, PROPS and REGIONS sections. For example, the PORO, PERM and NTG arrays from the GRID section and the resulting PORV and TRANX, TRANY and TRANZ arrays. The data is used in post-processing software, for example OPM ResInsight, to visualize the static grid properties and the tabular PVT and relative permeability data, and is not used per se by the simulator.

The data is written to file if the INIT keyword in the GRID section has been activated (*INIT – Activate the INIT File Option)*. Similar to the EGRID file, the INIT file can either be written out in formatted form as ASCII i.e. text files, if the FMTOUT keyword has been activated, or binary format if the FMTOUT keyword has not been activated. Normally, this option is always activated by the user and when activated the binary form of the file is used.

There are two files types associated with the INIT data:

1) an index file (INIT Index) that contains a list of variables, the array type (character, integer, or real) and the size of the arrays that are written out to the initialization data file and,

2) the initialization data file (INIT Data) that contains the actual data written out at the beginning of the run once the model has been initialized.

The commercial simulator writes out both files, whereas OPM Flow only writes out the INIT Data file, as the data can be read by most post-processing software, including OPM ResInsight, using only the INIT Data file as input.

The following sections outline the format for these two file types.

F.6.1 INIT INDEX FILE

This file type is not supported or required by OPM Flow or OPM ResInsight to read or write the static data written to the INIT Data file.

F.6.2 INIT DATA FILE SPECIFICATION

This file type contains the global and any LGR grid property data present in the model[404], for example porosity (PORO) and permeability data arrays (PERMX,PERMY, and PERMZ) for properties allocated to each grid cell, as well as the fluid and rock property functions tabular data. The overall structure of this file is similar to the RESTART file with the individual keywords being structured as shown in Table F.13.

Reference Section	Global Section Keywords	Global Data Keywords	LGR Section Keywords	LGR Data Keywords	Notes
F.6.3	INTEHEAD LOGIHEAD DOUBHEAD				Global grid header keywords that defines the integer, logical and double precision variables for this header keyword.
F.6.4	Property Keywords	PORV PORO etc.			Global grid property keywords that defines the global grid property data defined in the GRID and Edit sections. For example, PORV, NTG, PERMX etc.)
F.6.5			LGR LGRHEADI LGRHEADQ LGRHEADD		Defines the start of an LGR section of keywords and the LGR header keywords
F.6.5			INTEHEAD LOGIHEAD DOUBHEAD		LGR grid header keywords defining the same data as the global keywords, except for the given LGR. This set of keywords are only written once for each LGR.
F.6.4			Property Keywords	PORV PORO etc.	LGR property keywords; same format as the global grid keywords.
F.6.5			LGR LGRHEADI LGRHEADQ LGRHEADD		The LGR header keywords
F.6.5			INTEHEAD LOGIHEAD DOUBHEA		The LGR header keywords which are only written once for each LGR.
F.6.4			Property Keywords	PORV PORO etc.	LGR property keywords
			etc.		Repeated for each LGR in the model.
F.6.5			LGRSGONE		Marks the end of LGR section of keywords.
F.6.7	TABDIMS				TABDIMS defines the tables dimensions, for the subsequent TAB keywords.

[404] *Several keywords are used for both global and LGR grids, for example the INTEHEAD, LOGIHEAD, DOUBHEAD, PROPERTY and REGION keywords and arrays.*

Reference Section	Global Section Keywords	Global Data Keywords	LGR Section Keywords	LGR Data Keywords	Notes
F.6.8	TAB				Rock and fluid tabular property data keywords.
F.6.8	CON				CON keyword defines the constant PVT dead oil data (the data associated with RSCONST, RSCONSTT, RVCONT and RVCONSTT keywords).
F.6.4	Region Keywords	PVTNUM SATNUM etc.			Region property data for the global grid as defined in the REGION section (PVTNUM, SATNUM etc.). The keyword is repeated to account for all region property data.
F.6.5			LGR LGRHEADI LGRHEADQ LGRHEADD		The LGR header keywords
F.6.4			Region Keywords	PVTNUM SATNUM etc.	LGR region property data for the LGR grid.
F.6.5			LGR LGRHEADI LGRHEADQ LGRHEADD		The LGR header keywords
F.6.4			Region Keywords	PVTNUM SATNUM etc.	LGR region property data for the LGR grid.
			etc.		Repeated for each LGR in the model.
F.6.5			LGRSGONE		Marks the end of LGR section of keywords.
F.6.4	End-Point Keywords	SWATINIT SWL etc.			Global grid saturation and end-point data keywords. The keyword uses the same keywords outlined in the GRID and PROPS section, for example, the SWL, SWATINIT, KRG, PCW, etc. arrays
F.6.5			LGR LGRHEADI LGRHEADQ LGRHEADD		The LGR header keywords
F.6.4			End-Point Keywords	SWATINIT SWL etc.	LGR grid saturation and end-point data keywords.

Reference Section	Global Section Keywords	Global Data Keywords	LGR Section Keywords	LGR Data Keywords	Notes
F.6.5			LGR LGRHEADI LGRHEADQ LGRHEADD		The LGR header keywords
F.6.4			End-Point Keywords	SWATINIT SWL etc.	LGR grid saturation and end-point data keywords.
			etc.		
F.6.5			LGRSGONE		Marks the end of LGR section of keywords.
F.6.4	NNC Keywords	TRANNC HEATNNC etc.			Non-neighbor connection property data ("NNC") for the global grid. The keyword is repeated to account for various NNC arrays.
F.6.5			LGR LGRHEADI LGRHEADQ LGRHEADD		The LGR header keywords
F.6.4			NNC Keywords	TRANNC TRANGL etc.	LGR NNC property data.
F.6.5			LGR LGRHEADI LGRHEADQ LGRHEADD		The LGR header keywords
F.6.4			NNC Keywords	TRANNC TRANGL etc.	LGR NNC property data.
			etc.		Repeated for each LGR in the model.
F.6.5			LGRSGONE		Marks the end of LGR section of keywords.

Notes:

1) Cells shaded in gray indicate the combination is not applicable and cells colored orange indicate that this item is not currently supported by OPM Flow.

Table F.13: INIT Data File Structure

F.6.3 INIT DATA – STATIC DATA HEADER KEYWORDS

The INIT Data – Static Data Header Keyword section is the first data set that should be read or written to the INIT Data file. This data set includes INTEHEAD, LOGIHEAD and DOUBHEAD keywords that define versus parameters used in subsequent keywords in the INIT Data file. This is then followed by a series of global grid PROPERTY data keywords that define the various global grid property arrays for each grid cell in the model, as well as the static tabular data (relative permeability tables and PVT tables), etc.

The structure for this set of keywords is defined in Table F.14.

No.	Keyword Name	INIT Data – Static Data Header Keywords Table F.14			Status Or Value
		Keyword	No. of Entries	Data Type	
	INTEHEAD	Global static header keyword for the INIT Data file that defines the start of INIT static data set, and contains the integer variables for this header keyword. NIHEAD is the number of entries associated with this keyword, and is dependent on the features being used in the model.			Required
1-1	Format	INTEHEAD	NIHEAD	INTE	
2-1		Undefined			0
2-2		Undefined			0
2-3	Data	Unit system used in the simulation, set to 1 for metric, 2 for field, and 3 for laboratory.			
2-4 to 2-8		Undefined.			0
2-9	Data	DIMENS key word in the RUNSPEC section - the number of grid blocks in the x-direction.			NX
2-10	Data	DIMENS key word in the RUNSPEC section - the number of grid blocks in the y-direction.			NY
2-11	Data	DIMENS key word in the RUNSPEC section - the number of grid blocks in the z-direction.			NZ
2-12	Data	Number of global active cells in the model.			NACTIV
2-13	Data	Not used.			0
2-14	Data	Type of grid, set to 0 for Irregular Corner-Point Grids, 1 for Unstructured Grids, or 2 for a mixture of Irregular Corner-Point, Unstructured Grids (Mixed Grids) and 3 for Cartesian grids.			1
2-15	Data	Type of phases in the model, set to 1 for oil, 2 for water, 3 for oil-water, 4 for gas, 5 for oil-gas, 6 for gas-water and 7 for oil-water-gas.			PHASE
2-16 to 2-64		Undefined.			0
2-65	Data	First part of start date of the run, DAY, the value should be greater than or equal to one and less than or equal to 31 for the calendar days.			DAY

No.	Keyword Name	INIT Data – Static Data Header Keywords Table F.14			Status Or Value
		Keyword	No. of Entries	Data Type	
2-66	Data	Second part of start date of the run, MONTH, the value should be greater than or equal to one and less than or equal to 12 for the calendar month.			MONTH
2-77	Data	Third part of start date of run, YEAR, a positive four digit integer value of the start year, which must be specified fully by four digits, that is 1986.			YEAR
2-68 to 2-94		Undefined.			0
2-95	Data	Simulator code used to generate the file, set to 100 for Schlumberger's ECLIPSE 100, 300 for Schlumberger's ECLIPSE 300, 500 for Schlumberger's ECLIPSE 300 Thermal, 700 for Schlumberger's INTERSECT simulator, 800 for Schlumberger's FrontSim simulator, or a negative value for other simulators. OPM Flow uses a value of 100.			IPROG
2-96 to 206		Undefined.			0
2-207	Data	The first part of the current simulation time in the form HH;MM:SS, that is HH for hours. The value should be greater than or equal to zero and less than or equal to 23.			IHOURZ
2-208	Data	The second part of the current simulation time in the form HH;MM:SS, that is MM for minutes. The value should be greater than or equal to zero and less than or equal to 59.			IMINTS
2-209 to 410		Undefined.			0
411	Data	The third part of the current simulation time in the form HH;MM:SS, that is SS for seconds. The value should be greater than or equal to zero and less than or equal to 59.			ISECND
	LOGIHEAD	Global static data keyword for the INIT Data file that defines the logical variables (T for true and F for false) for this keyword. NLHEAD is the number of entries associated with this keyword, and is dependent on the features being used in the model. Undefined items should be set to F (False).			Required
1-1	Format	LOGIHEAD	NLHEAD	LOGI	
2-1	Data	Dissolved gas is present in the model indicator, set to T if present in the model else set to F.			DISGAS
2-2	Data	Vaporized oil is present in the model indicator, set to T if present in the model else set to F.			VAPOIL
2-3	Data	SATOPTS in the RUNSPEC section - directional relative permeability curves are active in the model.			DIRECT

No.	Keyword Name	INIT Data – Static Data Header Keywords Table F.14			Status Or Value
		Keyword	No. of Entries	Data Type	
2-4	Data	SATOPTS keyword in the RUNSPEC section - reversible relative permeability (black-oil) are active in the model.			IRREVERS
		Radial grid geometry has been used for a compositional model indicator, set to T or F. Note that currently OPM Flow does not support radial grids and therefore this item should be set to F.			RADIAL
2.5	Data	Radial grid geometry has been used for a black-oil model indicator, set to T or F. Note that currently OPM Flow does not support radial grids and therefore this item should be set to F.			RADIAL
		Reversible relative permeability (compositional) are active in the model.			IRREVERS
2.6		Undefined.			F
2-7	Data	The hysteresis option has been activated in the model indicator, set to T or F.			HYSTER
2-8 to 2-14	Data	Undefined.			F
2-15	Data	The dual porosity option has been activated in the model indicator, set to T or F. Note that currently OPM Flow does not support dual porosity grids and therefore this item should be set to F.			DUALPORO
2-16		Undefined.			F
2-17	Data	ENDSCALE keyword in the RUNSPEC section - enable end-point scaling indicator.			ENDSCALE
2-18	Data	ENDSCALE keyword in the RUNSPEC section - directional end-point scaling indicator.			DIRECT
2.19	Data	ENDSCALE keyword in the RUNSPEC section - reversible end-point scaling indicator.			IRREVERS
2-20	Data	SCALECRS keyword in the PROPS section - alternative three point end-point scaling indicator.			SCALECRS
2-21 to 2-35		Undefined.			F
2-36	Data	Miscible displacement has been activated in the model indicator, set to T or F. See the *MISCIBLE – Define Miscibility Todd-Longstaff Parameters* keyword in the RUNSPEC section for further information.			
2-37 to 2-55		Undefined.			F

No.	Keyword Name	INIT Data – Static Data Header Keywords Table F.14			Status Or Value
		Keyword	No. of Entries	Data Type	
2-56	Data	The scale water capillary pressure at maximum water saturation (that is minimum pressure) has been activated in the model, set to T or F.			
2-57	Data	The scale water capillary pressure at maximum water saturation (that is minimum pressure) has been activated in the model, set to T or F.			
2-58 to 2-127		Undefined.			0
2-128	Data	Coal Bed Methane option has been activated in the compositional model indicator, set to T or F. Note that currently OPM Flow does not support the Coal Bed Methane option and therefore this item should be set to F.			F
	DOUBHEAD	Global static data keyword for the INIT Data file that defines the double precision REAL variables for this keyword. NDHEAD is the number of entries associated with this keyword, and is dependent on the features being used in the model.			Required
1-1	Format	DOUBHEAD	NDHEAD	DOUB	
2-1 to 2-185		Undefined			0.0

Notes:

1) Rows shaded in gray indicate the start of a keyword.
2) Cells colored orange in the No. column indicate that this item is not supported by OPM Flow and those colored red are either "Not Used" or "Undefined".
3) For formatted output all character variables, including the keywords should be enclosed in single quotes, for example the SEQNUM should be written out as 'SEQNUM '.

Table F.14: INIT Data – Static Data Header Keywords

Example: INIT Data – Static Header Keywords

The following example shows a typical formatted INIT Data header set of keywords.

```
'INTEHEAD'         249 'INTE'
 -955283513      200400           2       -2345       -2345       -2345
      -2345       -2345          10          10           1         100
         10       -2345           7       -2345           0           1
          0           2           2           0           0           0
        110         108         109           3          97          93
      -2345       -2345          19          38          53       -2345
         97          93         146           5           0           1
         15          24           8           5           2           4
          0           0           0           0           0           0
          0           0           0           0           0           0
          0           0           0           0          19          10
       1982           0           0           0           1           0
          0           0           0           0           1          10
          0           0          12           1          25           1
 ……….……..
'LOGIHEAD'          79 'LOGI'
 T  F  F  T  F  F  F  F  T  F  F  F  F  F  F  F  T  F  T  F  F  F  F  F  F
 F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F
 F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F
 F  F  F  F
'DOUBHEAD'         185 'DOUB'
  0.00000000000000D+00   0.10000000000000D+01   0.36500000000000D+03
  0.10000000149012D+00   0.15000000596046D+00   0.30000000000000D+01
  0.30000001192093D+00   0.10000000149012D+00   0.10000000149012D+00
 -0.10000000200409D+21   0.10000000000000D+01   0.10000000000000D+01
  0.10000000000000D+01   0.00000000000000D+00   0.00000000000000D+00
  0.10000000000000D+01   0.10000000000000D-03
 ……….……..
```

End of Example

F.6.4 INIT DATA – STATIC GRID ARRAY DATA KEYWORDS

This series of keywords define various static global and LGR grid property arrays for each grid cell in the model. The grid array keywords use the same grid property keywords described in the GRID section of the manual. For example, the DEPTH, PORO, PERMX, PERMY, PERMZ, NTG, TOPS arrays etc. In addition, the keywords also include the calculated pore volumes (PORV) and transmissibility arrays (TRANX, TRANY and TRANZ) after processing the GRID and EDIT sections. Note the first property should always be the PORV array for both global and LGR grid types.

This format is also used to define the global grid REGION, NNC and ENPOINTS section arrays, with the keyword name corresponding to the array name. For example, for the REGION keywords the same keywords outlined in the REGION section of the manual; that is EQLNUM, FIPNUM, PVTNUM, ROCKNUM, SATNUM are utilized as the keywords in this section.

The structure for this type of data and the keywords are defined in Table F.15.

No.	Keyword Name	INIT Data – Static Grid Array Data Keywords Table F.15			Status Or Value
		Keyword	No. of Entries	Data Type	
	PROPERTY	The PROPERTY keyword defines the start of a grid property array section of the INIT file. PROPERTY is the keyword name of the array being written out, as outlined in the GRID section. For example, DEPTH, PORO, PERMX, PERMY, PERMZ, NTG, TOPS keywords and the calculated PORV, TRANX, TRANY and TRANZ arrays. The PROPERTY keyword (items 1-1 and 2-1) is repeated for each grid property. The PORV property must be the first data set for this keyword type and is written out in full for each cell in the model (that is NXYZ = NX x NY x NZ values). Note the PORV array values for inactive cells are set to a constant value of zero. Subsequent grid property arrays can be in any order and only the active cells are written out (NACTIV).			Required
1-1	Format	PROPERTY	NXYZ or NACTIV	REAL	
2-1	Data	PROPERTY data set.			
LGR Property Data for Each LGR Keyword and Data					
Table and Constant Keywords (TABDIMS, TAB and CON)					
	REGION	The REGION keyword defines the start of grid region array section. REGION is the keyword name of the region array being written out, as outlined in the REGIONS section. For example, EQLNUM, FIPNUM, PVTNUM, ROCKNUM, SATNUM, etc. For analytical aquifers in the model, the AQUIFERA variable name is used to indicate which cells are connected to a given aquifer. If a cell is connected to an aquifer number N, then the cell value will be set to 2(N-1) in the array. The REGION keyword (items 1-1 and 2-1) is repeated for each global region array and the arrays can be written out in any order.			Required
1-1	Format	REGION	NACTIV	INTE	
2-1	Data	REGION data set.			

No.	Keyword Name	INIT Data – Static Grid Array Data Keywords Table F.15			Status Or Value
		Keyword	No. of Entries	Data Type	
LGR Region Keywords and Data for Each Grid					
	ENDPOINT	The ENDPOINT keywords define the start of a grids saturation and end-point data arrays. ENDPOINT is the keyword name of the saturation (SWL, SWATINIT, etc.) array or end-point array (PWC, KROW, etc.) being written out, as described in the GRID and PROPS sections. The type of data for the keyword should be set to REAL for arrays with real values (SWL, SWATINIT, etc) or INTE for integer arrays (ENDNUM etc.). ENDPOINT arrays can be in any order and only the active cells are written out (NACTIV). This keyword (items 1-1 and 2-1) is repeated for each saturation and end-point array being written out. The data written out is dependent on the various options declared in the RUNSPEC section as well as the data declared in the GRID and PROPS sections.			Required
1-1	Keyword	ENDPOINTS	NACTIV	INTE or REAL	
2-1	Data	ENDPOINT data set.			
LGR Saturation End-Point Keywords and Data for Each LGR					
	NNC	The NNC keyword defines the start of a grids non-neighbor connection array data. NNC is the keyword name of the non-neighbor connection array being written out, and should be set to one of the following keywords: 1) TRANNNC: the non-neighbor connection transmissibilities within the grid. Always required even if there are no NNC, in which case NUMNCC is set to zero. 2) DIFFNNC: the non-neighbor connection diffusivities within the grid. Only required if the Diffusivity option has been activated by the DIFFUSE keyword in the RUNSPEC section. This option is currently not supported by OPM Flow 3) HEATNNC: the non-neighbor connection thermal transmissibilities within the grid. Only required if OPM Flow's THERMAL option has been activated by the THERMAL keyword in the RUNSPEC section. The following two additional NNC keywords are required for when the data for an LGR is being defined: 1) TRANCC: the non-neighbor connection transmissibilities within the LGR (always required for an LGR). 2) TRANGL: the non-neighbor connections transmissibilities between the LGR and the global grid (always required for an LGR). Number of data values for this keyword data set should set to the number of non-neighbor connection to be read or written out (NUMNNC) for the TRANNC, DIFFNNC, HEATNCC, TRANCC, and TRANGL arrays. The NNC keyword (items 1-1 and 2-1) are repeated for each non-neighbor connection array being written out.			Required
1-1	Format	NNC	NUMNCC	REAL	
2-1	Data	NNC data set.			
LGR Non-Neighbor Connection Keywords for Each LGR					

No.	Keyword Name	INIT Data – Static Grid Array Data Keywords Table F.15			Status Or Value
		Keyword	No. of Entries	Data Type	

Notes:

1) Rows shaded in gray indicate the start of a keyword and rows colored orange indicate the location of LGR data if LGRs are present in the model.

2) For formatted output all character variables, including the keyword should be enclosed in single quotes, for example the PORV and PROPERTY fields should be written out as 'PORV ' and 'DEPTH ', etc.

Table F.15: INIT Data – Static Grid Array Data Keywords

Example: INIT Data – Static Grid Array Data Keywords

The following example shows a typical formatted global grid property data keyword.

```
 'PORV    '          100 'REAL'
   0.10686456E+07   0.10686456E+07   0.10686456E+07   0.10686456E+07
   0.10686456E+07   0.10686456E+07   0.10686456E+07   0.10686456E+07
  ..................................
   0.10686456E+07   0.10686456E+07   0.10686456E+07   0.10686456E+07
   0.10686456E+07   0.10686456E+07   0.10686456E+07   0.10686456E+07
 'DX      '          100 'REAL'
   0.10000000E+04   0.10000000E+04   0.10000000E+04   0.10000000E+04
   0.10000000E+04   0.10000000E+04   0.10000000E+04   0.10000000E+04
  ..................................
   0.10000000E+04   0.10000000E+04   0.10000000E+04   0.10000000E+04
   0.10000000E+04   0.10000000E+04   0.10000000E+04   0.10000000E+04
 'DY      '          100 'REAL'
   0.10000000E+04   0.10000000E+04   0.10000000E+04   0.10000000E+04
   0.10000000E+04   0.10000000E+04   0.10000000E+04   0.10000000E+04
  ..................................
   0.10000000E+04   0.10000000E+04   0.10000000E+04   0.10000000E+04
   0.10000000E+04   0.10000000E+04   0.10000000E+04   0.10000000E+04
 'DZ      '          100 'REAL'
   0.20000000E+02   0.20000000E+02   0.20000000E+02   0.20000000E+02
   0.20000000E+02   0.20000000E+02   0.20000000E+02   0.20000000E+02
  ..................................
   0.20000000E+02   0.20000000E+02   0.20000000E+02   0.20000000E+02
   0.20000000E+02   0.20000000E+02   0.20000000E+02   0.20000000E+02
 'PERMX   '          100 'REAL'
   0.50000000E+03   0.50000000E+03   0.50000000E+03   0.50000000E+03
  ..................................
   0.50000000E+03   0.50000000E+03   0.50000000E+03   0.50000000E+03
   0.50000000E+03   0.50000000E+03   0.50000000E+03   0.50000000E+03
..................................
TABULAR DATA
..................................
 'PVTNUM  '          100 'INTE'
           1           1           1           1           1           1
           1           1           1           1           1           1
..................................
           1           1           1           1           1           1
 'SATNUM  '          100 'INTE'
           1           1           1           1           1           2
           1           1           1           2           2           2
..................................
 'EQLNUM  '          100 'INTE'
           1           1           1           1           1           1
           1           1           1           1           1           1
 ..................................
```

End of Example

F.6.5 INIT DATA - LGR STATIC DATA HEADER KEYWORDS

The file structure for this series of keywords consists of two sets of keywords, the first set is repeated each time an LGR data set is written out and the second set, which is the same as global static header keywords in section INIT Data – Static Data Header Keywords, is only written once. The first set of keywords define an LGR's properties, LGR Name for example, and always proceeds an LGR data set for a given LGR; that is this series of keywords are repeated for each LGR data set in the model. This is then followed by the second set of static data header keywords, or the actual array or table keywords. Termination of the LGR data set occurs after the last LGR data set has been written out using the LGRSGONE keyword, which switches input back to the global grid properties. Table F.16 outlines the structure of the INIT Data - LGR Static Header Keywords and their affiliated data.

Note that currently OPM Flow does not support LGR grids and therefore this set of keywords cannot be used by OPM Flow.

No.	Keyword Name	INIT Data - LGR Static Header Keywords Table F.16 (Repeated Each Time An LGR Each Data Set Is Written Out)			Status Or Value
		Keyword	No. of Entries	Data Type	
	LGR	LGR static data header keyword for the INIT Data file that defines the name of the LGR.			Always Required
1-1	Keyword	LGR	1	CHAR	
2-1	Data	Name of the LGR.			
	LGRHEADI	LGR that defines the integer variables for this keyword.			Always Required
1-1	Format	LGRHEADI	45	INTE	
2-1 to 2-45	Data	Undefined.			
	LGRHEADQ	LGR grid keyword for the INIT Data file that defines the logical variables (T for true and F for false) for this keyword.			Always Required
1-1	Format	LGRHEADQ	5	LOGI	
2-1 to 2-5	Data	Undefined			
	LGRHEADD	LGR grid keyword for the INIT Data file that defines the double precision REAL variables for this keyword.			Always Required
1-1	Format	LGRHEADD	5	DOUB	5
2-1 to 2-5	Data	Undefined.			

No.	Keyword Name	INIT Data - LGR Static Header Keywords Table F.16 (Repeated Each Time An LGR Each Data Set Is Written Out)			Status Or Value
		Keyword	No. of Entries	Data Type	
	LGRSGONE	The LGRSGONE keyword marks the end of the LGR section. Number of entries is always set to zero. There is no data set associated with this keyword.			Always Required
1-1	Format	LGRSGONE	0	MESG	
	INTEHEAD	The keywords are the same as the INIT Data Static Data Header keywords but the data pertains to the current LGR. See Table F.14 for a detailed description of this keyword type.			Required Only Once
	LOGIHEAD	See Table F.14 for a detailed description of this keyword type.			Required Only Once
	DOUBHEAD	See Table F.14 for a detailed description of this keyword type.			Required Only Once

Notes:

1) Rows shaded in gray indicate the start of a keyword.
2) Cells colored orange in the No. column indicate that this item is not supported by OPM Flow and those colored red are either "Not Used" or "Undefined".
3) For formatted output all character variables, including the keyword name should be enclosed in single quotes, for example the LGR keyword should be written out as 'LGR '.

Table F.16: INIT Data - LGR Static Data Header Keywords

This series of keywords are repeated for each LGR in the model followed by the pertinent data and in the order outlined in Table F.13 and with the last LGR data set terminated by the LGSGONE keyword to mark the end of the LGR input section.

Note

The LGR, LGRHEADI, LGRHEADQ and LGRHEADD keywords always proceeds and LGR data sets and the LGRSGONE keyword terminates the end of the LGR data section, which causes the data to revert back to the global grid data.

Example: INIT Data - LGR Static Header Keywords

The following example shows a typical formatted INIT Data - LGR Grid Header Keyword data set for a single LGR grid named LGR-1. As this is the first time LGR data is being written out both sets of LGR header keywords are written out followed by the property array data.

```
'LGR     '           1 'CHAR'
'LGR-1   '
'LGRHEADI'          45 'INTE'
          1         100       -2345       -2345       -2345       -2345
      -2345       -2345       -2345       -2345       -2345       -2345
      -2345       -2345       -2345       -2345       -2345       -2345
      -2345       -2345       -2345       -2345       -2345       -2345
      -2345       -2345       -2345       -2345       -2345       -2345
      -2345       -2345       -2345       -2345       -2345       -2345
      -2345       -2345       -2345       -2345       -2345       -2345
      -2345       -2345       -2345
'LGRHEADQ'           5 'LOGI'
 F  F  F  F  F
'LGRHEADD'           5 'DOUB'
  0.00000000000000D+00  -0.10000000200409D+21  -0.10000000200409D+21
 -0.10000000200409D+21  -0.10000000200409D+21
…………………………….……………..

'INTEHEAD'         249 'INTE'
 -955283513      200400           2       -2345       -2345       -2345
      -2345       -2345           2           2           2           8
…………………………….……………..
          1           6           1          10           1          10
          1           1           1           0          30           3
         18          10           9
'LOGIHEAD'          79 'LOGI'
 T  F  F  T  F  F  F  F  T  F  F  F  F  F  F  F  T  F  T  F  F  F  F  F  F
 F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F
 F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F
 F  F  F  F
'DOUBHEAD'         185 'DOUB'
  0.00000000000000D+00   0.10000000000000D+01   0.36500000000000D+03
  0.10000000149012D+00   0.15000000596046D+00   0.30000000000000D+01
…………………………….……………..
  0.10000000000000D+01   0.00000000000000D+00   0.00000000000000D+00
  0.10000000000000D+01   0.10000000000000D-03
…………………………….……………..
'PORV    '           8 'REAL'
  0.13358070E+06   0.13358070E+06   0.13358070E+06   0.13358070E+06
  0.13358070E+06   0.13358070E+06   0.13358070E+06   0.13358070E+06
'DX      '           8 'REAL'
  0.50000000E+03   0.50000000E+03   0.50000000E+03   0.50000000E+03
  0.50000000E+03   0.50000000E+03   0.50000000E+03   0.50000000E+03
'DY      '           8 'REAL'
  0.50000000E+03   0.50000000E+03   0.50000000E+03   0.50000000E+03
  0.50000000E+03   0.50000000E+03   0.50000000E+03   0.50000000E+03
'DZ      '           8 'REAL'
  0.10000000E+02   0.10000000E+02   0.10000000E+02   0.10000000E+02
  0.10000000E+02   0.10000000E+02   0.10000000E+02   0.10000000E+02
'PERMX   '           8 'REAL'
  0.50000000E+03   0.50000000E+03   0.50000000E+03   0.50000000E+03
  0.50000000E+03   0.50000000E+03   0.50000000E+03   0.50000000E+03
…………………………….……………..
```

End of Example

F.6.6 INIT DATA - LGR GRID ARRAY DATA KEYWORDS

The file structure for this set of keywords is identical to the Static Grid Array Data Keywords described in Table F.15, except that LGR header keywords prefix the actual data keywords. Both the LGR header and data keywords are repeated for each LGR in the model, and the last LGR data set is terminated by the LGSGONE keyword to mark the end of the LGR input section. The keyword description for this type of data set is outlined in Table F.17.

Note that currently OPM Flow does not support LGR grids and therefore this series of keywords cannot be used by OPM Flow.

No.	Keyword Name	INIT Data - LGR Grid Array Data Keywords Table F.17 (Repeated For Each LGR In The Model)	Status Or Value
1-1	LGR	LGR Header Keywords (see Table F.16).	Required
	LGRHEADI		
	LGRHEADQ		
	LGRHEADD		
1-2	INTEHEAD	LGR Header Keywords (see Table F.16).	Required Only Once
	LOGIHEAD		
	DOUBHEAD		
1-3	PROPERTY REGION NNC or ENPOINIT	LGR array data set for the current LGR – same format as the INIT Data - Grid Array Data Keywords (see Table F.15).	Required
1-4	LGRSGONE	LGR Header Section termination keyword (see Table F.16).	Required

Notes:

1) Rows shaded in gray indicate the start of a keyword.
2) Cells colored orange in the No. column indicate that this item is not supported by OPM Flow.
3) For formatted output all character variables, including the keyword should be enclosed in single quotes, for example the LGR keyword should be written out as 'LGR '.

Table F.17: INIT Data - LGR Grid Array Data Keywords

This keyword data set is repeated for each LGR in the model with the last LGR data set terminated by the LGSGONE keyword to mark the end of the LGR data section.

Example: INIT Data - LGR Static Grid Array Data Keywords

The following example shows a typical formatted INIT Data - LGR Static Grid Array Data Keywords for the region data set with two LGR grids named LGR-1 and LGR-2.

```
'LGR     '           1 'CHAR'
'LGR-1   '
'LGRHEADI'          45 'INTE'
         1         100       -2345       -2345       -2345       -2345
     -2345       -2345       -2345       -2345       -2345       -2345
     -2345       -2345       -2345       -2345       -2345       -2345
     -2345       -2345       -2345       -2345       -2345       -2345
     -2345       -2345       -2345       -2345       -2345       -2345
     -2345       -2345       -2345       -2345       -2345       -2345
     -2345       -2345       -2345       -2345       -2345       -2345
     -2345       -2345       -2345
'LGRHEADQ'           5 'LOGI'
 F  F  F  F  F
'LGRHEADD'           5 'DOUB'
  0.00000000000000D+00  -0.10000000200409D+21  -0.10000000200409D+21
 -0.10000000200409D+21  -0.10000000200409D+21
'PVTNUM  '           8 'INTE'
         1           1           1           1           1           1
         1           1
'SATNUM  '           8 'INTE'
         1           1           1           1           2           2
         2           2
'EQLNUM  '           8 'INTE'
         1           1           1           1           1           1
         1           1
'FIPNUM  '           8 'INTE'
         1           1           1           1           1           1
         1           1
'LGR     '           1 'CHAR'
'LGR-2   '
'LGRHEADI'          45 'INTE'
         2         100       -2345       -2345       -2345       -2345
     -2345       -2345       -2345       -2345       -2345       -2345
     -2345       -2345       -2345       -2345       -2345       -2345
     -2345       -2345       -2345       -2345       -2345       -2345
     -2345       -2345       -2345       -2345       -2345       -2345
     -2345       -2345       -2345       -2345       -2345       -2345
     -2345       -2345       -2345
'LGRHEADQ'           5 'LOGI'
 F  F  F  F  F
'LGRHEADD'           5 'DOUB'
  0.00000000000000D+00  -0.10000000200409D+21  -0.10000000200409D+21
 -0.10000000200409D+21  -0.10000000200409D+21
'PVTNUM  '           8 'INTE'
         1           1           1           1           1           1
         1           1
'SATNUM  '           8 'INTE'
         1           2           1           2           2           1
         2           1
'EQLNUM  '           8 'INTE'
         1           1           1           1           1           1
         1           1
'FIPNUM  '           8 'INTE'
         1           1           1           1           1           1
         1           1
'LGRSGONE'           0 'MESS'
```

End of Example

F.6.7 INIT DATA – STATIC TABULAR DATA HEADER KEYWORD

The INIT Data – Static Tabular Data Header Keyword precedes the TAB keyword and consists of just one keyword followed by a series of integer values that define the dimensions of the data in the TAB keyword. TABDIMS is then followed by the TAB keyword that contains the actual tabular data in one continuous array.

The structure for this keyword is defined in Table F.18.

No.	Keyword Name	INIT Data – Static Tabular Data Header Keyword Table F.18			Status Or Value
		Keyword	No. of Entries	Data Type	
	TABDIMS	Tabular header keyword for the INIT Data file that defines the start of the tabular data and contains the base addresses and dimension of subsequent tabular array keyword TAB. Missing values should be defaulted to one.			Required
1-1	Format	TABDIMS	100	INTE	
2-1	Data	Total size of TAB data array, value should be the same as that on the TAB keyword.			NTABDA
2-2	Data	Location of the first entry in the TAB array for the rock property table.			IBROCK
2-3	Data	Maximum umber of rock property tables.			NTROCK
2-4	Data	Location of the first entry in the TAB array of rock compaction data.			IBROCC
2-5	Data	Maximum number of pressure nodes in rock compaction data table.			NPROCC
2-6	Data	Maximum number of rock compaction data tables.			NTROCC
2-7	Data	Location of the first entry in the TAB array for the oil PVT property tables.			IBPVTO
2-8	Data	Location of the first entry in the TAB array for the oil pressure nodes.			JBPVTO
2-9	Data	Maximum number of composition nodes in oil PVT tables.			NRPVTO
2-10	Data	Maximum number of pressure nodes in oil PVT tables.			NPPVTO
2-11	Data	Maximum number of oil PVT property tables.			NTPVTO
2-12	Data	Location of the first entry in the TAB array for water PVT property tables.			IBPVTW
2-13	Data	Maximum number of water property tables.			NTPVTW
2-14	Data	Location of the first entry in the TAB array for gas PVT property tables.			IBPVTG
2-15	Data	Location of the first entry in the TAB array for the gas pressure nodes.			JBPVTG
2-16	Data	Maximum number of composition nodes in the gas PVT tables.			NRPVTG
2-17	Data	Maximum number of pressure nodes in the gas PVT tables.			NPPVTG
2-18	Data	Maximum number of gas PVT property tables.			NTPVTG
2-19	Data	Location of the first entry in the TAB array for surface density tables.			IBDENS
2-20	Data	Maximum number of surface densities tables.			NTDENS
2-21	Data	Location of the first entry in the TAB array for water saturation tables.			IBSWFN

No.	Keyword Name	INIT Data – Static Tabular Data Header Keyword Table F.18			Status Or Value
		Keyword	No. of Entries	Data Type	
2-22	Data	Maximum number of saturation nodes in water saturation tables.			NSSWFN
2-23	Data	Maximum number of water saturation tables.			NTSWFN
2-24	Data	Location of the first entry in the TAB array for gas saturation tables .			IBSGFN
2-25	Data	Maximum number of saturation nodes in gas saturation tables.			NSSGFN
2-26	Data	Maximum number of gas saturation tables.			NTSGFN
2-27	Data	Location of the first entry in the TAB array for oil saturation tables.			IBSOFN
2-28	Data	Location of the first entry in the TAB array for array of connate water saturations.			IBSWCO
2-29	Data	Maximum number of saturation nodes in the oil saturation tables.			NSSOFN
2-30	Data	Maximum number of oil saturation tables			NTSOFN
2-31 to 40		Undefined			1
2-41	Data	Location of the first entry in the TAB array to the VE table data.			IBVETB
2-42	Data	Maximum number of items in a VE table.			NSVETB
2-43	Data	Maximum number of VE tables.			NTVETB
2-44	Data	Location of the first entry in the TAB array to the threshold pressure array.			IBTHPR
2.45	Data	Location of the first entry in the TAB array to SCALELIM data			IBSLIM
2.46	Data	Maximum number of end-point versus depth nodes.			NSENDP
2.47	Data	Maximum number of end-point versus depth tables.			NTENDP
2.48	Data	Location of the first entry in the TAB array to the reference temperature RTEMP used in PVZG keyword in the PROPD section.			IBRTEM
2.49	Data	Location of the first entry in the TAB array to the value specified by the TOLCRIT keyword. Note that this is a single value applied to all the saturation tables.			IBCTOL
2.50		Undefined			1
2-51	Data	Location of the first entry in the TAB array to LANGMUIR table.			IBLANG
2-52	Data	Maximum number of columns in LANGMUIR tables.			NCLANG
2-53	Data	Maximum number of rows in the LANGMUIR tables			NSLANG
2-54	Data	Maximum number of LANGMUIR tables			NTLANG
2-55	Data	Location of the first entry in the TAB array to a LANGSOLV table.			IBLNG2

No.	Keyword Name	INIT Data – Static Tabular Data Header Keyword Table F.18			Status Or Value
		Keyword	No. of Entries	Data Type	
2-56	Data	Location of the first entry in the TAB array to a COALPP table.			IBCADP
2-57	Data	Location of the first entry in the TAB array to a COALADS table.			IBCADS
2-58	Data	Location of the first entry in the TAB array to a ROCKPAMA table.			IBROCP
2-59	Data	Maximum number of tables of ROCKPAMA tables			NTRPMA
2-60 to 2-100		Undefined			1

Notes:

1) Rows shaded in gray indicate the start of a keyword.
2) Cells colored orange in the No. column indicate that this item is not supported by OPM Flow and those colored red are either "Not Used" or "Undefined".
3) For formatted output all character variables, including the keywords should be enclosed in single quotes, for example the PORV and PROPERTY fields should be written out as 'PORV ' and 'DEPTH ', etc.

Table F.18: INIT Data – Static Tabular Data Header Keyword

Example: INIT Data – Static Tabular Data Header Keyword

The following example shows a typical formatted INIT Data – Static Tabular Data Header Keyword and the affiliated data.

```
'TABDIMS '         100 'INTE'
     77006           1           2           1          60           1
        17       36017          60          60           2       36799
         2       37139       73139          60          60           2
         5           2       36809          33           2       73259
        33           2       36137       36797          66           2
     73589           5       73644          20           5       74044
        20           5           0           0       74444          25
         2       74494       75835          20           1       77003
         0           0           0           0           0           0
         0           0           0           0           0           0
         0           0           0           0           0           0
         0           0           0           0           0           0
         0           0           0           0           0           0
         0           0           0           0           0           0
         0           0           0           0           0           0
         0           0           0           0           0           0
         0           0           0           0
```

End of Example

F.6.8 INIT DATA – STATIC TABULAR AND CONSTANT DATA ARRAY KEYWORDS

The INIT Data – Static Tabular and Constant Data Array Keyword section contains the actual tabular data in one continuous array, followed by the constant data array. The tabular array data is parse according to the phases present in the model and the location and dimension data given by the TABDIMS keyword.

The structure for this set of keywords is defined in Table F.19.

No.	Keyword Or Variable Name	INIT Data – Static Tabular And Constant Data Array Keyword Table F.19			Status Or Value
		Keyword	No. of Entries	Data Type	
	TAB	Tabular data array keyword for the INIT Data file that defines the start of the tabular data array. The total number of entries is given by the NTABDA and the value should be the same as declared on the TABDIMS keyword.			Required
1-1	Format	TABDIMS	NTABDA	DOUB	
2-1	Data	Tabular data array. The data should be written out in double precision and missing data should be set to less than or equal to -10.0 x 1020 or greater than or equal to 10.0 x 1020.			
	CON	Constant data array that contains the constant dead oil, dead vaporized oil, and saturation pressure PVT entered via the RSCONST/RSCONSTT and RVCONT/RVCONSTT keywords in the PROPS section.			Required
1-1	Format	CON	NPVT x 3	DOUB	
2-1	Data	The data consists of three values for each PVT data set (NPVT), RS, RV and the saturation pressure of the active phase in the model (bubble-point or dew point).			

Notes:

1) Rows shaded in gray indicate the start of a keyword.
2) Note both TAB and CON keywords should be enclosed in single quotation marks, that is 'TAB ' and 'CON '.

Table F.19: INIT Data – Static Tabular and Constant Data Array Keywords

The phases present in the model are given on the INTEHEAD keyword (Table F.14) in location 2-15 where the type of phases in the model are set via an integer: 1 for oil, 2 for water, 3 for oil-water, 4 for gas, 5 for oil-gas, 6 for gas-water and 7 for oil-water-gas. This data is used to access the phase dependent data (PVT, saturation functions tables etc.) combine with the location (or base address) and the dimension data given on the TABDIMS keyword in Table F.18.

Note that data given in Table F.18 refers to the maximum values and therefore missing data should be set to the default value.

Example: INIT Data – Static Tabular and Constant Data Array Keywords

The following example shows a typical formatted data set for this type set of keywords and the affiliated data. Here the CON array has two PVT tables but the values are all set to zero indicating that dead oil and vaporized dead oil are not active phases in the model.

```
'TAB     '       73576 'DOUB'
   0.497000000000000D-05   0.248000000000000D-05   0.000000000000000D+00
  -0.200000000000000D+21  -0.200000000000000D+21  -0.200000000000000D+21
  -0.200000000000000D+21  -0.200000000000000D+21  -0.200000000000000D+21
 ..............................................
  -0.200000000000000D+21  -0.200000000000000D+21  -0.200000000000000D+21
   0.521000000000000D-05   0.261000000000000D-05   0.000000000000000D+00
  -0.200000000000000D+21  -0.200000000000000D+21  -0.200000000000000D+21
'CON     '           6 'DOUB'
   0.000000000000000D+00   0.000000000000000D+00   0.000000000000000D+00
   0.000000000000000D+00   0.000000000000000D+00   0.000000000000000D+00
 ..............................................
```

End of Example

F.7 RESTART FILES – DYNAMIC DATA (SOLUTION, GROUPS, WELLS, CONNECTIONS, AQUIFERS ETC.)

RESTART files contain the solution data arrays for each active cell in the model, and represents a "snapshot" of the reservoir state at a reporting time step, for example the pressure (PRESSURE, PPCW, and PPCG), fluid saturations (SGAS, SOIL and SWAT), and fluid state (RS and RV) for all active cells. The PPCW and PPCG arrays are the water and capillary solution arrays and the RS and RV arrays are the gas-oil and vaporized-oil ratio (condensate-gas ratio) solution arrays. The data written to the RESTART file is dependent on the fluid phases and options invoked in the model, as well as any additional user requested solution data. The default arrays written out are always sufficient to enable the simulator to restart from a previous run[405]. This was the original purpose of the RESTART file, that is saving the reservoir state in such a manner, is to enable a "restart" of the simulation.

RESTART files also contain contain the necessary information to do restart runs for Groups, Wells, Connections, Aquifer data etc. This information enables restart runs based on Schedule information read from the Restart file only, using the SKIPREST keyword in the SCHEDULE section.

As of the OPM Flow 2019-04 release the simulator can "restart" from the commercial simulator's generated RESTART files as well as the commercial simulator being to "restart" from the OPM Flow's generated RESTART files. See section *2.2 Running OPM Flow 2023-10 From The Command Line* for additional information.

As RESTART files contain a complete description of the reservoir state, they are also used in post processing software to visualize the reservoir solutions through time, for example by OPM ResInsight. And as the functionality of the both OPM Flow and the commercial simulator have expanded over time additional solution arrays have been added to the RESTART file to enable "restarts" (POLYMER – polymer saturations), as well as to write out additional user specific solution arrays (FIPOIL - fluid in-place oil array), that are not necessary for a "restart", but are consider useful in understanding the reservoir performance.

The frequency and type of data written to the RESTART file is controlled via the RPTSCHED and RPTRST keywords in the SCHEDULE, with the latter having greater functionality and flexibility. For example, to request that the standard restart data be written out every month using the RPTRST keyword:

```
--
--       RESTART CONTROL BASIC = 4 (YEARLY) 5 (MONTHLY)
--
RPTRST
         BASIC=5                                                    /
```

The next example requests that the standard restart data be written at every report time step until this switch is reset and all the restarts are kept. In addition to the standard the data the gas, oil and water relative permeability data will also be written out at each report time step.

```
--
--       RESTART CONTROL BASIC = 4 (YEARLY) 5 (MONTHLY)
--
RPTRST
         BASIC=2  KRG   KRO   KRW                                   /
```

RESTART solution data is written out via two separate files; the RESTART Index file that specifies and defines the format and data type written to the RESTART Data file, and the RESTART Data file that contains the solution data for each active cell in the model for the requested report time steps. The commercial simulator writes out both files, where as OPM Flow only writes out the RESTART Data file, as the data can be read by most post-processing software, including OPM ResInsight, using only the RESTART Data file as input. Secondly, both the commercial simulator and OPM Flow can be "restarted" using just the RESTART Data file.

The following sections outline the format for these two file types.

[405] *For example in a three phase (oil, gas and water) runs only two phase saturations are needed for a "restart" run, as the third phase can be calculated in by summing two phases and subtracting from one.*

F.7.1 RESTART INDEX FILES

This file type is not supported or required by OPM Flow or OPM ResInsight to read or write restart files.

F.7.2 RESTART DATA FILE SPECIFICATION

This file type contains the global and LGR grid solution data property data[406] (PRESSURE, SGAS, SOIL and SWAT) and the group, well and connection data[407]. The overall structure of this file is similar to the INIT file, and the individual data keywords are structured as presented in Table F.20.

Reference Section	Global Section Keywords	Global Data Keywords	LGR Data Keywords	Global Section Keywords	Notes
F.7.3	SEQNUM				Global grid header keyword that defines the start of a RESTART global grid time step data set.
F.7.3	INTEHEAD LOGIHEAD DOUBHEAD				Global grid header keyword that defines the integer, logical and double precision variables for this header keyword.
F.7.4	IGRP SGRP etc.				Group, well, and connection data status keywords for this reporting time step.
F.7.6	IAAQ SAAQ ect.				Aquifer definition arrays.
F.7.7	HIDDEN				Defines HIDDEN solution data, that is a list of solution arrays NOT to be read by post processing software.
F.7.8	STARTSOL				This keyword marks the start of the solution variable section for the global grid.
F.7.8		PRESSURE SWAT etc.			Solution array section (PRESSURE, SGAS, SOIL, SWAT, etc.) for the Global grid.
F.7.8	LGRNAMES				A list of LGRs in this report step
F.7.8	ENDSOL				Marks the end of the solution variable section for the global grid.
F.5.1			LGR LGRHEADI LGRHEADQ LGRHEADD		LGR grid header keyword for that defines the start of a RESTART LGR data set.
F.7.3			INTEHEAD LOGIHEAD DOUBHEAD		LGR header keyword and defines the integer, logical and double precision variables for this header keyword.

[406] *Several keywords are used for both global and LGR grids, for example the INTEHEAD, LOGIHEAD, DOUBHEAD keywords and SOLUTION arrays.*

[407] *The group, well, and connection production data is not stored on the RESTART file, but is instead stored on the SUMMARY file. The RESTART file contains the group, well and connection configurations as the they change versus time. For example, the status of a well (OPEN or SHUT), or the connections open to flow in a well.*

Reference Section	Global Section Keywords	Global Data Keywords	LGR Data Keywords	Global Section Keywords	Notes
F.7.4			IGRP SGRP etc.		Group, well, and connection data status keywords for this reporting time step.
F.7.7			HIDDEN		Defines HIDDEN solution data section.
F.7.8			STARTSOL		Start of LGR solution array section.
F.7.8			Solution Keywords	PRESSURE SWAT etc.	LGR solution array section – same as global grid
F.7.8			ENDSOL		End of the solution variable section for the current LGR grid.
F.5.1			ENDLGR		End of current LGR section.
F.5.1			LGR LGRHEADI LGRHEADQ LGRHEADD		LGR grid header keywords
F.7.4			IGRP SGRP etc.		Group, well, and connection data status keywords for this reporting time step.
F.7.7			HIDDEN		Defines HIDDEN solution data section.
F.7.8			STARTSOL		Start of solution array section.
F.7.8			Solution Keywords	PRESSUE SWAT etc.	LGR solution array section.
F.7.8			ENDSOL		End of the solution variable section.
F.5.1			ENDLGR		End of current LGR section.
F.7.3	SEQNUM				Global grid header keyword that defines the start of a RESTART global grid time step data set.

Notes:

1) Cells shaded in gray indicate the combination is not applicable and cells colored orange indicate that this item is not currently supported by OPM Flow.

Table F.20: RESTART Data File Structure

The individual keywords are described in the following sections.

F.7.3 RESTART DATA - HEADER KEYWORDS

This set of keywords[408] are the first set of keywords that should be read or written to the RESTART file. This data set includes INTEHEAD, LOGIHEAD and DOUBHEAD keywords that define versus parameters used in subsequent keywords in the RESTART file for the global and LGR grids.

The structure for this type of keyword is defined in Table F.21.

No.	Keyword Name	RESTART Data - Header Keywords Table F.21			Status Or Value
		Keyword	No. of Entries	Data Type	
	SEQNUM	Global grid header keyword for the RESTART Data file that defines the start of a RESTART global grid time step data set for unified restart files only.			Optional
1-1	Format	SEQNUM	1	INTE	
2-1	Data	Report time step number.			0
	INTEHEAD	Global grid header keyword for the RESTART Data file that defines the integer variables for this time step. NIHEAD is the number of entries associated with this keyword, and is dependent on the features being used in the model. Undefined values should be set to zero.			Required
1-1	Format	INTEHEAD	NIHEAD	INTE	
2-1	Data	An encoded integer corresponding to the time the file was created. For files not originating from the commercial simulator, this value may be set to zero.			ISNUM
2-2	Data	Simulator version, should be set to zero.			VERSION
2-3	Data	Unit system used in the simulation, set to 1 for metric, 2 for field, and 3 for laboratory			
2-4 to 2-8	Data	Undefined.			0
2-9	Data	DIMENS key word in the RUNSPEC section - the number of grid blocks in the x-direction.			NX
2-10	Data	DIMENS key word in the RUNSPEC section - the number of grid blocks in the y-direction.			NY
2-11	Data	DIMENS key word in the RUNSPEC section - the number of grid blocks in the z-direction.			NZ
2-12	Data	Number of global active cells in the model .			NACTIV
2-13	Data	Not used.			0
2-14	Data	Not used.			0
2-15	Data	Type of phases in the model, set to 1 for oil, 2 for water, 3 for oil-water, 4 for gas, 5 for oil-gas, 6 for gas-water and 7 for oil-water-gas.			PHASE
2-16	Data	Undefined.			0
2-17	Data	Number of wells in the model.			NWELLS

[408] *The RESTART Data – Header keywords are used both for global and LGR grids with the global grid data for the global grid and the individual LGR data for the LGR grids. The LGR data is preceded by a series of LGR header keywords and terminated by an LGR termination keyword.*

No.	Keyword Name	RESTART Data - Header Keywords Table F.21			Status Or Value
		Keyword	No. of Entries	Data Type	
2-18	Data	Maximum number of grid block connections per well.			NCWMAX
2-19	Data	Actual number of groups.			NGRP
2-20	Data	Maximum number of wells belonging to a group in the model.			NWGMAX
2-21	Data	Maximum number of groups for this model.			NGMAXZ
2-22 to 2-24	Data	Not used.			
2-25	Data	Number of values per well in the IWEL array (default value is 97).			NIWELZ
2-26	Data	Number of values per well in the SWEL array.			NSWELZ
2.27	Data	Number of values per well in the XWEL array.			NXWELZ
2-28	Data	Number of eight character words per well in the ZWEL array.			NZWELZ
2-29 to 2-32	Data	Not used.			
2-33	Data	Number of values per completion in ICON array (default 19).			NICONZ
2-34	Data	Number of values per completion in SCON array.			NSCONZ
2-35	Data	Number of values per completion in XCON array.			NXCONZ
2-36		Not used.			0
2-37	Data	Number of values per group in IGRP array.			NIGRPZ
2-38	Data	Number of values per group in SGRP array.			NISGRPZ
2-39	Data	Number of values per group in XGRP array.			NIXGRPZ
2-40	Data	Number of values per group in ZGRP array.			NIZGRPZ
2-41		Not used.			
2-42	Data	Maximum number of analytical aquifer connections.			NCAMAX
2-43	Data	Number of values per aquifer in the IAAQ array.			NIAAQZ
2-44	Data	Number of values per aquifer in the SAAQ array.			NSAAQZ
2-45	Data	Number of values per aquifer in the XAAQ array.			NXAAQZ
2-46	Data	Number of values per aquifer connection in the ICAQ array.			NICAQZ
2-47	Data	Number of values per aquifer connection in the SCAQ array.			NSCAQZ
2-48	Data	Number of values per aquifer connection in the ACAQ array.			NXCAQZ
2-49 to 2-51		Not used.			
2-52	Data	Index indicating if group control is used or not (1 – for GCONPROD group control, 2 for GCONINJE control, or 0 if no group control)			NGCTRL

No.	Keyword Name	RESTART Data - Header Keywords Table F.21			Status Or Value
		Keyword	No. of Entries	Data Type	
2-53 to 2.58		Undefined.			
2-59	Data	Parameter to determine the nominated phase for the guide rate.			NGRNPH
2-60 to 2-64		Undefined.			
2-65	Data	Calendar day of report step, the value should be greater than or equal to one and less than or equal to 31 for the calendar days.			DAY
2-66	Data	Calendar month of report step, the value should be greater than or equal to one and less than or equal to 12 for the calendar month.			MONTH
2-67	Data	Calendar year of report step, a positive four digit integer value of the start year, which must be specified fully by four digits, that is 2019.			YEAR
2-68	Data	Numerical solver step, that is the number of solver steps the simulator has performed so far.			NUM_SOLVER _STEP
2-69	Data	Report step.			REPORT_STEP
2-70 to 2-71		Undefined.			0
2-72	Data	Index for WHISTCTL keyword.			WHISTC
2-73 to 2-80		Undefined			
2-81	Data	TUNING keyword in the SCHEDULE section - the maximum number of Newton iterations for a time step.			NEWTMX
2-82	Data	TUNING keyword in the SCHEDULE section - the minimum number of Newton iterations for a time step			NEWTMN
2-83	Data	TUNING keyword in the SCHEDULE section - the maximum number of linear iterations within a Newton iteration.			LITMAX
2-84	Data	TUNING keyword in the SCHEDULE section - the minimum number of linear iterations within a Newton iteration.			LITMIN
2-85		Undefined.			
2-86		Undefined.			
2-87	Data	TUNING keyword in the SCHEDULE section - the maximum number of iterations within a well flow calculation.			MXWSIT
2-88	Data	TUNING keyword in the SCHEDULE section - he maximum number of iterations for solving the bottom-hole pressure for wells under tubing head pressure control within a well flow calculation.			MXWPIT
2-89		Undefined.			

No.	Keyword Name	RESTART Data - Header Keywords Table F.21			Status Or Value
		Keyword	No. of Entries	Data Type	
2-90	Data	REGDIMS and TABDIMS keywords in the RUNSPEC section. The maximum number of regions in the FIPNUM region array. Note that this parameter may also be set on the TABDIMS keyword as well. If NTFIP is set in both places then the maximum value should be used.			NTFIP
2-91 to 2-94		Undefined.			
2-95	Data	Simulator code used to generate the file, set to 100 for Schlumberger's ECLIPSE 100, 300 for Schlumberger's ECLIPSE 300, 500 for Schlumberger's ECLIPSE 300 Thermal, 700 for Schlumberger's INTERSECT simulator, 800 for Schlumberger's FrontSim simulator, or a negative value for other simulators. OPM Flow uses a value of 100.			IPROG
2-96 to 2-99		Undefined.			0
2-100	Data	REGDIMS keyword in the RUNSPEC section. The total maximum number of regions. The FIPNUM regions are defined by (2-90). if additional sets of fluid in-place regions are used, as per the FIPxxx series of fluid in-place region keywords, then NMFIPR is the sum of all FIP regions.			NMFIPR
2-101 to 2- 131					
2-132	Data	NETWORK keyword in the RUNSPEC section - maximum number of nodes in an extended network model or zero if the extended network option has not been activated.			NODMAX
2-133	Data	NETWORK keyword in the RUNSPEC section - maximum number of branches in an extended network model or zero if the extended network option has not been activated.			NBRMAX
2-134	Data	Number of items per branch in the IBRAN array when the extended network model is active, or zero if the extended network option has not been activated.			NIBRAN
2-135	Data	Number of items per branch in the RBRAN array when the extended network model is active, or zero if the extended network option has not been activated.			NRBRAN
2-136	Data	Number of items per node in the INODE array when the extended network model is active, or zero if the extended network option has not been activated.			NINODE
2-137	Data	Number of items per node in the RNODE array when the extended network model is active, or zero if the extended network option has not been activated.			NRNODE
2-138	Data	Number of items per node in the ZNODE array when the extended network model is active, or zero if the extended network option has not been activated.			NZNODE

No.	Keyword Name	RESTART Data - Header Keywords Table F.21			Status Or Value
		Keyword	No. of Entries	Data Type	
2-139	Data	Number of items in the INOBR array.			NINOBR
2-140 to 2-156		Undefined.			0
2-157	Data	The number of actions in the data set.			NO_ACT
2-158	Data	Maximum number of lines of schedule data for ACTION keyword - including ENDACTIO.			MAX_LINES
2-159	Data	Maximum number of eight character strings per input line of Action data (rounded up from input deck).			MAXSPRLINE
2-160 to 2-162		Undefined.			
2-163	Data	Maximum number of aquifer connections actually used in the model.			NGCAUS
2-164	Data	Maximum number of wells in the model.			NWMAXZ
2-165 to 2-174		Undefined.			0
2-175	Data	Number of multi-segmented wells defined with the WELSEG keyword in the SCHEDULE section, for when multi-segment wells have been activated, or zero otherwise.			NSEGWL
2-176	Data	WSEGDIMS keyword in the RUNSPEC section - maximum number of multi-segment wells, for when multi-segment wells have been activated, or zero otherwise.			NSWLMX
2-177	Data	WSEGDIMS keyword in the RUNSPEC section - maximum number of segments per multi-segment well, for when multi-segment wells have been activated, or zero otherwise.			NSEGMX
2-178	Data	WSEGDIMS keyword in the RUNSPEC section - maximum number of branches per multi-segment well, including the main branch, for when multi-segment wells have been activated, or zero otherwise.			NLBRMX
2-179	Data	Number of entries per segment in the multi-segment well ISEG array, for when multi-segment wells have been activated, or zero otherwise.			NISEGZ
2-180	Data	Number of entries per segment in the multi-segment well RSEG array, for when multi-segment wells have been activated, or zero otherwise. The number of data items per well segment in the RSEG restart vector is dependent on the number of active phases (oil, gas, water) in the simulation run. In particular, the following relations hold:			NRSEGZ

Scenario	Phases	NRSEGZ
Single Phase	Oil, Gas, or Water	126
Two Phase	Oil/Gas, Oil/Water, Gas/Water	134
Three Phase	Oil/Gas/Water	146

No.	Keyword Name	RESTART Data - Header Keywords Table F.21			Status Or Value
		Keyword	No. of Entries	Data Type	
2-181	Data	Number of entries per segment in the multi-segment well ILBR array, for when multi-segment wells have been activated, or zero otherwise.			NILBRZ
2-181 to 2-206		Undefined.			0
2-207	Data	The first part of the current simulation time in the form HH;MM:SS, that is HH for hours. The value should be greater than or equal to zero and less than or equal to 23.			IHOURZ
2-208	Data	The second part of the current simulation time in the form HH;MM:SS, that is MM for minutes. The value should be greater than or equal to zero and less than or equal to 59.			IMINTS
2-209 to 2-223		Undefined.			0
2-224	Data	Number of INTEGER lines in the AQUNUM data set.			NILAQN
2-225	Data	Number of REAL lines in the AQUNUM data set.			NIRAQN
2-226		Undefined.			0
2-227	Data	Number of entries in the AQUNUM data set.			NUMAQN
2-228 to 2-234		Undefined.			0
2-235	Data	Number of items in the ICOT array.			NICOTZ
2-236	Data	Number of items in the XCOT array.			NXCOTZ
2-237	Data	Number of items in the IWET array.			NIWETZ
2-238	Data	Number of items in the XWET array.			NXWETZ
2-239	Data	Number of items in the IGRT array.			NIGRTZ
2-240	Data	Number of items in the XGRT array.			NXGRTZ
2-241	Data	Number of tracers in the model plus two.			NSTRA2
2-242 to 2-245		Undefined.			0
2-246	Data	Maximum number of conditions per action.			MAXACTC
2-253	Data	Maximum umber of analytical aquifers in the model.			MAAQID
2.254 to 2-262		Undefined.			0
2-263	Data	No of Field UDQ data (parameters)			NOFUDQS
2-264	Data	No of Group UDQ data (parameters)			NOGUDQS

No.	Keyword Name	RESTART Data - Header Keywords Table F.21			Status Or Value
		Keyword	No. of Entries	Data Type	
2-265		Undefined.			
2-266		Undefined.			
2-267	Data	No of Well UDQ data (parameters)			NOWUDQS
2-268	Data	Integer seed value for the RAND			UDQPAR_I
2-269		Undefined.			
2-270		Undefined.			
2.271	Data	Maximum number of Chord segment links per multi-segment well for when multi-segment wells have been activated, or zero otherwise.			NCRDMX
2-272 to 2-290		Undefined.			0
2-291		Number of integer IUADs			NOIUADS
2-292		Number of integer IUAPs			NOIUAPS
2-293 to 2-296		Undefined.			
2-297	Data				RSEED
2-298 to 2-410		Undefined.			
2-411	Data	The third part of the current simulation time in the form HH;MM:SS, that is SS for seconds. The value should be greater than or equal to zero and less than or equal to 59.			ISECND
	LOGIHEAD	Global grid keyword for the RESTART Data file that defines the logical variables (T for true and F for false) for this time step. NLHEAD is the number of entries associated with this keyword, and is dependent on the features being used in the model. Undefined items should be set to F (False).			Required
I-1	Format	LOGIHEAD	NLHEAD	LOGI	
2-1	Data	Dissolved gas is present in the model indicator, set to T if present in the model else set to F.			DISGAS
2-2	Data	Vaporized oil is present in the model indicator, set to T if present in the model else set to F.			VAPOIL
2-3	Data	SATOPTS in the RUNSPEC section - directional relative permeability curves are active in the model.			DIRECT
2-4	Data	SATOPTS keyword in the RUNSPEC section - reversible relative permeability curves (black-oil) are active in the model.			IRREVERS

No.	Keyword Name	RESTART Data - Header Keywords Table F.21 Keyword / No. of Entries / Data Type	Status Or Value
		Radial grid geometry has been used for a compositional model indicator, set to T or F. Note that currently OPM Flow does not support radial grids and therefore this item should be set to F.	RADIAL
2-5	Data	Radial grid geometry has been used for a black-oil model indicator, set to T or F. Note that currently OPM Flow does not support radial grids and therefore this item should be set to F.	RADIAL
		Reversible relative permeability (compositional) are active in the model.	IRREVERS
2-6		Undefined.	
2-7	Data	Enable hysteresis indicator.	HYSER
2.8 to 2-14		Undefined.	F
2-15	Data	The dual porosity option has been activated in the model indicator, set to T or F. Note that currently OPM Flow does not support dual porosity grids and therefore this item should be set to F.	DUALPORO
2-16		Undefined.	F
2-17	Data	ENDSCALE keyword in the RUNSPEC section - enable end-point scaling indicator.	ENDSCALE
2-18	Data	ENDSCALE keyword in the RUNSPEC section - directional end-point scaling indicator.	DIRECT
2-19	Data	ENDSCALE keyword in the RUNSPEC section - reversible end-point scaling indicator.	IRREVERS
2-20	Data	SCALECRS keyword in the PROPS section - alternative three point end-point scaling indicator.	SCALECRS
2-21 to 2-30		Undefined.	F
2-31	Data	Coal Bed Methane option has been activated in the black-oil model indicator, set to T or F. Note that currently OPM Flow does not support the Coal Bed Methane option and therefore this item should be set to F.	F
2-32 to 2-38		Undefined.	F
2-39	Data	Constant oil compressibility (PVCDO) indicator.	ConstCo
2-40 to 2-75		Undefined.	

No.	Keyword Name	RESTART Data - Header Keywords Table F.21			Status Or Value
		Keyword	No. of Entries	Data Type	
2-76	Data	Multi-segment wells are present in the model indicator.			HasMSWells
2-77 to 2-127		Undefined.			
2-128		Coal Bed Methane option has been activated in the compositional model indicator, set to T or F. Note that currently OPM Flow does not support the Coal Bed Methane option and therefore this item should be set to F.			F
	DOUBHEAD	Global grid keyword for the RESTART Data file that defines the double precision REAL variables for this time step. NDHEAD is the number of entries associated with this keyword, and is dependent on the features being used in the model. Undefined values should be set to zero.			Required
1-1	Format	DOUBHEAD	NDHEAD	DOUB	
2-1	Data	The simulation REPORT TIME STEP, expressed in days for all units, except for the LAB system of units which should be expressed in hours.			
2-2	Data	TUNING keyword in the SCHEDULE section - maximum length of next time step.			TSINIT
2-3	Data	TUNING keyword in the SCHEDULE section - maximum length of time step after next.			TSMAXZ
2-4	Data	TUNING keyword in the SCHEDULE section - minimum length of all time steps.			TSMINZ
2-5	Data	TUNING keyword in the SCHEDULE section - minimum length of all chopped time steps.			TSMCHP
2-6	Data	TUNING keyword in the SCHEDULE section - maximum growth rate a time step can be increased by, subject to the maximum allowable time step size set by TSMAXZ.			TSFMAX
2-7	Data	TUNING keyword in the SCHEDULE section - minimum decay rate a time step can be decreased by, subject to the minimum allowable time step size set by TSMINZ.			TSFMIN
2-8	Data	TUNING keyword in the SCHEDULE section - the decay rate a time step can be decreased by after the number of target iterations has been exceeded.			TSFCNV
2-9	Data	TUNING keyword in the SCHEDULE section - the time truncation error target.			TRGTTE
2-10	Data	TUNING keyword in the SCHEDULE section - thee non-linear convergence error.			TRGCNV
2-11	Data	TUNING keyword in the SCHEDULE section - the target material balance error.			TRGMBE
2-12	Data	TUNING keyword in the SCHEDULE section - the linear convergence error target.			TRGLCV

No.	Keyword Name	RESTART Data - Header Keywords Table F.21			Status Or Value
		Keyword	No. of Entries	Data Type	
2-13 to 2-16		Undefined			
2-17	Data	TUNING keyword in the SCHEDULE section - the maximum time truncation error.			XXXTTE
2-18	Data	TUNING keyword in the SCHEDULE section - the maximum non-linear convergence error.			XXXCNV
2-19	Data	TUNING keyword in the SCHEDULE section - the maximum mass balance error.			XXXMBE
2-20	Data	TUNING keyword in the SCHEDULE section - the maximum linear convergence error.			XXXLCV
2-21	Data	TUNING keyword in the SCHEDULE section - the maximum well flow convergence error.			XXXWFL
2-22 to 2-82		Undefined.			
2-83	Data	TUNING keyword in the SCHEDULE section - target pressure change within a time step.			TRGDPR
2-84	Data	TUNING keyword in the SCHEDULE section - time step growth factor of the time step after a convergence failure.			TFDIFF
2-85	Data	TUNING keyword in the SCHEDULE section - the maximum pressure change at the last Newton iteration.			DDPLIM
2.86	Data	TUNING keyword in the SCHEDULE section - the maximum saturation change at the last Newton iteration.			DDSLIM
2.87		Undefined.			
2-88	Data	GUIDERAT keyword in the SCHEDULE section - guide rate parameter A.			GRPAR_A
2-89	Data	GUIDERAT keyword in the SCHEDULE section - guide rate parameter B.			GRPAR_B
2-90	Data	GUIDERAT keyword in the SCHEDULE section - guide rate parameter C.			GRPAR_C
2-91	Data	GUIDERAT keyword in the SCHEDULE section - guide rate parameter D.			GRPAR_D
2-92	Data	GUIDERAT keyword in the SCHEDULE section - guide rate parameter E.			GRPAR_E
2-93	Data	GUIDERAT keyword in the SCHEDULE section - guide rate parameter F.			GRPAR_F
2-94 to 2-97		Undefined.			
2.98	Data	Guide rate parameter delay interval.			GRPAR_INT
2-99		Undefined.			
2-100	Data	TUNING keyword in the SCHEDULE section - the maximum throughput ratio over a time step.			THRURPT

No.	Keyword Name	RESTART Data - Header Keywords Table F.21			Status Or Value
		Keyword	No. of Entries	Data Type	
2-101	Data	TUNING keyword in the SCHEDULE section - maximum tolerable pressure change within a time step.			XXXDPR
2-102	Data	TUNING keyword in the SCHEDULE section - the target fluid in-place error in Local Grid Refinements.			TRGFIP
2-103	Data	TUNING keyword in the SCHEDULE section - target surfactant change when the Surfactant Model is active in the run.			TRGSFT
2-104 to 2-144		Undefined.			
2-145	Data	Guide rate parameter damping factor.			GRPAR_DAMP
2-146 to 2-161		Undefined.			
2-161	Data	The simulation START time, expressed in days for all units, except for the LAB system of units which should expressed in hours.			
2-162	Data	Cumulative time from the start of the simulation, expressed in days for all units, except for the LAB system of units which should expressed in hours. That is the sum of values (2-1) and (2-161).			
2-163 to 2-212		Undefined.			0
2-213	Data	UDQPARAM item number 2 (Permitted range (+/-) of user-defined quantities).			UDQPAR_2
2-214	Data	UDQPARAM item number 3 (Value given to undefined elements when outputting data).			UDQPAR_3
2-215	Data	UDQPARAM item number 4 (fractional equality tolerance used in == <= etc. functions).			UDQPAR_4

Notes:

1) Rows shaded in gray indicate the start of a keyword.
2) Cells colored red in the No. column indicate that this item is either "Not Used" or "Undefined".
3) For formatted output all character variables, including the keyword should be enclosed in single quotes, for example the DOUBHEAD keyword should be written out as 'DOUBHEAD'.

Table F.21: RESTART Data - Header Keywords

Example: RESTART Data - Header Keywords

The following example shows a typical formatted RESTART Data - Header keywords.

```
'INTEHEAD'         249 'INTE'
 -955283513      200400           2       -2345       -2345       -2345
      -2345       -2345          10          10           1         100
         10       -2345           7       -2345           0           1
          0           2           2           0           0           0
        110         108         109           3          97          93
      -2345       -2345          19          38          53       -2345
         97          93         146           5           0           1
         15          24           8           5           2           4
          0           0           0           0           0           0
          0           0           0           0           0           0
          0           0           0           0          19          10
       1982           0           0           0           1           0
          0           0           0           0           1          10
          0           0          12           1          25           1
      -2345       -2345           8           8           5           1
          2           1           2           0         100           2
          7           0         -11           1           0           1
          0           0           0          10          10           1
        100           1           1           1           1          10
         10           1          19           1           0           0
          0           0           0           1           1           1
          0           0           0           0           0           0
          0          14          10          10          16           1
          1           1           1           1           2           1
          1           1           1           1           1          31
        108           0           0           0           0           0
          0          50          10           4           5           9
          0           2           8           8          12           1
         25           1      -32767      -32767      -32767      -32767
          0           1           1           1          18          86
          5           1           1           1           1          18
         86      -32767      -32767           0           0         109
         53         146           8           0          19         110
          0           0           1           1          86           0
          0           0           0           0           0           1
          0           0           0           0           1           0
          0           1           0          -1          12           0
          0          10          13           1           0           0
          0           0           2           0           0        3600
          1           6           1          10           1          10
          1           1           1           0          30           3
         18          10           9
'LOGIHEAD'          79 'LOGI'
 T  F  F  T  F  F  F  F  T  F  F  F  F  F  F  F  T  F  T  F  F  F  F  F  F
 F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F
 F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F
 F  F  F  F
'DOUBHEAD'         185 'DOUB'
  0.00000000000000D+00   0.10000000000000D+01   0.36500000000000D+03
  0.10000000149012D+00   0.15000000596046D+00   0.30000000000000D+01
  0.30000001192093D+00   0.10000000149012D+00   0.10000000149012D+00
 ..............................................
 -0.10000000200409D+21   0.10000000000000D+01   0.10000000000000D+01
  0.10000000000000D+01   0.00000000000000D+00   0.00000000000000D+00
  0.10000000000000D+01   0.10000000000000D-03
```

End of Example

F.7.4 RESTART DATA – GROUP, WELL AND CONNECTION DATA KEYWORDS

The Restart Data – Well and Group Completion Keywords[409] define various parameters associated with groups, wells and connection keywords in the RESTART file for both global and LGR grids.

The structure for this type of keyword is based on the data type as summarized below:

1) The group data set keywords are specified in Table F.22 and consists of the IGRP, SGRP, XGRP, and ZGRP keywords.
2) Table F.23 contains the multi-segment well keywords that consist of the ISEG, RSEG, ILBS, ILBR and ICRD keywords.
3) For wells the keywords are IWEL, SWEL, XWEL and ZWEL and Table F.24 describes these keywords.
4) Well connection keywords are specified in Table F.25 and consist of the ICON, SCON, and XCON keywords.
5) Tracer keywords are listed in Table F.26 and consists of the ICOT, XCOT, IWET, XWET, IGRT, and XGRT keywords.
6) Finally, the network keywords INODE, IBRAN, INOBR, RNODE, RBRAN, and ZNODE are specified in Table F.27.

Note that if multi-segment wells, tracers and networks are absent from the simulation deck then this data will not be written to the RESTART Data file.

The following table outlines the keywords for groups.

No.	Keyword Name	RESTART Data – Group, Well And Connection Keywords Table F.22 - Group Keywords			Status Or Value
		Keyword	**No. of Entries**	**Data Type**	
	IGRP	Keyword defines the integer Group, well, and connection data for this time step. Undefined values should be set to zero.			Required
1-1	Format	IGRP	NSGRPZ x NGMAXZ	INTE	
2-1 to 2-NWGMAX	Data	Integer group data array IGRP(NIGRPZ,NGMAXZ) where NIGRPZ and NGMAXZ are defined on the INTEHEAD keyword. Entries 1 to NWGMAX define the index of each well in the group, if this is a well group, or the index of each child group in this group if it is a node group. Note that undefined items in this array may be set to zero.			
2 - (NWGMAX + 1)		The number of wells or child groups belonging to this group.			

[409] *The RESTART Data – Group, Well and Connection keywords are used both for global and LGR grids with groups, well and connection data for the global grid, and wells and connection data for wells contained in the individual LGRs for the LGR grids. The LGR data is preceded by a series of LGR header keywords and terminated by an LGR termination keyword.*

No.	Keyword Name	RESTART Data – Group, Well And Connection Keywords Table F.22 - Group Keywords			Status Or Value
		Keyword	No. of Entries	Data Type	
2 - (NWG MAX + 2)		A parameter indicating production group control mode, set to: 3 : NONE or FIELD 4 : Oil rate. 5 : Water rate. 6 : Gas rate. 7 : Liquid rate. 8 : Reservoir volume rate.			
2 - (NWG MAX + 6)		A parameter that defines if the group is controlled by its own targets/limits, or is open to respond to higher level control, or is controlled by a higher level group (see details in code). 0 : If group is injection group.			
2 - (NWG MAX + 7)		A parameter that is dependent on the higher level group control active and guide rate defined for the group. 0 : No group guide rate, or the group is the FIELD group. 1 : Unknown. 2 : Unknown. 3 : Unknown. 4 : Unknown. 5 : Unknown. 6 : Unknown. 7 : Unknown. 8 : Group guide rate defined and higher level guide rate control.			
2 - (NWG MAX + 8)		A parameter that defines which rate reduction option is used for a production group. If rate reduction is used, the value is generally four (4). 0 : FIELD group.			
2 - (NWG MAX + 11)		A parameter describing the production control item defined via item two, the TARGET variable, on the GCONPROD keyword: 0 : NONE or FIELD. 1 : Oil rate. 2 : Water rate. 3 : Gas rate. 4 : Liquid rate. 5 : Reservoir volume rate.			
2 - (NWG MAX + 17)		A parameter describing the water injection control defined via item three, the TARGET variable, on the GCONINJE keyword: 0 : NONE or FIELD. 1 : Rate. 2 : Reservoir volume rate. 3 : Re-injection fraction. 4 : Voidage replacement.			

No.	Keyword Name	RESTART Data – Group, Well And Connection Keywords Table F.22 - Group Keywords			Status Or Value
		Keyword	No. of Entries	Data Type	
2 - (NWG MAX + 18)		A parameter for a water injection group that defines if the group is controlled by its own targets/limits, or is open to respond to higher level control, or is controlled by a higher level group (see details in code).			
2 - (NWG MAX + 22)		A parameter dependent on the gas injection control defined via item three, the TARGET variable, on the GCONINJE keyword: 0 : NONE or FIELD. 1 : Rate. 2 : Reservoir volume rate. 3 : Re-injection fraction. 4 : Voidage replacement.			
2 - (NWG MAX + 23)		A parameter for a gas injection group that defines if the group is controlled by its own targets/limits, or is open to respond to higher level control, or is controlled by a higher level group (see details in code).			
2 - (NWG MAX + 27)		A variable that defines the group type with: 0 : for a well group,. 1 : for a node group.,For a satellite group, and 2 : for a slave group.			
2 - (NWG MAX + 28)		The level of the group, with 0 representing the Field group.			
2 - (NWG MAX + 29)		The index of the parent group.			
2 - (NWG MAX + 34)		The sum of the number of active production and injection wells for the group.			
2 - (NWG MAX + 89)		Group sequence number according to the sequence in the input deck, equals NGMAXZ for the FIELD group.			
	SGRP	Keyword defines the real group data array SGRP(NIGRPZ, NGMAXZ) where NIGRPZ and NGMAXZ are defined on the INTEHEAD keyword.			Required
1-1	Format	SGRP	NSGRPZ x NGMAXZ	REAL	
2-1 to 2-6		Undefined.			
2-7	Data	Group's oil production target/limit.			

No.	Keyword Name	RESTART Data – Group, Well And Connection Keywords Table F.22 - Group Keywords			Status Or Value
		Keyword	No. of Entries	Data Type	
2-8	Data	Group's water production target/limit.			
2-9	Data	Group's gas production target/limit.			
2-10	Data	Group's liquid production target/limit.			
2-11	Data	Group's oil surface volume injection rate target/limit.			
2-12	Data	Group's oil reservoir volume injection rate target/limit.			
2-13	Data	Group's oil re-injection fraction target/limit.			
2-14	Data	Group's oil voidage injection fraction target/limit.			
2-15		Undefined.			
2-16	Data	Group's water surface volume injection rate target/limit.			
2-17	Data	Group's water reservoir volume injection rate target/limit.			
2-18	Data	Group's water re-injection fraction target/limit.			
2-19	Data	Group's water voidage injection fraction target/limit.			
2-20		Undefined.			
2-21	Data	Group's gas surface volume injection rate target/limit.			
2-22	Data	Group's gas reservoir volume injection rate target/limit.			
2-23	Data	Group's gas re-injection fraction target/limit.			
2-24	Data	Group's gas voidage injection fraction target/limit.			
	XGRP	Keyword defines the double precision group data array XGRP(NIGRPZ,NGMAXZ) where NIGRPZ and NGMAXZ are defined on the INTEHEAD keyword.			Required
1-1	Format	XGRP	NSGRPZ x NGMAXZ	DOUB	
2-1	Data	Group's oil production rate.			
2-2	Data	Group's water production rate.			
2-3	Data	Group's gas production rate.			
2-4	Data	Group's liquid production rate.			
2-5		Undefined.			
2-6	Data	Group's water injection rate.			
2-7	Data	Group's gas injection rate.			
2-8		Undefined.			
2-9	Data	Group's producing water cut.			

No.	Keyword Name	RESTART Data – Group, Well And Connection Keywords Table F.22 - Group Keywords			Status Or Value
		Keyword	No. of Entries	Data Type	
2-10	Data	Group's producing gas/oil ratio.			
2-11	Data	Group's total cumulative oil production.			
2-12	Data	Group's total cumulative water production.			
2-13	Data	Group's total cumulative gas production.			
2-14	Data	Group's total cumulative reservoir voidage production.			
2-15	Data	Undefined.			
2-16	Data	Group's total cumulative water injection.			
2-17	Data	Group's total cumulative gas injection.			
2-18	Data	Group's total cumulative reservoir volume injection.			
2-19		Undefined.			
2-20		Undefined.			
2-21		Undefined.			
2-22		Undefined.			
2-23	Data	Group's oil production potential.			
2-24	Data	Group's water production potential.			
2-25 to 2-86		Undefined.			
2-86	Data	Group's producer guide rate for oil.			
2-87	Data	Group's producer guide rate for water.			
2-88	Data	Group's producer guide rate for gas.			
2-89	Data	Group's producer guide rate for reservoir voidage volume.			
2-90	Data	Group's injection guide rate for oil.			
2-91		Undefined.			
2-92	Data	Group's injection guide rate for water.			
2-93	Data	Second copy of group's injection guide rate for water. Not fully characterized.			
2-94	Data	Groups injection guide rate for gas.			
2-95 to 2-127		Undefined.			

No.	Keyword Name	RESTART Data – Group, Well And Connection Keywords Table F.22 - Group Keywords			Status Or Value
		Keyword	No. of Entries	Data Type	
2-128	Data	Second copy of group's producer guide rate for oil. Not fully characterized.			
2-129	Data	Second copy of group's producer guide rate for water. Not fully characterized.			
2-130	Data	Second copy of group's producer guide rate for gas. Not fully characterized.			
2-131	Data	Second copy of group's producer guide rate for reservoir voidage volume. Not fully characterized.			
2-132 to 2-135		Undefined.			
2-136	Data	Group's total cumulative oil production (observed/historical rates).			
2-137		Undefined.			
2-138		Undefined.			
2-139		Undefined.			
2-140	Data	Group's total cumulative water production (observed/historical rates).			
2-141	Data	Group's total cumulative water injection (observed/historical rates).			
2-142		Undefined.			
2-143		Undefined.			
2-144	Data	Group's total cumulative gas production (observed/historical rates).			
2-145	Data	Group's total cumulative gas injection (observed/historical rates).			
	ZGRP	Keyword defines the character GROUP data for this keyword.			Required
1-1	Format	ZGRP	NSGRPZ x NGMAXZ	CHAR	
2-1	Data	Character group data array ZGRP(NIGRPZ,NGMAXZ) where NIGRPZ and NGMAXZ are defined on the INTEHEAD keyword.			

Table F.22: RESTART Data – Group, Well and Connection Keywords (Groups)

The next table (Table F.23) contains the multi-segment well keywords that consist of the ISEG, RSEG, ILBS, ILBR and ICRD keywords.

No.	Keyword Name	RESTART Data – Group, Well And Connection Keywords Table F.23 - Multi-Segment Well Keywords			Status Or Value
		Keyword	No. of Entries	Data Type	
	ISEG	Integer multi-segment well data array ISEG (NISEGZ, NSEGMX, NSWLMX) where NISEGZ, NSEGMX, and NSWLMX are defined on the INTEHEAD keyword.			Optional
1-1	Format	ISEG	NISEGZ x NSEGMX x NSWLMX	INTE	
2-1	Data	Segment number (one-based). Special ordering – see code for details.			
2-2	Data	Outlet segment (one-based). Defines the multi-segment outlet segment number starting with 0 for the segment nearest wellhead (NISEGZ = 2).			
2-3	Data	Inflow segment current branch (one-based).			
2-4	Data	Branch number (one-based). Defines the multi-segment branch for this segment number 1 for main stem and 0 if not active (NISEGZ = 4).			
2-5 to 2-11		Undefined.			
2-12	Data	Segment type.			
2-123 to 2-18		Undefined.			
2-19	Data	ICD scaling mode.			
2-20	Data	ICD open/shut flag.			
	RSEG	Double precision multi-segment well data array RSEG (NISEGZ, NSEGMX, NSWLMX) where NISEGZ, NSEGMX, and NSWLMX are defined on the INTEHEAD keyword.			Optional
1-1	Format	RSEG	NISEGZ x NSEGMX x NSWLMX	DOUB	
2-1	Data	Segment's distance to outlet.			
2-2	Data	Segment's depth differential to outlet.			
2-3	Data	Internal diameter of segment.			
2-4	Data	Roughness parameter of segment.			
2-5	Data	Cross-sectional area of segment.			
2-6	Data	Physical volume of segment.			
2-7	Data	Segment's distance to BHP reference node.			
2-8	Data	Segment's depth differential to BHP reference node.			

No.	Keyword Name	RESTART Data – Group, Well And Connection Keywords Table F.23 - Multi-Segment Well Keywords			Status Or Value
		Keyword	No. of Entries	Data Type	
2-9	Data	Normalized total segment flow rate.			
2-10	Data	Normalized Water flow rate fraction.			
2-11	Data	Normalized Gas flow rate fraction.			
2-12	Data	Segment pressure.			
2-13 to 2-30		Undefined.			
2-31	Data	Very close to normalized water flow rate fraction.			
2-32 to 2-39		Undefined.			
2-40	Data	Undefined.			
2-41	Data	Length of valve.			
2-42	Data	Cross-sectional area of valve.			
2-43	Data	Valve's dimensionless flow coefficient.			
2-44	Data	Maximal cross-sectional valve area.			
2-45 to 2-86		Undefined.			
2-87	Data	Device base strength.			
2-88		Undefined.			
2-89	Data	Calibrated fluid density.			
2-90	Data	Calibrated fluid viscosity.			
2-91	Data	Critical water fraction.			
2-92	Data	Transition region width.			
2-93	Data	Maximum emulsion ratio.			
2-94 to 2-97		Undefined.			
2-98	Data	Maximum valid flow rate.			

No.	Keyword Name	RESTART Data – Group, Well And Connection Keywords Table F.23 - Multi-Segment Well Keywords			Status Or Value
		Keyword	No. of Entries	Data Type	
2-99 to 2-102		Undefined.			
2-103	Data	ICD length.			
2-104	Data	Valve area fraction.			
2-105 to 2-111		Undefined.			
	ILBS	Integer multi-segment well data array for lateral branches ILBR (NLBRMX, NSWLMX) where NLBRMX and NSWLMX are defined on the INTEHEAD keyword.			Optional
1-1	Format	ILBS	NLBRMX x NSWLMX	INTE	
2-1 to no_branches	Data	The segment number of the first segment in the branch for branch number two and upwards.			
	ILBR	Integer multi-segment well data array for chords ILBR (NILBRZ, NLBRMX, NSWLMX) where NILBRZ, NLBRMX, and NSWLMX are defined on the INTEHEAD keyword.			Optional
1-1	Format	ILBR	NILBRZ x NLBRMX x NSWLMX	INTE	
2-1	Data	Outlet segment number.			
2-2	Data	Number of segments in the branch.			
2.3	Data	First segment in branch.			
2-4	Data	Last segment in branch.			
2-5	Data	Branch number minus one.			
	ICRD	Integer multi-segment well data array for chords ICRD (2, NCRDMX, NSWLMX) where NCRDMX, and NSWLMX are defined on the INTEHEAD keyword.			Optional
1-1	Format	ILBR	2 x NCRDMX x NSWLMX	INTE	
2-1	Data	ICRD (2, NCRDMX, NSWLMX) array.			

Table F.23: RESTART Data – Group, Well and Connection Keywords (Multi-Segment Wells)

The wells keywords are IWEL, SWEL, XWEL and ZWEL and Table F.24 below describes these keywords in detail.

No.	Keyword Name	RESTART Data – Group, Well And Connection Keywords Table F.24 - Well Keywords			Status Or Value
		Keyword	No. of Entries	Data Type	
	IWEL	Well integer data array IWEL(NIWELZ, NWELLS) where NIWELX and NWELS are defined on the INTEHEAD keyword. Undefined values should be set to zero.			
1-1	Format	IWEL	NIWELZ x NWELLS	INTE	Required
2-1	Data	Defines the location of the wellhead in the grid for the I dimension (NIWELZ = 1).			
2-2	Data	Defines the location of the wellhead in the grid for the J dimension (NIWELZ = 2).			
2-3	Data	Layer ID (one-based) of top/first connection.			
2-4	Data	Layer ID (one-based) of bottom/last connection.			
2-5	Data	Number of active cells connected to a well.			
2-6	Data	Index (one-based) of well's current group.			
2-7	Data	Defines the well type, set 1 for a production well, 2 for an oil injection well, 3 for a water injection well or 4 for a gas injection well (NIWELZ = 7).			
2-8	Data	Well's active target control mode (constraint).			
2-9		Undefined.			
2-10		Undefined.			
2-11	Data	Defines the well status, set to greater than zero for producing and less than or equal to zero for being shut (NIWELZ =11).			
2-12	Data	ID (one-based) of well's current VFP table.			
2-13		Undefined.			
2-14		Undefined.			
2-15		Undefined.			
2-16	Data	Well's requested control mode from simulation deck (WCONINJE, WCONPROD).			
2-17 to 2-22		Undefined.			
2-23	Data	Cross flow indicator.			
2-24 to 2-42		Undefined.			

No.	Keyword Name	RESTART Data – Group, Well And Connection Keywords Table F.24 - Well Keywords			Status Or Value
		Keyword	No. of Entries	Data Type	
2-43	Data	Defines the LGR index for a well with local completions in a LGR (NIWELZ = 43).			
2-44 to 2-48		Undefined.			
2-49	Data	Defines the well friction indicator, set to non-zero for horizontal wells (NIWELZ = 49).			
2-50	Data	Well's requested control mode from simulation deck (WCONHIST, WCONINJH).			
2-51 to 2-70		Undefined.			
2-71	Data	Multi-segment well ID; value of one for regular wells, or zero for multi-segment wells.			
2-72	Data	Number of well segments: value of one for regular wells, or the number of segments for multi-segment wells.			
2-73 to 2-98		Undefined.			
2-99	Data	Well's completion ordering scheme, set to: 0 : Connections ordered along well track (increasing MD) 1 : Connections ordered by increasing true vertical depth. Not really supported in OPM Flow. 2 : Connections listed in order of appearance in simulation model's COMPDAT keyword.			
	SWEL	Well real data array SWEL(NSWELZ, NWELLS) where NSWELX and NWELS are defined on the INTEHEAD record.			
1-1	Format	SWEL	NSWELZ x NWELLS	REAL	Required
2-1	Data	Well's current oil rate production target.			
2-2	Data	Well's current water rate production target.			
2-3	Data	Well's current gas rate production target.			
2-4	Data	Well's current liquid rate production target.			
2-5	Data	Well's current reservoir voidage rate production target.			
2-6	Data	Well's tubing head pressure target.			
2-7	Data	Well's bottom hole pressure target.			
2-8		Undefined.			

No.	Keyword Name	RESTART Data – Group, Well And Connection Keywords Table F.24 - Well Keywords			Status Or Value
		Keyword	No. of Entries	Data Type	
2-9		Undefined.			
2-10		Well's reference depth for BHP.			
2-11		Well's artificial lift quantity.			
2-12 to 2-17		Undefined.			
2-18		Well's drainage radius - item seven from the WELSPECS keyword.			
2-19 to 2-20		Undefined.			
2-25		Item two from WEFAC; this value is repeated at two locations.			
2-26 to 2-31		Undefined.			
2-32		Item two from WEFAC keyword.			
2-33		Undefined.			
2-34		Well's historical/observed liquid rate target/limit.			
2-35 to 2-54		Undefined.			
2-55		Well's historical/observed gas rate target/limit.			
2-56		Well's historical/observed bottom hole pressure target/limit.			
	XWEL	Well's injection and production data array XWEL(NXWELZ, NWELLS).			
1-1	Format	XWEL	NSWELZ x NWELLS	DOUB	Required
2-1	Data	Well's oil production rate.			
2-2	Data	Well's water production rate.			
2-3	Data	Well's gas production rate.			
2-4	Data	Well's liquid production rate.			
2-5	Data	Well's reservoir voidage production rate.			
2-6	Data	Well's tubing head pressure.			
2-7	Data	Well's flowing/producing bottom hole pressure.			
2-8	Data	Well's producing water cut.			

No.	Keyword Name	RESTART Data – Group, Well And Connection Keywords Table F.24 - Well Keywords			Status Or Value
		Keyword	No. of Entries	Data Type	
2-9	Data	Well's producing gas/oil ratio.			
2-10 to 2-18		Undefined.			
2-19	Data	Well's total cumulative oil production.			
2-20	Data	Well's total cumulative water production.			
2-21	Data	Well's total cumulative gas production.			
2-22	Data	Well's total cumulative reservoir voidage production.			
2-23		Undefined.			
2-24	Data	Well's total cumulative water injection.			
2-25	Data	Well's total cumulative gas injection.			
2-26	Data	Well's total cumulative reservoir volume injection.			
2-27 to 2-34		Undefined.			
2-35	Data	Well's producing gas formation volume factor.			
2-36		Undefined.			
2-37		Undefined.			
2-38		Undefined.			
2-39 to 2-41		Undefined.			
2-42	Data	Well's current BHP Target/Limit.			
2-43 to 2-48		Undefined.			
2-49	Data	Well's "primary" guide rate (oil for producers, preferred phase for injectors).			
2-50	Data	Well's producer guide rate for water.			
2-51	Data	Well's producer guide rate for gas.			
2-52 to 2-68	Data	Undefined.			

No.	Keyword Name	RESTART Data – Group, Well And Connection Keywords Table F.24 - Well Keywords			Status Or Value
		Keyword	No. of Entries	Data Type	
2-69	Data	Well's producer guide rate for reservoir voidage volume.			
2-70 to 2-75		Undefined.			
2-76	Data	Well's total cumulative oil production (observed/historical rates).			
2-77	Data	Well's total cumulative water production observed/historical rates).			
2-78	Data	Well's total cumulative gas production (observed(historical rates).			
2-79 to 2-81		Undefined.			
2-82	Data	Well's total cumulative water injection (observed/historical rates).			
2-83	Data	Well's total cumulative gas injection (observed/historical rates).			
2-84 to 2-91		Undefined.			
2-92		Second copy of well's primary guide rate. Not fully characterized.			
2-93	Data	Second copy of well's producer guide rate for water. Not fully characterized.			
2-94	Data	Second copy of well's producer guide rate for gas. Not fully characterized.			
2-95	Data	Second copy of well's producer guide rate for reservoir voidage. Not fully characterized.			
2-96 to 2-122		Undefined.			
2-123	Data	Well's voidage production rate.			
2-124	Data	Well's voidage production rate.			
	ZWEL	Well character data array ZWEL(NXWELZ, NWELLS) where NXWELX and NWELS are defined on the INTEHEAD keyword. All strings are eight characters in length.			
1-1	Format	ZWEL	NSWELZ x NWELLS	CHAR	Required
2-1	Data	Defines the well name consisting of eight characters (NXWELZ = 1).			
2-2	Data	Defines the name of the Well List the well belongs to, set to "blank" for the well does not belong to Well List (NXWELZ = 2), that is the ACTIONX name.			

No.	Keyword Name	RESTART Data – Group, Well And Connection Keywords Table F.24 - Well Keywords			Status Or Value
		Keyword	No. of Entries	Data Type	
2-3	Data	Defines the end of time step action for the well (NXWELZ = 3).			

Table F.24: RESTART Data – Group, Well and Connection Keywords (Wells)

The well connection keywords are specified in Table F.25 and consist of the ICON, SCON, and XCON keywords.

No.	Keyword Name	RESTART Data – Group, Well And Connection Keywords Table F.25 - Connection Keywords			Status Or Value
		Keyword	No. of Entries	Data Type	
	ICON	Connection integer data array ICON(NICONZ, NCWMAX, NWELLS) where NICONZ, NCWMAX, and NWELLS are defined on the INTEHEAD keyword. Data keywords are required for each completion in the well and the number of data items, N, should be NICONZ x NCWMAX x NWELLS. Undefined values should be set to zero.			
1-1	Format	ICON	N	INTE	
2-1	Data	Defines the well connect index (NICONZ = 1).			0
2-2	Data	Defines the location of connection in the grid for the I dimension (NICONZ = 2).			0
2-3	Data	Defines the location of connection in the grid for the J dimension (NICONZ = 3).			0
2-4	Data	Defines the location of connection in the grid for the K dimension (NICONZ = 4).			0
2-5		Undefined.			0
2-6	Data	Defines the status of the connection, set to less than or equal to zero for being shut or greater than zero for being open (NICONZ = 5).			0
2-7		Drainage saturation function table number.			0
2-8		Undefined.			0
2-9		Undefined.			0
2-10		Imbibition saturation function table number.			0
2-11		Undefined.			0
2-12		Undefined.			0
2-13		Completion ID (1-based).			0
2-14	Data	Defines the direction of connection, set to one for the x-direction, two for the y-direction, three for the z-direction. The default value of zero indicates the z-direction (NICONZ = 14).			0

No.	Keyword Name	RESTART Data – Group, Well And Connection Keywords Table F.25 - Connection Keywords			Status Or Value
		Keyword	No. of Entries	Data Type	
2-15	Data	Defines, for multi-segment wells, the segment number the connect belongs to, for conventional wells the default value of zero should be used (NICONZ = 15).			
	SCON	Connection real data array SCON(NSCONZ, NCWMAX, NWELLS) where NSCONZ, NCWMAX, and NWELLS are defined on the INTEHEAD keyword. Data are required for each completion in the well and the number of data items, N, should be NSCONZ x NCWMAX x NWELLS.			Required
1-1	Format	ICON	N	REAL	
2-1	Data	Connection transmissibility factor NSCONZ = 1 in the SCON array.			
2-2	Data	Connection center depth.			
2-3	Data	Connection diameter.			
2-4	Data	Connection KH, that is the net thickness times the permeability, NSCONZ = 4 in the SCON array.			
2-5	Data	Skin factor - item 'SKIN' from COMPDAT keyword in the SCHEDULE section.			
2-6 to 2-11		Undefined.			
2-12	Data	Connection transmissibility factor.			
2-13 to 2-20		Undefined.			
2-21	Data	Distance to end of connection in segment.			
2-22	Data	Distance to start of connection in segment.			
2-23 to 2-29		Undefined.			
2-30	Data	Undefined.			
2-31	Data	Undefined.			
2-32 to 2-40		Undefined.			
2-41	Data	Connection factor defined in input deck (set to one), or zero for undefined in input deck.			

No.	Keyword Name	RESTART Data – Group, Well And Connection Keywords Table F.25 - Connection Keywords			Status Or Value
		Keyword	No. of Entries	Data Type	
	XCON	Connection double precision data array XCON(NXCONZ, NCWMAX, NWELLS) where NXCONZ, NCWMAX, and NWELLS are defined on the INTEHEAD keyword. The number of data items, N, should be NXCONZ x NCWMAX x NWELLS			Required
1-1	Format	ICON	N	DOUB	
2-1	Data	Oil surface rate.			
2-2	Data	Water surface rate.			
2-3	Data	Gas surface rate.			
2-4 to 2-34		Undefined.			
2-35		Connection pressure value.			
2-36 to 2-49		Undefined.			
2-50		Reservoir voidage rate.			

Table F.25: RESTART Data – Group, Well and Connection Keywords (Connections)

The tracer keywords are listed in Table F.26 and consists of the ICOT, XCOT, IWET, XWET, IGRT, and XGRT keywords. Not this series of keywords are only available if tracers are present in the run.

No.	Keyword Name	RESTART Data – Group, Well And Connection Keywords Table F.26 - Tracer Keywords			Status Or Value
		Keyword	No. of Entries	Data Type	
	ICOT	Tracer connection integer data array ICOT(NICOTZ, NSTRA2, NCWMAX, NWELLS) where NICOTZ, NSTRA2, NCWMAX, and NWELLS are defined on the INTEHEAD keyword. The number of data items, N, should be NICONZ x NSTRA2 x NCWMAX x NWELLS.			Optional
1-1	Format	ICOT	N	INTE	
2-1	Data	ICOT(NICOTZ, NSTRA2, NCWMAX, NWELLS) array.			
	XCOT	Tracer connection double precision data array ICOT(NICOTZ, NSTRA2, NCWMAX, NWELLS) where NICOTZ, NSTRA2, NCWMAX, and NWELLS are defined on the INTEHEAD keyword. The number of data items, N, should be NICONZ x NSTRA2 x NCWMAX x NWELLS.			Optional
1-1	Format	XCOT	N	DOUB	
2-1	Data	XCOT(NICOTZ, NSTRA2, NCWMAX, NWELLS) array.			

No.	Keyword Name	RESTART Data – Group, Well And Connection Keywords Table F.26 - Tracer Keywords			Status Or Value
	IWET	Tracer well integer data array IWET(NIWETZ, NSTRA2, NWMAXZ, NWELLS) where NIWETZ, NSTRA2, and NWMAXZ are defined on the INTEHEAD keyword. The number of items, N, should be NIWETNZ x NSTRA2 x NWMAXZ.			Optional
1-1	Format	IWET	N	INTE	
2-1	Data	IWET(NIWETZ, NSTRA2, NWMAXZ, NWELLS) array.			
	XWET	Tracer connection double precision data array XWET(NXWETZ, NSTRA2, NWMAXZ) where NXWETZ, NSTRA2, and NWMAXZ are defined on the INTEHEAD keyword. The number of items, N, should be NXWETNZ x NSTRA2 x NWMAXZ.			Optional
1-1	Format	XWET	N	DOUB	
2-1	Data	XWET(NXWETZ, NSTRA2, NWMAXZ) array.			
	IGRT	Tracer well integer data array IGRT(NIGRTZ, NSTRA2, NWMAXZ, NWELLS) where NIGRTZ, NSTRA2, and NWMAXZ are defined on the INTEHEAD keyword. The number of elements, N, should be NIGRTNZ x NSTRA2 x NWMAXZ.			Optional
1-1	Format	IGRT	N	INTE	
2-1	Data	IGRT(NIGRTZ, NSTRA2, NWMAXZ, NWELLS) array.			
	XGRT	Tracer connection double precision data array XGRT(NXGRTZ, NSTRA2, NWMAXZ) where NXGRTZ, NSTRA2, and NWMAXZ are defined in the INTEHEAD keyword. The number of data items, N, should be NXGRTNZ x NSTRA2 x NWMAXZ.			Optional
1-1	Format	XGRT	N	DOUB	
2-1	Data	XWET(NXWETZ, NSTRA2, NWMAXZ) array.			

Table F.26: RESTART Data – Group, Well and Connection Keywords (Tracers)

Finally, the network keywords INODE, IBRAN, INOBR, RNODE, RBRAN, and ZNODE are specified in Table F.27. Again, this series of keywords are only available if networks are present in the run.

No.	Keyword Name	RESTART Data – Group, Well And Connection Keywords Table F.27 - Network Keywords			Status Or Value
		Keyword	No. of Entries	Data Type	
	INODE	Network node integer data array INODE(NINODE, NODMAX) where NINODE and NODMAX are defined in the INTEHEAD keyword.			Optional
1-1	Format	INODE	NINODE x NODMAX	INTE	
2-1	Data	INODE(NINODE, NODMAX) array			
	IBRAN	Network node branch integer data array IBRAN(NIBRAN, NBRMAX) where NIBRAN and NBRMAX are defined in the INTEHEAD keyword.			Optional
1-1	Format	IBRAN	NIBRAN x NBRMAX	INTE	

No.	Keyword Name	RESTART Data – Group, Well And Connection Keywords Table F.27 - Network Keywords			Status Or Value
2-1	Data	IBRAN(NIBRAN, NBRMAX)			
	INOBR	Network node-branch integer data array INOBR(NIOBR) where NIOBR are defined in the INTEHEAD keyword.			Optional
1-1		INOBR	NIOBR	INTE	
2-1		INOBR(NIOBR) array.			
	RNODE	Network node double precision data array RNODE(NRNODE, NODMAX) where NRNODE and NODMAX are defined in the INTEHEAD keyword.			Optional
1-1	Format	RNODE	NRNODE x NODMAX	DOUB	
2-1	Data	RNODE(NRNODE, NODMAX) array.			
	RBRAN	Network node branch integer data array RBRAN(NRBRAN, NBRMAX) where NRBRAN and NBRMAX are defined in the INTEHEAD keyword.			Optional
1-1	Format	RBRAN	NRBRAN x NBRMAX	DOUB	
2-1	Data	RBRAN(NRBRAN, NBRMAX) array.			
	ZNODE	Network node character data array ZNODE(NZNODE, NODMAX) where NZNODE and NODMAX are defined in the INTEHEAD keyword.			Optional
1-1	Format	ZNODE	NZNODE x NODMAX	CHAR	
2-1	Data	ZNODE(NZNODE, NODMAX) array.			

Notes:

1) Rows shaded in gray indicate the start of a keyword.
2) Cells colored red in the No. column indicate that this item is either "Not Used" or "Undefined".
3) For formatted output all character variables, including the keyword, should be enclosed in single quotes, for example the ZNODE keyword should be written out as 'ZNODE '.

Table F.27: RESTART Data – Group, Well and Connection Keywords (Network)

Example: RESTART Data – Group, Well and Connection Keywords

The following example shows a typical formatted RESTART Data – Group, Well and Connection keyword data set, showing the IGRP, SGRP, XGRP and ZGRP keywords.

```
 'IGRP    '        2864 'INTE'
          1           0           0           0           0           0
          0           0           0           0           0           0
          0           0           0           0           0           0
          0           0           0           0           0           0
 ……….……..
          0           0           0           0           0           0
          0           0           0           0           0           0
          0           0           0           0           0           0
          0           0           0           0           0          16
         16           0           0           0           0           0
'SGRP    '        1488 'REAL'
  0.10000000E+21   0.10000000E+21  -0.10000000E+21   0.10000000E+21
  0.00000000E+00   0.00000000E+00   0.10000000E+21   0.10000000E+21
  0.10000000E+21   0.10000000E+21   0.10000000E+21   0.10000000E+21
 ……….……..
  0.00000000E+00   0.00000000E+00   0.10000000E+21   0.10000000E+21
  0.00000000E+00   0.10000000E+21   0.00000000E+00   0.00000000E+00
  0.00000000E+00   0.00000000E+00   0.00000000E+00   0.00000000E+00
  0.00000000E+00   0.00000000E+00   0.00000000E+00   0.10000000E+01
'XGRP    '        2784 'DOUB'
  0.000000000000000D+00   0.000000000000000D+00   0.000000000000000D+00
  0.000000000000000D+00   0.000000000000000D+00   0.000000000000000D+00
  0.000000000000000D+00   0.000000000000000D+00   0.000000000000000D+00
 ……….……..
  0.000000000000000D+00   0.000000000000000D+00   0.000000000000000D+00
  0.000000000000000D+00   0.000000000000000D+00   0.000000000000000D+00
  0.000000000000000D+00   0.000000000000000D+00   0.000000000000000D+00
  0.000000000000000D+00
'ZGRP    '          80 'CHAR'
'MANI-C  ' '        ' '        ' '        ' 'B1-DUMMY' '        '
'        ' '        ' '        ' 'MANI-D1 ' '        ' '        '
'        ' 'INJE    ' '        ' '        ' '        ' 'PROD    '
'        ' '        ' 'MANI-B1 ' '        ' 'MANI-B2 ' '        '
'MANI-D2 ' '        ' '        ' '        ' '        ' 'MANI-E1 ' '        '
'        ' '        ' '        ' 'MANI-E2 ' '        ' '        '
'        ' 'MANI-K1 ' '        ' '        ' '        ' '        ' 'MANI-K2 '
'        ' '        ' '        ' '        ' 'MANI-F  ' '        ' '        '
'        ' '        ' 'WI-GSEG ' '        ' '        ' '        ' '        '
'D2-DUMMY' '        ' '        ' '        ' '        ' 'FIELD   ' '        '
'        ' '        ' '        '
```

End of Example

F.7.5 RESTART DATA – UDQ AND ACTIONX KEYWORDS

The restart data for the UDQ and ACTIONX keywords are written out in a set of UDQ and ACTIONX arrays. If there are no UDQ or ACTIONX data, the corresponding arrays are not written to the restart file and only the relevant arrays are written out.

The structure for this type of keyword is based on the data type as summarized below:

1) The UDQ data set keywords are specified in Table F.28 and consists of the IUDQ, IUAD, ZUDN, ZUDL, IGPH, IUAP, DUDW, DUDG, and DUDF keywords.

2) The ACTIONX data set keywords are specified in Table F.29 and consists of the keywords IACT, SACT, ZACT, ZLACT, ZACN, IACN and SACN.

No.	Keyword Name	RESTART Data – UDQ And ACTIONX Keywords Table F.28 - UDQ Keywords			Status Or Value
		Keyword	No. of Entries	Data Type	
	IUDQ	Keyword defines the integer properties as per the UDQ variable.			Required
1-1	Format	IUDQ	No. UDQ's x 3	INTE	
2-1	Data	0 : ASSIGN UDQ 1 : Undefined 2 : DEFINE UDQ			
2-2	Data	-4 : ASSIGN UDQ (-1) times the number of operators in expression: DEFINE UDQ (see code for details)			
2-3	Data	Sequence number of UDQ pr type (F, G, etc.)			
	IUAD	This keyword defines the various controls for wells, groups, etc.			Required
1-1	Format	IUAD	No. IUAD's x 5	INTE	
2-1	Data	Set to: 1) 200000 + 19 for GCONPROD and ORAT 2) 300000 + 19 for GCONPROD and WRAT 3) 400000 + 19 for GCONPROD and GRAT 4) 500000 + 19 for GCONPROD and LRAT 5) 300000 + 4 for WCONPROD + ORAT 6) 400000 + 4 for WCONPROD + WRAT 7) 500000 + 4 for WCONPROD + GRAT 8) 600000 + 4 for WCONPROD + LRAT 9) 300000 + 3 for WCONINJE + ORAT 10) 400000 + 3 for WCONINJE + RATE (surface rate) 11) 500000 + 3 for WCONINJE + RESV (reservoir vol rate) 12) 1000000 + 27 for CECON + minimum oil rate			
2-2	Data	Sequence number of UDQ used (from input sequence) for the actual constraint/target.			
2-3	Data	Set to one (1).			

No.	Keyword Name	RESTART Data – UDQ And ACTIONX Keywords Table F.28 - UDQ Keywords			Status Or Value
		Keyword	No. of Entries	Data Type	
2-4	Data	Number of times the UDQ variable is used.			
2-5	Data	The sequence number for the first use of the actual UDQ, equal to one (1) plus the (sum over the earlier UDQ's of the number of uses of the respective UDQ's).			
	IUAP	The keyword defines the sequence number of the actual well, group etc. for each UDQ-Use in WCONPROD, GCONPROD, GCONINJE etc.			Required
1-1	Format	IUAP	No. IUAP's	INTE	
2-1	Data	Equals the well, group etc. sequence number for the UDQ-Variable used in actual target, limit.			
	IGPH	This keyword defines an integer number per group, and for the FIELD which is the last group in the list).			Required
1-1	Format	IGPH	Max Number Groups in the Field plus one.	INTE	
2-1	Data	The value of the integer denotes the phase (or component) for injection group control (GCONINJE): 0 : No injection control. 1 : Oil phase injection control. 2 : Water phase injection. 3 : Gas injection.			
	ZUDN	Keyword pairs of data for each UDQ.			Required
1-1	Format	ZUDN	No. UDQ's x 2	CHAR	
2-1	Data	UDQ name (maximum of eight characters per UDQ).			
2-2	Data	UDQ's units (maximum of eight characters per UDQ).			
	ZUDL	This keyword contains the EXPRESSION variable (item three) on the UDQ keyword in the input deck "Data for operation".			Required
1-1	Format	ZUDL	No. UDQ's x 16	CHAR	
2-1 to 2-16	Data	A character string that defines the EXPRESSION variable, "Data for operation", split into strings of eight characters each.			
	DUDW	Keyword containing the values of all the well UDQ's.			Required
1-1	Format	DUDW	No. Well UDQs x Max No. of Wells.	DOUBLE	
2-1 to no of well UDQ's	Data	The value of the UDQ for the actual well (repeated for all wells in the well input sequence).			
	DUDG	Keyword containing the values of all the group UDQ's.			Required

No.	Keyword Name	RESTART Data – UDQ And ACTIONX Keywords Table F.28 - UDQ Keywords			Status Or Value
		Keyword	No. of Entries	Data Type	
1-1	Format	DUDG	No. Group_UDQs x Max No. of Groups.	DOUBLE	
2-1 to no of group UDQ's	Data	The value of UDQ for the actual group (repeated for all groups in the group input sequence).			
	DUDF	Keyword containing the values of all the field UDQ's.			Required
1-1	Format	DUDF	No. Field UDQs	DOUBLE	
2-1 to no of field UDQ's	Data	Value of UDQ for field UDQs			

Table F.28: RESTART Data – UDQ Keywords

No.	Keyword Name	RESTART Data – UDQ And ACTIONX Keywords Table F.29 - ACTIONX Keywords			Status Or Value
		Keyword	No. of Entries	Data Type	
	IACT	The keyword defines the integer properties for the ACTIONX variable.			Required
1-1	Format	IACT	No. of Actions x 9	INTE	
2-1	Data	0 : Undefined.			
2-2	Data	Number of lines of SCHEDULE data.			
2-3	Data	1 : For condition and previous condition equal to AND, and combinations OR/AND 2 : For all conditions and previous conditions equal to OR			
2-4	Data	7 : Undefined.			
2-5	Data	0 : Undefined.			
2-6	Data	The number of times the action is triggered.			
2-7	Data	0 : Undefined.			
2-8	Data	0 : Undefined.			
2-9	Data	The number of conditions for the action.			
	IACN	Keyword defines various parameters for			Required
1-1	Format	IACN	No. Actions x 26 x Max No. Conditions per Action.	INTE	

No.	Keyword Name	RESTART Data – UDQ And ACTIONX Keywords Table F.29 - ACTIONX Keywords			Status Or Value
		Keyword	No. of Entries	Data Type	
2-1 to 2-10	Data	0 : Undefined.			
2-11	Data	The type of quantity for the condition: 1 : For a field quantity (number of flowing producing wells). 2 : For a well quantity 3 : For a (node) group quantity. 9 : For a well group quantity 10 : For the DAY. 11 : For the MONTH. 12 : For the YEAR.			
2-12	Data	Quantity type for the Right Hand Side ("RHS") quantity: 1 : For field variables. 2 : For well variables? 3 : For group variables. 8 : For constant values.			
2-13	Data	Index for relational operator (<, =, >): 0 : For LHS quantity greater RHS quantity. 1 : For LHS quantity less than or equal to RHS quantity.			
2-14	Data	Relates to operator: 1 : AND 2 : OR			
2-17	Data	Defines the operator used in ACTIONX for defined quantities: 1 : For the > operator. 2 : For the < operator. 3 : For the >= operator. 4 : For the <= operator. 5 : For the = operator.			

No.	Keyword Name	RESTART Data – UDQ And ACTIONX Keywords Table F.29 - ACTIONX Keywords			Status Or Value
		Keyword	No. of Entries	Data Type	
2-18	Data	The value relates to the operator for the triggering condtion and if the right hand quantity is a constant or not. There is one value for each condition and the value is determined as follows: 1) For the first condition in an ACTIONX statement iacn[17] = 0 2) For the second, third etc. conditions iacn[17] is determined using the following logic: If the previous condition has a constant rhs => iacn[17] = 0. If previous rhs is of type {W,G, F} and If previous boolean operator is AND => iacn[17] = 1 If previous boolean operator is OR => iacn[17] = 0 Note as mentioned previously, the No. column in this table and all tables in APPENDIX F:, the count is based on one (offset one); however, in C++ the base is zero (offset zero) and therefore C++ programmers must subtract one from this column to obtain the correct reference. Hence, the iacn[17] reference is the correct reference for C++.			
	SACT	Keyword defines various parameters for an ACTIONX keyword			Required
1-1	Format	SACT	No. Actions x 5	REAL	
2-1 to 2-5	Data	0 : Undefined			
	ZACT	Keyword defines the name of an action			Required
1-1	Format	ZACT	No. Actions x 4	CHAR	
2-1	Data	The name of an ACTION (maximum eight characters).			
	ZLACT	Keyword containing the actions defined for an ACTIONX keyword.			Required
1-1	Format	ZLACT	No_Actions x Max No. of Input Lines x Max No. Characters per Input Line.	CHAR	
2-1 to 2-?	Data	Contains the actions defined for an ACTIONX keyword.			
	ZACN	Keyword containing the EXPRESSION parameter (item 3) on the UDQ keyword, the "Data for operation".			Required
1-1	Format	ZACN	No_Actions x Max No. of Conditions x 13	CHAR	
2-1	Data	Left hand side quantity if (DAY, MONTH or YEAR).			
2-2	Data	Right hand side quantity if (Well or Group).			
2-3	Data	Operator.			
2-4	Data	Well name – left hand quantity, if it is a well quantity.			
2-5	Data	Well name – right hand quantity, if it is a well quantity.			

No.	Keyword Name	RESTART Data – UDQ And ACTIONX Keywords Table F.29 - ACTIONX Keywords			Status Or Value
		Keyword	No. of Entries	Data Type	
2-6	Data	Group name – left hand quantity, if it is a group quantity.			
2-7	Data	Group name – right hand quantity, if it is a group quantity.			
	SACN				Required
1-1	Format	SACN	No. of Actions x Max No. Conditions x 16	DOUBLE	
2-1	Data	0 : Undefined			
2-2	Data	0 : Undefined			
2-3	Data	Value of condition: 1 : If Month otherwise zero.			
2-4	Data	0 : Undefined			
2-5	Data	0 : Well or Field data, or 1 : Group data.			
2-6	Data	Value of right hand side.			
2-7	Data	Value of left hand side.			
2-8	Data	Value of right hand side.			
2-9	Data	Value of left hand slide.			
2-10	Data	Value of right hand side.			

Table F.29: RESTART Data – ACTIONX Keywords

Example: RESTART Data – Group, Well and Connection Keywords

The following example shows a typical formatted RESTART Data – UDQ – data.

```
'ZUDN    '          22 'CHAR'
'WUWI3   ' '        ' 'WUPR1   ' '        ' 'WUPR3   ' '        ' 'WUPR4   '
'        ' 'WUPR5   ' '        ' 'GUOPR1  ' '        ' 'GULPR1  ' '        '
'GUOPR2  ' '        ' 'GULPR2  ' '        ' 'GUOPR3  ' '        ' 'GULPR3  '
'        '
'ZUDL    '         176 'CHAR'
'(GOPR 'L' 'OWER' *0' '.931 *0.' '50 *1)  ' '        ' '        ' '        '
'        ' '        ' '        ' '        ' '        ' '        ' '        '
'        ' '        ' '1/(WWCT ' ''OP*' ) ' '        ' '        ' '        '
'        ' '        ' '        ' '        ' '        ' '        ' '        '
'        ' '        ' '        ' '        ' 'SORTA(WU' 'PR1)     ' '        '
'        ' '        ' '        ' '        ' '        ' '        ' '        '
'        ' '        ' '        ' '        ' '        ' '        ' '1/(WWCT '
''OPU*' )' '        ' '        ' '        ' '        ' '        ' '        '
'        ' '        ' '        ' '        ' '        ' '        ' '        '
'        ' 'SORTA(WU' 'PR4)     ' '        ' '        ' '        ' '        '
'        ' '        ' '        ' '        ' '        ' '        ' '        '
'        ' '        ' '        ' '(2500 - ' 'GOPR LOW' 'ER *0.03' ' - GOPR '
'UPPER *0' '.15) * 0' '.899     ' '        ' '        ' '        ' '        '
'        ' '        ' '        ' '        ' '        ' '        ' '(4500 - ' 'GOPR LOW'
'ER *0.03' ' - GOPR ' 'UPPER *0' '.15) * 0' '.899     ' '        ' '        '
'        ' '        ' '        ' '        ' '        ' '        ' '        '
'(2500 - ' 'GOPR LOW' 'ER *0.03' ' - GOPR ' 'UPPER *0' '.15) * 0' '.920     '
'        ' '        ' '        ' '        ' '        ' '        ' '        '
'        ' '        ' '(4500 - ' 'GOPR LOW' 'ER *0.03' ' - GOPR ' 'UPPER *0'
'.15) * 0' '.920     ' '        ' '        ' '        ' '        ' '        '
'        ' '        ' '        ' '        ' '(2500 - ' 'GOPR LOW' 'ER *0.03'
' - GOPR ' 'UPPER *0' '.15) * 0' '.880     ' '        ' '        ' '        '
'        ' '        ' '        ' '        ' '        ' '        ' '(4500 - '
'GOPR LOW' 'ER *0.03' ' - GOPR ' 'UPPER *0' '.15) * 0' '.880     ' '        '
'        ' '        ' '        ' '        ' '        ' '        ' '        '
'        '
'IUDQ    '          33 'INTE'
         2          -4          1          2          -1          2
         2          -1          3          2          -1          4
         2          -1          5          2          -4          1
         2          -4          2          2          -4          3
         2          -4          4          2          -4          5
         2          -4          6
```

End of Example

F.7.6 RESTART DATA – AQUIFER DATA KEYWORDS

This set of keywords are used to define the various aquifer arrays written out by OPM Flow. If there are no aquifers in the model then this set of keywords are skipped. If an analytical or numerical model aquifer is present in the model, a complete set of keywords, for a given aquifer type, should be written out. For example, if a numerical model is defined in the model then the IAQN and RAQN keywords should be written to the file.

The structure for this set of keywords is dependent on the aquifer type, for analytical aquifers the format is outlined in Table F.30 and numerical aquifers are characterized in Table F.31.

No.	Keyword Name	RESTART Data – Aquifer Data Keywords Table F.30 - Analytical Aquifer Keywords			Status Or Value
		Keyword	No. of Entries	Data Type	
	IAAQ	Analytical aquifers integer data array IAAQ(NIAAQZ, MAAQID) where NIAAQZ and MAAQIDR are defined in the INTEHEAD keyword.			Optional
1-1	Format	IAAQ	NIAAQZ x MAAQID	INTE	
2-1	Data	Aquifer connection number.			
2-2	Data	Water property table.			PVTNUM
2-3 to 2-9		Undefined.			
2-10	Data	Analytical aquifer type, set to zero for Fetkovich aquifers and one for CarterTracy aquifers.			
2-11	Data	Type Related 2.			
	SAAQ	Analytical aquifers real data array SAAQ(NSAAQZ, MAAQID) where NSAAQZ and MAAQIDR are defined in the INTEHEAD keyword.			Optional
1-1	Format	SAAQ	NSAAQZ x MAAQID	REAL	
2-1	Data	Total Compressibility.			COMP
2-2	Data	Fetkovich analytical aquifer pore volume.			PORV
		Carter-Tracy aquifer external radius.			RE
2-3	Data	Fetkovich analytical aquifer productivity index.			PI
		Carter-Tracy aquifer permeability.			PERM
2-4	Data	Fetkovich analytical aquifer time constant.			
		Carter-Tracy aquifer porosity.			PORO
2-5	Data	Analytical aquifer pressure.			PRESS
2-6	Data	Analytical aquifer datum depth.			DATUM
2-7	Data	Carter-Tracy aquifer thickness.			DZ
2-8	Data	Carter-Tracy aquifer influence angle.			ANGLE
2-9	Data	Carter-Tracy aquifer mass density of water.			

No.	Keyword Name	RESTART Data – Aquifer Data Keywords Table F.30 - Analytical Aquifer Keywords			Status Or Value
		Keyword	No. of Entries	Data Type	
2-10	Data	Carter-Tracy aquifer water viscosity.			
	XAAQ	Analytical aquifers double precision data array XAAQ(NXAAQZ, MAAQID) where NXAAQZ and MAAQIDR are defined in the INTEHEAD keyword.			Optional
1-1	Format	XAAQ	NXAAQZ x MAAQID	DOUB	
2-1	Data	Aquifer flow rate (AAQR: N).			
2-2	Data	Dynamic aquifer pressure (AAQP: N).			
2-3	Data	Liquid volume produced from the aquifer into the reservoir (AAQT: N).			
2-4	Data	Total aquifer influx coefficient across all aquifer connections.			
2-5	Data	Reciprocal time constant for Carter-Tracy aquifer.			
2-6	Data	Influx constant "beta" for Carter-Tracy aquifer.			
2.7	Data	Unknown.			
2-8	Data	Unknown.			
2-9	Data	Dimensionless time for Carter-Tracy aquifer (AAQTD: N)			
2-10	Data	Dimensionless pressure for Carter-Tracy aquifer (AAQPD: N)			
	IAQL	Analytical aquifers integer aquifer list data array IAQL(NIAQLX, MXNALI, MXAAQL) where NIAQLX, MXNALI, and MXAAQL are defined in the INTEHEAD keyword.			Optional
1-1	Format	IAQL	NIAQLX x MXNALI x MXAAQL	INTE	
2-1	Data	IAQL(NIAQLX, MXNALI, MXAAQL) array.			
	ZAQL	Analytical aquifers character aquifer list data array ZAQL(NZAQLX, MXNALI) where NZAQLX, and MXNALI are defined in the INTEHEAD keyword.			Optional
1-1	Format	ZAQL	NZAQLX x MXNALI	CHAR	
2-1	Data	ZAQL(NZAQLX, MXNALI) array.			
	ICAQ	Analytical aquifers integer aquifer connection data array MAAQID arrays, each of size ICAQ (NICAQZ, NGCAUS) where NICAQZ, NGCAUS, and MAAQID are defined in the INTEHEAD keyword.			Optional
1-1	Format	ICAQ	NICAQZ x NGCAUS x MAAQID	INTE	
2-1	Data	Reservoir connecting cell in the I-direction.			I
2-2	Data	Reservoir connecting cell in the J-direction.			J
2-3	Data	Reservoir connecting cell in the K-direction.			K

No.	Keyword Name	RESTART Data – Aquifer Data Keywords Table F.30 - Analytical Aquifer Keywords			Status Or Value
		Keyword	No. of Entries	Data Type	
2-4	Data	Connecting cell status, with active equal to one otherwise zero.			
2-5	Data	Connecting "face" of the reservoir cell, where I- = 1, I+ = 2, J- = 3, J+ = 4, K- = 5, and K+ = 6.			AQUFACE
	SCAQ	Analytical aquifers real aquifer connection data array MAAQID arrays, each of size SCAQ (NSCAQZ, NGCAUS) where NSCAQZ, NGCAUS, and MAAQID are defined in the INTEHEAD keyword.			Optional
1-1	Format	SCAQ	NICAQZ x NGCAUS x MAAQID	REAL	
2-1	Data	Connection's fraction of total aquifer influx coefficient.			
2-2	Data	Connection's effective face area divided by aquifer's total influx coefficient.			
	ACAQ	Analytical aquifers double precision aquifer connection data array MAAQID arrays, each of size aCAQ (NACAQZ, NGCAUS) where NACAQZ, NGCAUS, and MAAQID are defined in the INTEHEAD keyword.			Optional
1-1	Format	ACAQ	NACAQZ x NGCAUS x MAAQID	DOUB	
2-1	Data	ACAQ(NSCAQZ, NGCAUS, MAAQID) array.			

Notes:

1) Rows shaded in gray indicate the start of a keyword.
2) For formatted output all character variables, including the keywords should be enclosed in single quotes, for example the ACAQ keyword should be written out as 'ACAQ '.

Table F.30: RESTART Data – Analytical Aquifer Data Keywords

And for numerical aquifers the format is outlined in Table F.31.

No.	Keyword Name	RESTART Data – Aquifer Data Keywords Table F.31 - Numerical Aquifer keywords			Status Or Value
		Keyword	No. of Entries	Data Type	
	IAQN	Numerical aquifers integer data array IAQN (NIIAQN, NUMAQN) where NIIAQN and NUMAQN are defined in the INTEHEAD keyword.			Optional
1-1	Format	IAAQ	NIIAQN x NUMAQN	INTE	
2-1	Data	Aquifer identification number.			AQUID
2-2	Data	Aquifer coordinate in the I-direction			I
2-3	Data	Aquifer coordinate in the J-direction			J
2-4	Data	Aquifer coordinate in the K-direction			K
2-5	Data	Aquifer Water Property Table.			PVTNUM
2-6	Data	Aquifer SATNUM table.			SATNUM
	RAQN	Numerical aquifers doubl eprecision data array RAQN (NIRAQN, NUMAQN) where NIIAQN and NUMAQN are defined in the INTEHEAD keyword.			Optional
1-1	Format	RAAQ	NIRAQN x NUMAQN	REAL	
2-1	Data	Aquifer inflow area.			AREA
2-2	Data	Aquifer length.			LENGTH
2-3	Data	Aquifer porosity			PORO
2-4	Data	Aquifer permeability.			PERM
2-5	Data	Aquifer datum depth.			DATUM
2-6	Data	Aquifer pressure.			PRESS
2-7	Data	Unknown			
2-8	Data	Unknown			
2-9	Data	Unknown			
2-10	Data	Total aquifer pore-volume.			
2-11	Data	Aquifer inflow rate (ANQR: N).			
2-12	Data	Total liquid volume produced from aquifer (AQNT: N).			
2-13	Data	Dynamic aquifer pressure (ANQP: N).			

Notes:

1) Rows shaded in gray indicate the start of a keyword.
2) For formatted output all character variables, including the keyword, should be enclosed in single quotes, for example the RAQN keyword should be written out as 'RAQN '.

Table F.31: RESTART Data – Numerical Aquifer Data Keywords

Example: RESTART Data – Aquifer Data Keywords

The following example shows a typical formatted RESTART Data – Aquifer Data keyword data set for an analytical aquifer.

```
 'IAAQ    '          15 'INTE'
          3           1           0           0           0           0
          0           0           0           1           1           1
          0           0           0
'SAAQ    '          24 'REAL'
  0.29999999E-04   0.24000000E+04   0.20000000E+02   0.10000000E+00
  0.49619409E+04   0.54000000E+04   0.14000000E+03   0.13333334E-01
  0.63151001E+02   0.31000000E+00   0.00000000E+00   0.43846342E+00
  0.00000000E+00   0.00000000E+00   0.00000000E+00   0.00000000E+00
  0.00000000E+00   0.00000000E+00   0.00000000E+00   0.00000000E+00
  0.00000000E+00   0.00000000E+00   0.00000000E+00   0.00000000E+00
'XAAQ    '           8 'DOUB'
  0.63857101611063D+03   0.49619408179004D+04   0.26831697328212D+05
  0.15000000000000D+06   0.23624684055200D-01   0.36097145428688D+02
  0.00000000000000D+00   0.00000000000000D+00
'ICAQNUM '           1 'INTE'
          1
'ICAQ    '          15 'INTE'
          5           1           1          13           2           5
          1           2          14           2           5           1
          3          15           2
'SCAQNUM '           1 'INTE'
          1
'SCAQ    '           6 'REAL'
  0.33333334E+00   0.33333334E+00   0.33333334E+00   0.33333334E+00
  0.33333334E+00   0.33333334E+00
```

End of Example

F.7.7 RESTART DATA – HIDDEN KEYWORD

This keyword[410] defines solution arrays that have been exported by OPM Flow that should not be processed by post-processing software.

The structure for this type of keyword is defined in Table F.32.

No.	Keyword Name	RESTART Data – Hidden Keyword Table F.32			Status Or Value
		Keyword	No. of Entries	Data Type	
	HIDDEN	The keyword defines the solution arrays that have not been exported by the simulator, where N is the number of solution variable names that have been "hidden", each enclosed in single quotes and of eight characters in length.			Required
1-1	Format	HIDDEN	N	CHAR	
2-1	Data	Character list of solution arrays to be hidden from post processing software.			
Notes: 1) For formatted output all character variables, including the keyword, should be enclosed in single quotes, for example the HIDDEN keyword should be written out as 'HIDDEN '.					

Table F.32: RESTART Data – Hidden Keyword

Example: RESTART Data – Hidden Keyword

The following example shows a typical formatted hidden data set, showing 51 Solution arrays that should be hidden.

```
'HIDDEN  '          51 'CHAR'
'ISTHGGI ' 'SWMAX   ' 'SWHY1   ' 'SWHY2   ' 'SWHY3   ' 'ISTHW   ' 'SOMAX   '
'SGHY1   ' 'SGHY2   ' 'SGHY3   ' 'ISTHG   ' 'SGMAX   ' 'SHMAX   ' 'WATQUIES'
'GASQUIES' 'PRESROCC' 'SPDOIL  ' 'SPDWATER' 'SPDGAS  ' 'SWINIT  ' 'SGINIT  '
'POLYMAX ' 'EIP0    ' 'EIP     ' 'TINIT   ' 'IOCN    ' 'SGMIN   ' 'SGSTRD  '
'SWSTRD  ' 'RISTRD  ' 'SGSTRI  ' 'SWSTRI  ' 'RISTRI  ' 'SGTRPD  ' 'SGSTRWD '
'SGSTRWI ' 'SWMIN   ' 'SGSTRWS ' 'RPTHMW  ' 'PPCW    ' 'GRADWELL' 'GRADCONN'
'SWHY4   ' 'SGHY4   ' 'GRADGRUP' 'GRADRESV' 'GRADFIP ' 'GRADSOL ' 'GRADEXP '
'IAQN    ' 'RAQN    '
```

End of Example

[410] *The RESTART Data – Hidden keyword format is used both for global and LGR grids.*

F.7.8 RESTART DATA - SOLUTION DATA KEYWORD

The solution data keywords[411] define the solution for each active cell for both global and LGR grids. This is then followed by a series of global data for a global entry or LGR data for a LGR entry.

The structure for this type of keyword is defined in Table F.33 for standard solution data and Table F.34 for tracers. A list of all solution names is tabulated in Table F.35.

No.	Keyword Name	RESTART Data - Solution Data Keyword Table F.33			Status Or Value
		Keyword	No. of Entries	Data Type	
	STARTSOL	This keyword marks the start of the solution variable section for both global and local grids. Note that there is no data associated with this keyword.			Required
1-1	Format	STARTSOL	0	MESG	
	LGRNAMES	This keyword defines a list of LGR names for the reporting time step, where N is the number of LGR names.			
1-1	Format	LGRNAMES	N	CHAR	
2-1	Data	A character list of LGR names with each name enclosed in single quotes and each having a length of eight characters.			
	SOLUTION	The SOLUTION keyword defines the start of a SOLUTION array data set, where SOLUTION is the name of the array being written out, as outlined in Table F.35. For example, PRESSURE, SWAT, SGAS, RS, RV etc. The SOLUTION keyword (items 1-1 and 2-1) is repeated for each SOLUTION array being written out. SOLUTION arrays can be in any order and only the active cells are written out (NACTIV).			Required
1-1	Format	SOLUTION	NACTIV	REAL	
2-1	Data	SOLUTION data set.			
	ENDSOL	This keyword marks the end of the solution variable section for both global and local grids. Note that there is no data associated with this keyword.			Required
1-1	Format	ENDSOL	0	MESG	

Notes:

1) Rows shaded in gray indicate the start of a keyword.
2) For formatted output all character variables, including the keyword, should be enclosed in single quotes, for example the SOLUTION array for PRESSURE should be written out as 'PRESSURE '.

Table F.33: RESTART Data - Solution Data keyword

For SOLUTION tracer concentration arrays the ZTRACER keyword must precede the SOLUTION tracer data array – see Table F.34 for a description of ZTRACER keyword.

[411] *The RESTART Data - Solution Data keywords are used both for global and LGR grids SOLUTION arrays. The LGR data is preceded by a series of LGR head keywords and terminated by an LGR termination keyword.*

No.	Keyword Name	RESTART Data – Solution Data Keyword For Tracer Concentration Name Table F.34			Status Or Value
		Keyword	No. of Entries	Data Type	
	ZTRACER	This keyword marks the start of the solution variable section for both global and local grids. The TRACER in the Data field is the name of the tracer concentration being processed.			Optional
1-1	Format	ZTRACER	2	CHAR	
2-1	Data	TRACER	' '		
	TRACER	TRACER is the name of the tracer concentration that is contained on the next keyword. Tracer arrays can be in any order and only the active cells are written out (NACTIV).			
1-1	Format	TRACER	NACTIV	REAL	
2-1	Data	Tracer solution data array.			

Notes:

1) The ZTRACER and the TRACER keywords are repeated for each tracer.
2) For formatted output all character variables, including the keyword, should be enclosed in single quotes, including the tracer name.

Table F.34: RESTART Data – Solution Data Keyword for Tracer Concentration Name

The SOLUTION keywords are listed in Table F.35.

Model Type		Model Option	Solution Data Keywords	Solution Variable Description	OPM Output Status
Black-Oil	Compositional				
		STANDARD	1OVERBO	Reciprocal of oil formation volume factor.	
		STANDARD	1OVERBW	Reciprocal of water formation volume factor.	
		STANDARD	CNV_DGAS	Worst cells depending on the gas saturation increment / Rv increment / Rs increment.	
		STANDARD	CNV_DPRE	Worst cells depending on the pressure increment.	
		STANDARD	CNV_DWAT	Worst cells depending on the water saturation increment.	
		STANDARD	CNV_GAS	Worst cells depending on the residual of gas equation.	
		STANDARD	CNV_OIL	Worst cells depending on the residual of the oil equation.	
		STANDARD	CNV_WAT	Worst cells depending on the residual of the water equation.	

Model Type		Model Option	Solution Data Keywords	Solution Variable Description	OPM Output Status
Black-Oil	Compositional				
		STANDARD	CONV_NEW	Number of Newtons required by each cell in order to satisfy the solution change convergence criteria at the last time step.	
		STANDARD	CONV_PRU	Worst cells depending on the pressure update.	
		STANDARD	CONV_VBR	Worst cells depending on the volume balance residual.	
		STANDARD	DRAINAGE	Drainage region numbers.	
		STANDARD	DRAINMIN	Drainage sink indicator.	
		STANDARD	FIPGAS	Gas fluid-in-place.	
		STANDARD	FIPOIL	Oil fluid-in-place.	
		STANDARD	FIPWAT	Water fluid-in-place.	
		STANDARD	GAS-DEN	Gas density.	
		STANDARD	GAS-POTN	Gas potential.	
		STANDARD	GAS-PRES	Gas phase pressure.	
		STANDARD	GAS-VISC	Gas viscosity.	
		STANDARD	IOVERBG	Reciprocal of gas formation volume factor.	
		STANDARD	ISTHG	Gas capillary pressure state.	
		STANDARD	ISTHW	Water capillary pressure state.	
		STANDARD	OIL-DEN	Oil density.	
		STANDARD	OIL-POTN	Oil potential.	
		STANDARD	OIL-VISC	Oil viscosity.	
		STANDARD	PBUB	Bubble point pressure.	
		STANDARD	PCOG	Oil-gas capillary pressure.	
		STANDARD	PCOW	Oil-water capillary pressure.	
		STANDARD	PDEW	Dew point pressure.	
		STANDARD	PORV	Pore volume at surface conditions.	
		STANDARD	PRESSURE	Pressure.	
		STANDARD	RFIPGAS	Gas fluid-in-place at reservoir conditions.	
		STANDARD	RFIPOIL	Oil fluid-in-place at reservoir conditions.	

Model Type		Model Option	Solution Data Keywords	Solution Variable Description	OPM Output Status
Black-Oil	Compositional				
■		STANDARD	RFIPWAT	Water fluid-in-place at reservoir conditions.	■
■		STANDARD	RPORV	Pore volume at reservoir conditions.	■
■		STANDARD	RS	Gas-oil ratio.	■
■		STANDARD	RSSAT	Saturated gas-oil ratio.	■
■		STANDARD	RV	Oil-gas ratio.	■
■		STANDARD	RVSAT	Saturated oil-gas ratio.	■
■		STANDARD	SFIPGAS	Gas fluid-in-place at surface/separator conditions.	■
	■	STANDARD	SFIPOIL	Oil fluid-in-place at surface/separator conditions.	■
	■	STANDARD	SFIPWAT	Water fluid-in-place at surface/separator conditions.	■
■	■	STANDARD	SGAS	Gas saturation.	■
■		STANDARD	SGMAX	Maximum gas saturation.	■
■		STANDARD	SGMIN	Minimum gas saturation.	■
■	■	STANDARD	SOIL	Oil saturation.	■
■		STANDARD	SOMAX	Maximum oil saturation.	■
■		STANDARD	SSOL	Solvent saturation.	■
■		STANDARD	STATES	Gas-oil state indicator.	■
■		STANDARD	SWAT	Water saturation.	■
■		STANDARD	SWMAX	Maximum water saturation.	■
■		STANDARD	SWMIN	Minimum water saturation.	■
■	■	STANDARD	WAT-DEN	Water density.	■
■	■	STANDARD	WAT-POTN	Water potential.	■
■	■	STANDARD	WAT-PRES	Water phase pressure.	■
■	■	STANDARD	WAT-VISC	Water viscosity.	■
	■	STANDARD	XMF	Liquid mole fractions.	■
	■	STANDARD	YMF	Vapor mole fractions.	■
	■	STANDARD	ZMF	Total mole fractions.	■
■		ALKALINE	ALKADS	Alkaline adsorption.	■

Model Type		Model Option	Solution Data Keywords	Solution Variable Description	OPM Output Status
Black-Oil	Compositional				
		ALKALINE	ALKALINE	Alkaline concentration.	
		ALKALINE	ALKMAX	Alkaline maximum historic concentration.	
		ALKALINE	PADMAX	Alkaline polymer maximum historic adsorption.	
		ALKALINE	PLADALK	Alkaline polymer adsorption multipliers.	
		ALKALINE	SFADALK	Alkaline surfactant adsorption multipliers.	
		ALKALINE	STMALK	Alkaline water/oil surface tension multipliers.	
		API	SDENO	Oil surface densities.	
		API	OILAPI	Oil API values.	
		AQUIFER	HYDH	Hydraulic head aquifer.	
		AQUIFER	HYDHFW	Fresh water hydraulic head aquifer.	
		BRINE	CNV_DSAL	Worst cells depending on the brine concentration increment.	
		BRINE	CNV_SAL	Worst cells depending on the residual of the brine equation (BRINEoption when used with polymer).	
		BRINE	ESALSUR	Effective salinity for surfactant.	
		BRINE	SALT	Brine concentration.	
		COAL	COALGAS	Coal gas concentration for coal bed methane option.	
		COAL	COALSOLV	Solvent concentration for coal bed methane option.	
		COAL	GASSATC	Initial coal gas saturated content for coal bed methane option.	
		COAL	MLANG	Langmuir scaling factors for coal bed methane option.	
		COAL	MLANGSLV	Langmuir scaling factors for solvent in coal bed methane option.	
		ENDSCALE	GASKR	Gas relative permeability.	
		ENDSCALE	OILKR	Oil relative permeability.	
		ENDSCALE	WATKR	Water relative permeability.	
		EXCAVATE	EXCAVNUM	Excavation status identifier.	
		EXCAVATE	TRANEXX/Y/Z	Transmissibilities in excavation runs.	

Model Type		Model Option	Solution Data Keywords	Solution Variable Description	OPM Output Status
Black-Oil	Compositional				
		FOAM	FOAM	Foam concentration.	
		FOAM	FOAM_HL	Foam half-life.	
		FOAM	FOAMADS	Foam adsorption.	
		FOAM	FOAMCNM	Foam capillary numbers.	
		FOAM	FOAMDCY	Foam decay.	
		FOAM	FOAMMAX	Foam maximum historic concentration.	
		FOAM	FOAMMOB	Foam mobility multiplier.	
		GIMODEL	CNV_DGGI	Worst cells depending on the dry injection gas parameter (Gi) increment.	
		GIMODEL	CNV_GGI	Worst cells depending on the residual of the Gi Pseudo-Compositional modeL.	
		GIMODEL	GGI	GI injected gas ratio.	
		GIMODEL	RFIPGGI	Dry injection gas in place at reservoir conditions.	
		GIMODEL	SFIPGGI	Dry injection gas in place at surface conditions.	
		POLYMER	CABINnnn	This is for use with the PLYTRRFA keyword only.	
		POLYMER	CNV_DPLY	Worst cells depending on the polymer concentration increment.	
		POLYMER	CNV_PLY	Worst cells depending on the residual of the polymer equation.	
		POLYMER	PADS	Adsorbed polymer concentrations.	
		POLYMER	POLYMAX	Maximum historic polymer concentration.	
		POLYMER	POLYMER	Polymer concentrations.	
		POLYMER	SFIPPLY,RFIPPLY	Polymer-in-place.	
		POLYMER	SFIPSAL,RFIPSAL	Salt-in-place (BRINE option when used with polymer).	
		ROCKCOMP	PRESROCC	Rock pressure values used for rock compaction model.	
		SOLVENT	CNV_DSOL	Worst cells depending on the solvent concentration increment.	
		SOLVENT	CNV_SOL	Worst cells depending on the residual of the solvent equation.	

Model Type		Model Option	Solution Data Keywords	Solution Variable Description	OPM Output Status
Black-Oil	Compositional				
		SOLVENT	RFIPSOL	Solvent-in-place at reservoir conditions.	
		SOLVENT	SFIPSOL	Solvent-in-place at surface conditions.	
		SURFACT	CATROCK	Divalent cation concentration associated with rock.	
		SURFACT	CATSURF	Divalent cation concentration associated with surfactant.	
		SURFACT	SURFACT	Surface interactions.	
		SURFACT	SURFADS	Adsorbed surfactant concentrations.	
		SURFACT	SURFCNM	Surfactant capillary numbers.	
		SURFACT	SURFMAX	Maximum surfactant concentrations.	
		SURFACT	SURFST	Surface tension in surfactant runs.	
		TEMP	TEMP	Temperature.	
OPM		THERMAL	TEMP	Temperature.	
		VE	EOGC	Maximum oil-gas contact.	
		VE	EOWC	Minimum oil-water contact.	
		VE	GWC	Gas-water contact.	
		VE	OGC	Oil-gas contact.	
		VE	OWC	Oil-water contact.	
		VE	POT_CORR	Initial contact corrected potential.	

Notes:

1) The Model Type columns indicate if the solution keyword applies to a black-oil model, a compositional model or both. Since OPM Flow is a black-oil model the compositional keywords are not valid, as shown by the orange cells under the Compositional column.
2) The Model Option column states the model option the solution keyword is associated with and a green cell in the Model Type column indicates that OPM Flow has this option.
3) Finally, the OPM Output Status column indicates if the current version of OPM Flow writes out the data associated with the solution keyword, with a green cell indicating it does and orange it does not.

Table F.35: RESTART Data - Solution Data Keywords

Example: RESTART Data – Solution Data Keyword

The following example shows a typical formatted global grid data set for this keyword set.

```
 'STARTSOL'           0 'MESS'
 'PRESSURE'       44431 'REAL'
   0.26889725E+03   0.26838983E+03   0.26826810E+03   0.26820352E+03
   0.26814493E+03   0.26809378E+03   0.26807767E+03   0.26806363E+03
   0.26802625E+03   0.26798474E+03   0.26795001E+03   0.26791434E+03
   0.26787915E+03   0.26783920E+03   0.26777118E+03   0.26769208E+03
.......................................
 'SWAT    '       44431 'REAL'
   0.10500000E+00   0.10500000E+00   0.10500000E+00   0.10500000E+00
   0.10500000E+00   0.10500000E+00   0.10500000E+00   0.14000000E+00
   0.14000000E+00   0.14500000E+00   0.14500000E+00   0.14500000E+00
   0.14500000E+00   0.14500000E+00   0.16000000E+00   0.16000000E+00
.......................................
 'SGAS    '       44431 'REAL'
   0.00000000E+00   0.89499998E+00   0.89499998E+00   0.89499998E+00
   0.89499998E+00   0.89499998E+00   0.89499998E+00   0.86000001E+00
   0.86000001E+00   0.85500002E+00   0.85500002E+00   0.85500002E+00
.......................................
 'RS      '       44431 'REAL'
   0.11697504E+03   0.12030303E+03   0.12023456E+03   0.12019825E+03
   0.12016528E+03   0.12013654E+03   0.12012746E+03   0.12011956E+03
   0.12009855E+03   0.12007520E+03   0.12005566E+03   0.12003561E+03
   0.12001582E+03   0.11999335E+03   0.11995509E+03   0.11991060E+03
   0.11987761E+03   0.11985194E+03   0.11983020E+03   0.11981972E+03
   0.11980798E+03   0.11977385E+03   0.11970672E+03   0.11963727E+03
  .......................................
 'RV      '       44431 'REAL'
   0.57879315E-04   0.57552861E-04   0.57483696E-04   0.57447018E-04
   0.57413723E-04   0.57384681E-04   0.57375513E-04   0.57367535E-04
   0.57346311E-04   0.57322733E-04   0.57302990E-04   0.57282738E-04
   0.57262747E-04   0.57240050E-04   0.57201407E-04   0.57156467E-04
   0.57123143E-04   0.57097215E-04   0.57075260E-04   0.57064670E-04
   0.57052810E-04   0.57018340E-04   0.56950528E-04   0.56880377E-04
.......................................
 'TRFIELD '          28 'DOUB'
   0.00000000000000D+00   0.00000000000000D+00   0.00000000000000D+00
   0.00000000000000D+00   0.00000000000000D+00   0.00000000000000D+00
   0.00000000000000D+00   0.00000000000000D+00   0.00000000000000D+00
   0.00000000000000D+00   0.00000000000000D+00   0.00000000000000D+00
   0.00000000000000D+00   0.00000000000000D+00   0.00000000000000D+00
   0.00000000000000D+00   0.00000000000000D+00   0.00000000000000D+00
   0.00000000000000D+00   0.00000000000000D+00   0.00000000000000D+00
   0.00000000000000D+00   0.00000000000000D+00   0.00000000000000D+00
   0.00000000000000D+00   0.00000000000000D+00   0.00000000000000D+00
   0.00000000000000D+00
 'ZTRACER '           2 'CHAR'
 'SEAF    ' '        '
 'SEAF    '       44431 'REAL'
   0.00000000E+00   0.00000000E+00   0.00000000E+00   0.00000000E+00
   0.00000000E+00   0.00000000E+00   0.00000000E+00   0.00000000E+00
   0.00000000E+00   0.00000000E+00   0.00000000E+00   0.00000000E+00
   0.00000000E+00   0.00000000E+00   0.00000000E+00   0.00000000E+00
   0.00000000E+00   0.00000000E+00   0.00000000E+00   0.00000000E+00
   0.00000000E+00   0.00000000E+00   0.00000000E+00   0.00000000E+00
   0.00000000E+00   0.00000000E+00   0.00000000E+00   0.00000000E+00
.......................................
```

End of Example

F.7.9 RESTART DATA - LGR SOLUTION DATA KEYWORDS

The file structure for this series of keywords is similar to the global grid RESTART Data keywords described previously. However, additional data that the defines the LGR properties (LGR Name for example) are included in this keyword definition and the LGR keyword is repeated for each LGR in the model. The keyword description is outlined in Table F.36.

Note that currently OPM Flow does not support LGR grids and therefore this keyword format cannot be used by OPM Flow.

No.	Keyword Name	RESTART Data - LGR Grid Property Data Keyword Table F.36 (Repeated For Each LGR In The Model)	Status Or Value
	LGR LGRHEADI LGRHEADQ LGRHEADD	LGR Grid Header keyword. See section F.5.1 RESTART Data - LGR Grid Header Keywords.	Required
	INTEHEAD LOGIHEAD DOUBHEAD	Header keyword. See section F.7.3 RESTART Data - Header Keywords.	
	IGRP SGRP etc.	Group, well, and connection data status keywords for this reporting time step. See section F.7.4 RESTART Data – Group, Well and Connection Data Keywords.	Required
	IAAQ SAAQ ect.	Aquifer definition arrays, only applicable if an aquifer is connected to an LGR. See section F.7.6 RESTART Data – Aquifer Data Keywords.	Optional
	HIDDEN	HIDDEN solution data, that is a list of solution arrays NOT to be read by post processing software. See section F.7.7 RESTART Data – Hidden Keyword.	Required
	SOLUTION	LGR SOLUTION array section (PRESSURE, SGAS, SOILS, WAT, etc.). See section F.7.8 RESTART Data - Solution Data keyword.	Required
	ENDSOL	LGR SOLUTION section termination keyword. See section F.7.8 RESTART Data - Solution Data keyword.	Required
	ENDLGR	LGR grid header section termination keywords. See section F.5.1 RESTART Data - LGR Grid Header Keywords.	Required

Notes:

1) Rows shaded in gray indicate the start of a keyword.
2) Cells colored orange in the No. column indicate that this item is not supported by OPM Flow.
3) For formatted output all character variables, including the keyword, should be enclosed in single quotes, for example the LGR keyword should be written out as 'LGR '.

Table F.36: RESTART Data – LGR Solution Data Keywords

This set of keywords are repeated for each LGR in the model with the last LGR data set terminated by the ENDSOL and ENDLGR keywords to mark the end of the LGR input - see the example.

Example: RESTART Data - LGR Solution Data Keywords

The following example shows a typical formatted data set with two LGR grids named LGR-1 and LGR-2.

```
 'LGR     '           1 'CHAR'
 'LGR-1   '
 'LGRHEADI'          45 'INTE'
           1         100       -2345       -2345       -2345       -2345
……….……..
       -2345       -2345       -2345
 'LGRHEADQ'           5 'LOGI'
  F  F  F  F  F
 'LGRHEADD'           5 'DOUB'
  0.00000000000000D+00  -0.10000000200409D+21  -0.10000000200409D+21
 -0.10000000200409D+21  -0.10000000200409D+21
 'INTEHEAD'         249 'INTE'
……….……..
 'LOGIHEAD'          79 'LOGI'
……….……..
 'DOUBHEAD'         185 'DOUB'
……….……..
 'IGRP    '         358 'INTE'
           0           0           0           0           0           0
……….……..
           0           0           0           0
 'SGRP    '         186 'REAL'
  0.00000000E+00   0.00000000E+00   0.00000000E+00   0.00000000E+00
……….……..
  0.00000000E+00   0.10000000E+01
 'XGRP    '         348 'DOUB'
  0.00000000000000D+00   0.00000000000000D+00   0.00000000000000D+00
……….……..
  0.00000000000000D+00   0.00000000000000D+00   0.00000000000000D+00
 'ZGRP    '          10 'CHAR'
 '        ' '        ' '        ' '        ' 'FIELD   ' '        '
 '        ' '        ' '
 'ICOT    '           0 'INTE'
 'XCOT    '           0 'DOUB'
 'IWET    '           0 'INTE'
 'XWET    '           0 'DOUB'
 'IGRT    '          18 'INTE'
……….……..
 'XGRT    '         180 'DOUB'
  0.00000000000000D+00   0.00000000000000D+00   0.00000000000000D+00
……….……..
  0.10000000200409D+21   0.10000000200409D+21   0.10000000200409D+21
 'DLYTIM  '          30 'DOUB'
  0.00000000000000D+00   0.00000000000000D+00   0.00000000000000D+00
……….……..
  0.00000000000000D+00   0.00000000000000D+00   0.00000000000000D+00
 'HIDDEN  '          51 'CHAR'
 'ISTHGGI ' 'SWMAX   ' 'SWHY1   ' 'SWHY2   ' 'SWHY3   ' 'ISTHW   ' 'SOMAX
'……….……..
 'IAQN    ' 'RAQN    '
 'STARTSOL'           0 'MESS'
 'PRESSURE'           9 'REAL'
  0.26823886E+03   0.26824997E+03   0.26826108E+03   0.26822879E+03
……….……..
 'SWAT    '           9 'REAL'
  0.10500000E+00   0.10500000E+00   0.10500000E+00   0.10500000E+00
……….……..
```

```
 'SGAS    '           9 'REAL'
   0.89499998E+00   0.89499998E+00   0.89499998E+00   0.89499998E+00
…….……..
 'RS      '           9 'REAL'
   0.12021812E+03   0.12022437E+03   0.12023062E+03   0.12021246E+03
…….……..
 'RV      '           9 'REAL'
   0.57467099E-04   0.57473411E-04   0.57479720E-04   0.57461380E-04
…….……..
 'ENDSOL  '           0 'MESS'
 'ENDLGR  '           1 'INTE'
           1
 'LGR     '           1 'CHAR'
 'LGR-2   '
 'LGRHEADI'          45 'INTE'
…….……..
 'LGRHEADQ'           5 'LOGI'
…….……..
 'LGRHEADD'           5 'DOUB'
…….……..
 'INTEHEAD'         249 'INTE'
…….……..
 'LOGIHEAD'          79 'LOGI'
…….……..
 'DOUBHEAD'         185 'DOUB'
…….……..
 'IGRP    '         358 'INTE'
…….……..
 'SGRP    '         186 'REAL'
…….……..
 'XGRP    '         348 'DOUB'
…….……..
 'ZGRP    '          10 'CHAR'
…….……..
 'ICOT    '           0 'INTE'
 'XCOT    '           0 'DOUB'
 'IWET    '           0 'INTE'
 'XWET    '           0 'DOUB'
 'IGRT    '          18 'INTE'
…….……..
 'XGRT    '         180 'DOUB'
…….……..
 'DLYTIM  '          30 'DOUB'
…….……..
 'HIDDEN  '          51 'CHAR'
…….……..
 'STARTSOL'           0 'MESS'
 'PRESSURE'           9 'REAL'
…….……..
 'SWAT    '           9 'REAL'
…….……..
 'SGAS    '           9 'REAL'
…….……..
 'RS      '           9 'REAL'
…….……..
 'RV      '           9 'REAL'
…….……..
 'ENDSOL  '           0 'MESS'
 'ENDLGR  '           1 'INTE'
           2
 'SEQNUM  '           1 'INTE'
           4
```

End of Example

F.8 RFT FILES – PRESSURE, SATURATION AND PRODUCTION LOG DATA

The Repeat Formation Tester ("RFT") file contains a well's depth pressure profile at the time the data is requested using the WRFT or WRFTPLT keywords in the SCHEDULE section. The data is meant to be used to compare the simulator's pressure profile with the field measured data collected from one of the open hole wire line logging tools: Formation Interval Tester ("FIT"), Repeat Formation Tester ("RFT") tool, Modular Dynamics Tester ("MDT"), or similar tool. The keywords also export the fluid saturations for the well connections at the same time.

In addition to the pressure and saturation data, the WRFTPLT keyword can also active the writing out of each well connection's fluid rates, connection factors and KH data, etc., collectively known as the Production Logging Tool ("PLT") data. The PLT data is used to compare with measured data from wire line production logging tools. Currently, output of the PLT data is not supported by OPM Flow.

F.8.1 RFT DATA FILE SPECIFICATION

The file structure for the RFT Data File consists of various keywords: a Time and Date keyword, a Well and Connection keyword followed by the requested optional keywords of RFT, PLT, Multi-Segment Well, and River keywords, that are written out depending on the data requested. A brief description of the keywords is given in Table F.37. A complete RFT data set is written to file for each well and time step as requested on the WRFT and WRFTPLT keywords.

Reference Section	Keyword Name	Keyword Type	Description
F.8.2	TIME	Time and Date Date	This data set contains the simulated time and date information that the date was written out.
	DATE		
F.8.3	WELLETC	Well and Connection Data	The Well and Connection keywords outlines the type and units being written out, the type of well and the well connections at the simulated time the data is written out to the RFT file.
	CONIPOS		
	CONJPOS		
	CONKPOS		
	HOSTGRID		
F.8.4	DEPTH	RFT Solution Data	RFT keyword data consists of the pressure and saturations for each declared connection in the well at the simulated time the data is written out to file.
	PRESSURE		
	SWAT		
	SGAS		
	CPLY		
	CPLAD		
	CBRI		

Reference Section	Keyword Name	Keyword Type	Description
F.8.5		PLT Solution Data	PLT series of keywords contain the production logging data which consists of the mid-point depth of each connection, the tubing length for the connection from the bottom-hole reference depth, the pressures and fluid rates for the connections. In addition, this data also includes the well pressures, fluid rates, connection transmissibility and KH vales, for the given simulated time. Currently this data set is not supported.
F.8.6		Multi-Segment Well Data	The Multi-Segment Well data set consists of pressure and flow rate data for segments and branches, currently this data set is not supported.
F.8.7		River Data	Data pertaining to the Rivers option, this option is not supported by OPM Flow.

Notes:

1) Reference Section cells colored orange indicate that this item is not supported by OPM Flow.

Table F.37: RFT Data File Format

F.8.2 RFT DATA FILE – TIME AND DATE KEYWORDS

The time and date keywords define the start of an RFT data set and declares the time and date for the subsequent property data written to file. The structure for these two keywords is summarized in Table F.38.

No.	Keyword Name	RFT Data File – Time And Date Keywords Table F.38			Status Or Value
		Keyword	No. of Entries	Data Type	
	TIME	This keyword marks the start of of an RFT file data set.			Required
1-1	Format	TIME	1	REAL	
2-1	Data	Simulation time the data is being written to file.			
	DATE	Three integer values representing the date of the data being written out.			Required
1-1	Format	DATE	3	INTE	
2-1	Data	Calendar day and should range between one and 31.			DAY
2-2	Data	Calendar month and should range between one and 12.			MONTH
2-3	Data	Calendar year in four digits, that is 2020.			YEAR

Notes:

1) Rows shaded in gray indicate the start of a keyword.
2) For formatted output all character variables, including the keyword should be enclosed in single quotes, for example the TIME should be written out as 'TIME '.

Table F.38: RFT Data File – Time and Date Keywords

The DATE keyword is immediate followed by the well and connection keywords.

F.8.3 RFT DATA FILE – WELL AND CONNECTION DATA KEYWORD

The well and connection data set of keywords specify the well and the well connection data as well as the system of units for the pressure and flow data being written to the RFT file. The structure for this set of keywords is outlined in Table F.39 and consists of WELLETC, CONIPOS, CONJPOS, CONKPOS and HOSTGRID keywords.

No.	Keyword Name	RFT Data File – Well And Connection Data Keyword Table F.39			Status Or Value
		Keyword	No. of Entries	Data Type	
	WELLETC	This keyword marks the start of the well and connection data set and comprises of 16 character variables all enclose in single quotes.			Required
1-1	Format	WELLETC	16	CHAR	
2-1	Data	Time units.			
2-2	Data	Well name.			
2-3	Data	Local Grid Refinement ("LGR") name or left blank if the well is not in an LGR. LGRs are not currently supported by OPM Flow.			Blank
2-4	Data	Depth units.			
2-5	Data	Pressure units.			
2-6	Data	Type of data being written out in subsequent keywords and should be set to R for RFT data, P for PLT data, or S for Segment data. The Segment data option is not currently supported in OPM Flow.			
2.7	Data	Well type data set to STANDARD for a standard well, FRICTION for a wellbore friction well, and MULTSEG for a multi-segment well. Only the STANDARD well option is currently supported by OPM Flow.			
2-8	Data	Liquid flow rate units.			
2-9	Data	Gas flow rate units.			
2-10	Data	Local volumetric flow rate units.			
2-11	Data	Flow velocity units.			
2-12	Data	Not used and should be a blank character string.			Blank
2-13	Data	Liquid and gas viscosity units.			
2-14	Data	Polymer and Brine concentration units.			
2-15	Data	Polymer and Brine flow rate units.			
2-16	Data	Adsorbed Polymer concentration units.			
	CONIPOS	This keyword defines the grid index in the x direction (I direction) for all NCON connections in the well associated with the RFT data set.			Required
1-1	Format	CONIPOS	NCON	INTE	
2-1	Data	Grid index in the I direction for each connection.			
	CONJPOS	This keyword defines the grid index in the y direction (J direction) for all NCON connections in the well associated with the RFT data set.			Required

No.	Keyword Name	RFT Data File – Well And Connection Data Keyword Table F.39			Status Or Value
		Keyword	No. of Entries	Data Type	
1-1	Format	CONJPOS	NCON	INTE	
2-1	Data	Grid index in the J direction for each connection.			
	CONKPOS	This keyword defines the grid index in the z direction (K direction) for all NCON connections in the well associated with the RFT data set.			Required
1-1	Format	CONKPOS	NCON	INTE	
2-1	Data	Grid index in the K direction for each connection.			
	HOSTGRID	Defines the LGR name for each of the NCON connections. LGR's have not been implemented in OPM Flow and therefore this keyword should contain NCON blank character strings.			Blank
1-1	Format	HOSTGRID	NCON	CHAR	Blank
2-1	Data	Connections host LGR grid name.			

Notes:

1) Rows shaded in gray indicate the start of a keyword.
2) For formatted output all character variables, including the keyword, should be enclosed in single quotes, for example the WELLETC should be written out as 'WELLETC '.

Table F.39: RFT Data File – Well and Connection Data Keyword

This series of keywords are immediate followed by the RFT solution keywords: DEPTH, PRESSURE, SWAT, and SGAS.

F.8.4 RFT DATA FILE – RFT SOLUTION DATA KEYWORD

The RFT solution keywords specify the actual solution data (pressure, saturation etc.) for each connection in the well, as described in Table F.40.

No.	Keyword Name	RESTART Data - Solution Data Keyword Table F.40			Status Or Value
		Keyword	No. of Entries	Data Type	
1-1	DEPTH	Connection grid block depth.			Required
1-1	Format	DEPTH	NCON	REAL	
2-1	Data	Depth data for each connection.			
	PRESSURE	Connection grid block pressure at the requested time.			Required
1-1	Format	PRESSURE	NCON	REAL	
2-1	Data	Pressure data for each connection.			
	SWAT	Connection grid block water saturation at the requested time.			Required
1-1	Format	SWAT	NCON	REAL	
2-1	Data	Water saturation data for each connection.			
2-4	SGAS	Connection grid block gas saturation at the requested time.			Required
1-1	Format	SGAS	NCON	REAL	
2-1	Data	Gas saturation data for each connection.			
	CPLY	Connection grid block polymer concentration at the requested time.			Optional
1-1	Format	CPLY	NCON	REAL	
2-1	Data	Polymer concentration data for each connection.			
	CPLAD	Connection grid block adsorbed polymer concentration at the requested time.			Optional
1-1	Format	CPLAD	NCON	REAL	
2-1	Data	Adsorbed polymer concentration data for each connection.			
	CBRI	Connection grid block brine concentration at the requested time.			Optional
1-1	Format	CBRI	NCON	REAL	
2-1	Data	Brine concentration data for each connection.			

Notes:

1) Rows shaded in gray indicate the start of a keyword.
2) Cells colored orange in the No. column indicate that this item is not supported by OPM Flow.
3) For formatted output all character variables, including the keyword, should be enclosed in single quotes, for example the PRESSURE keyword should be written out as 'PRESSURE '.

Table F.40: RFT Data File – RFT Solution Data Keywords

A complete standard RFT example showing the TIME, DATE, WELLECT, CONIPOS, CONJPOS, CONKPOS, DEPTH, PRESSURE, SWAT, and SGAS keywords, for one well at one time step is shown on the following page.

Example

The example shows all the keywords required to define a standard RFT data set for well B-4H which has 20 connection as of March 29, 1998. The units declared on the WELLETC keyword indicate the standard metric unit system.

```
'TIME    '           1 'REAL'
  0.14300000E+03
'DATE    '           3 'INTE'
         29           3        1998
'WELLETC '          16 'CHAR'
'  DAYS  ' 'B-4H    ' ' '        ' ' METRES ' '  BARSA ' 'R       ' 'STANDARD'
' SM3/DAY' ' SM3/DAY' ' RM3/DAY' ' M/SEC  ' ' '        ' '   CP    ' ' KG/SM3 '
' KG/DAY ' '  KG/KG '
'CONIPOS '          20 'INTE'
         10           10           10           10           10           10
         10           10           10            9           9            9
          9            9            9            9           9            9
          9            9
'CONJPOS '          20 'INTE'
         32           32           32           32           32           32
         32           32           32           32           32           32
         32           32           32           32           32           31
         31           31
'CONKPOS '          20 'INTE'
          1            2            3            5           6            7
          8            9           10           13           14           15
         16           17           18           19           20           20
         21           22
'HOSTGRID'          20 'CHAR'
'        ' '        ' '        ' '        ' '        ' '        ' '        '
'        ' '        ' '        ' '        ' '        ' '        ' '        '
'        ' '        ' '        ' '        ' '        ' '        '
'DEPTH   '          20 'REAL'
  0.25333979E+04   0.25452786E+04   0.25577705E+04   0.25742898E+04
  0.25797493E+04   0.25855930E+04   0.25918203E+04   0.25966951E+04
  0.26071150E+04   0.26206902E+04   0.26333235E+04   0.26461284E+04
  0.26576768E+04   0.26637878E+04   0.26665945E+04   0.26786790E+04
  0.27012888E+04   0.27033108E+04   0.27381892E+04   0.27746746E+04
'PRESSURE'          20 'REAL'
  0.27384171E+03   0.27406891E+03   0.27430649E+03   0.25063243E+03
  0.25072800E+03   0.25085713E+03   0.25126656E+03   0.25138820E+03
  0.25212048E+03   0.25290869E+03   0.25380568E+03   0.25471658E+03
  0.25669318E+03   0.25713290E+03   0.25733768E+03   0.26151834E+03
  0.26378452E+03   0.26398804E+03   0.28049472E+03   0.28405634E+03
'SWAT    '          20 'REAL'
  0.79952940E-01   0.10493817E+00   0.16989987E+00   0.10517924E+00
  0.10518920E+00   0.14012721E+00   0.14357868E+00   0.80211885E-01
  0.80138676E-01   0.80140039E-01   0.90355128E-01   0.19185907E+00
  0.12343995E+00   0.29019111E+00   0.14448708E+00   0.32566309E+00
  0.10000000E+01   0.10000000E+01   0.10000000E+01   0.10000000E+01
'SGAS    '          20 'REAL'
  0.92004704E+00   0.89506185E+00   0.83010012E+00   0.89231288E+00
  0.89236671E+00   0.14750090E+00   0.17341191E-01   0.64456672E-02
  0.00000000E+00   0.00000000E+00   0.00000000E+00   0.00000000E+00
  0.00000000E+00   0.00000000E+00   0.00000000E+00   0.00000000E+00
  0.00000000E+00   0.00000000E+00   0.00000000E+00   0.00000000E+00
```

End of Example

F.8.5 RFT DATA FILE – SOLUTION PLT DATA KEYWORDS

This data set is currently not supported.

F.8.6 RFT DATA FILE – SOLUTION MULTI-SEGMENT WELL KEYWORDS

This data set is currently not supported.

F.8.7 RFT DATA FILE – SOLUTION RIVER KEYWORDS

This data set is currently not supported.

F.9 SUMMARY FILES – TIME BASED VECTOR DATA

The SUMMARY files contain the variables requested to be written to the summary files via the keywords declared in the SUMMARY section. The data are used to generate line graphs of properties such as oil flow rate versus time, grid block pressure versus time, etc. See sections *11.2 Data Requirements* and *11.3 Keyword Definitions* that declare the variables to be written out to the SUMMARY files. The default behavior is write out the requested variables at each time step. As this can lead to large files, especially for full field simulation models, the *RPTONLY - Activate the Report Time Steps Only Option for the SUMMARY File* keyword allows one to write out the data only at a report time step instead.

There are three types of SUMMARY files, the first SUMMARY file is the OPM Flow specific Enhanced SUMMARY (*.ESMRY) output file that is optimized for fast loading of selected vectors by post-processing applications, including OPM ResInsight. This file type is standalone, that is there is no separate index file.

The other two files types associated with the time based vector data, is an index file (SUMMARY Index) that contains a list of variables and objects (wells, groups, connections etc.) that are to be written out to summary data file, and the summary data file (SUMMARY Data) that contains the actual data written out at the required time steps. The commercial simulator uses the term "MINISTEP" to describe the time the data is written, and as mentioned above, "MINISTEP" may be either a time step or a reporting time step.

F.9.1 ENHANCED SUMMARY DATA FILE KEYWORDS

Again, this is an OPM FLOW specific Enhanced SUMMARY (*.ESMRY) output file format that is optimized for fast loading of selected vectors by post-processing applications. The *.ESMRY file is re-written for every time step and load times are in the order of 30 times faster compared with fetching selected vectors using the *.UNSMRY file format in post processing software. The file type is compatible with OPM ResInsight and the option is activated via the enable-esmry=true command line option.

The structure for this series of keywords is outlined in Table F.41 and depends on the data requested to be written out to the SUMMARY files.

No.	Keyword Name	Enhanced SUMMARY File Keywords Table F.41			Status Or Value
		Keyword	No. of Entries	Data Type	
	START	Three integer values representing the start date of the simulation., and four integers representing the hour, minutes, seconds and micro seconds the start date.			Required
1-1	Format	START	7	INTE	
2-1	Data	First part of start date of the run, DAY, the value should be greater than or equal to one and less than or equal to 31 for the calendar days.			DAY
2-2	Data	Second part of start date of the run, MONTH, the value should be greater than or equal to one and less than or equal to 12 for the calendar month.			MONTH
2-3	Data	Third part of start date of run, YEAR, a positive four digit integer value of the start year, which must be specified fully by four digits, for example 1986.			YEAR
2-4	Data	The first part of the start time in the form HH;MM:SS:MS, that is HH for hours. The value should be greater than or equal to zero and less than or equal to 23.			IHOURZ
2-5	Data	The second part of the start time in the form HH;MM:SS:MS, that is MM for minutes. The value should be greater than or equal to zero and less than or equal to 59.			IMINTS

No.	Keyword Name	Enhanced SUMMARY File Keywords Table F.41			Status Or Value
		Keyword	No. of Entries	Data Type	
2-6	Data	The third part of the start time in the form HH;MM:SS:MS, that is SS for seconds. The value should be greater than or equal to zero and less than or equal to 59.			ISECND
2-7	Data	The fourth part of the start time in the form HH;MM:SS:MS, that is MS for milliseconds. The value should be greater than or equal to zero and less than or equal to 999.			
	RESTART	RESTART is only present for restarted runs and contains the root file name (absolute or relative path) to the case which this run was restarted from.			Optional
1-1	Format	RESTART	1	C0nn	
2-1	Data	The name of file, enclosed in quotes, which contains the name of the case the current run was re-started from. The length of the character string is set by the Data Type, C0nn, for example, C125 would mean that the RESTART filename character length is 125 characters.			
	RSTNUM	The RSTNUM keyword is only present for restart runs and defines the report time step this case was restarted from.			Optional
1-1	Format	RSTNUM	1	INTE	
2-1	Data	Restart report time step. number.			

	KEYCHECK	A list of all the summary variable mnemonics written to the file at each time step., where NLIST is the number of vectors and C0nn is the constant fixed length of each variable character string. For example:
		`'KEYCHECK' 6 'C010'`
		`'TIME ' 'YEARS ' 'FOPR ' 'GGOR:PROD '`
		`'TCPU ' 'WOPR:PROD1'`
		Where each vector name consists of 10 characters. For vectors associated with objects, groups, wells etc., the vector name is follows by a ":" and then the object name as shown in the above example.
					Required

1-1	Format	KEYCHECK	NLIST	C0nn	
2-1	Data	Each mnemonic string should be enclosed in quotes and have a character length of C0nn, for example 'C010' in the above example.			
	UNITS	Units			Required
1-1	Format	UNITS	NLIST	CHAR	
2-1	Data	Units for the associated summary variable mnemonics.			
	RSTEP	An array that holds one integer value for each time step. An integer value of one indicates time steps that are also report time steps and a zero indicates time steps only.			Required
1-1	Format	RSTEP	NSTEP	INTE	
2-1	Data	One value for each time step and the values should be either zero or one.			

No.	Keyword Name	Enhanced SUMMARY File Keywords Table F.41			Status Or Value
		Keyword	No. of Entries	Data Type	
	TSTEP	An array holding one integer value for each time step with the values representing the time step indices. Note that the values starts from zero, such that the Data will have values of the form 0, 1, 2,.. , NSTEP - 1, that is unless the RPTONLY keyword has been used in the SUMMARY section to write only the report time steps to the file. Values should range between 0 to NSTEP - 1			Required
1-1	Format	TSTEP	NSTEP	INTE	
2-1	Data	One value for each time step. with the value ranging from zero to NSTEP - 1.			
	VX	One data array (length = NSTEP) for each summary vector specified in array KEYCHECK. Naming of these arrays should be V0,V1,V2, … etc.			Required
1-1	Format	VX	NSTEP	REAL	
2-1	Data	One real value for each time step for one specific summary vector VX vector, where X ranges from zero to NLIST – 1, for the NLIST variables declared via the KEYCHECK keyword.			

Notes:

1) Keywords are order dependent.
2) For formatted output all character variables, including the keyword, should be enclosed in single quotes, for example the START and KEYCHECK keywords should be written out as 'START ' and 'KEYCHECK'.

Table F.41: Enhanced SUMMARY File Keywords

Example: Enhanced SUMMARY File Keywords

The following example shows a typical formatted Enhanced SUMMARY file set of keywords.

```
'START   '           7 'INTE'
          1          11        2018           0           0           0
          0
'KEYCHECK'           6 'C010'
'TIME      ' 'YEARS     ' 'FOPR      ' 'GGOR:PROD ' 'TCPU      ' 'WOPR:PROD1'
'UNITS   '           6 'CHAR'
'DAYS    ' '        ' 'SM3/DAY ' 'SM3/SM3 ' 'SECONDS ' 'SM3/DAY '
'RSTEP   '           4 'INTE'
          1          1           0           1
'TSTEP   '           4 'INTE'
          0          1           2           3
'V0      '           4 'REAL'
    1.0000000E+00    4.0000000E+00    9.0000000E+00    1.4000000E+01
'V1      '           4 'REAL'
    2.7378509E-03    1.0951404E-02    2.4640657E-02    3.8329910E-02
'V2      '           4 'REAL'
    0.0000000E+00    0.0000000E+00    0.0000000E+00    0.0000000E+00
'V3      '           4 'REAL'
    0.0000000E+00    0.0000000E+00    0.0000000E+00    0.0000000E+00
'V4      '           4 'REAL'
    6.0681427E-01    6.5710169E-01    7.0169014E-01    7.6383120E-01
'V5      '           4 'REAL'
    0.0000000E+00    0.0000000E+00    0.0000000E+00    0.0000000E+00
```

The example shows the file holding six summary vector and four time steps, with the first, second and fourth time steps being report time steps.

F.9.2 SUMMARY INDEX FILE KEYWORDS

The structure for this series of keywords is outlined in Table F.42, Table F.43, and Table F.44 and depends on the data requested to be written out to the SUMMARY Data File.

No.	Keyword Name	SUMMARY Index File Keywords (Global) Table F.42			Status Or Value
		Keyword	No. of Entries	Data Type	
	INTEHEAD	The integer header keyword contains two values indicating the units system and the simulator used to generate the file.			Required
1-1	Format	INTEHEAD	2	INTE	
2-1	Data	Unit system used in the simulation, set to 1 for metric, 2 for field, and 3 for laboratory			
2-2	Data	Simulator code used to generate the file, set to: 100 for Schlumberger's ECLIPSE 100, 300 for Schlumberger's ECLIPSE 300, 500 for Schlumberger's ECLIPSE 300 Thermal, 700 for Schlumberger's INTERSECT simulator, 800 for Schlumberger's FrontSim simulator, or a negative value for other simulators. OPM Flow uses a value of 100.			100
	RESTART	Contains the root file name of the case which this run was restarted from, if this was a restart run, other wise blank.			Required
1-1	Format	RESTART	9	CHAR	
2-1	Data	Maximum of 72 characters divided into eight character words. For example, if the data file is restarted from NOR01-PRED01-OPM2004-R00, then this data will be of the form: `'NOR01-PR' 'ED01-OPM' '2010-R00' '  ' '  ' '  ' '  ' '  '`			
	DIMENS	This keyword contains the number of vectors and objects written to the SUMMARY Data file at each time step.			Required
1-1	Format	DIMENS	6	REAL	
2-1	Data	NLIST is the number of values stored at each time step on the SUMMARY Data File.			
2-2	Data	The number of grid blocks in the x-direction (NX).			
2-3	Data	The number of grid blocks in the y-direction (NY).			
2-4	Data	The number of grid blocks in the z-direction (NZ).			
2-5	Data	Not Used.			0
2-6	Data	Report time step of the case which this run was restated from, if this was a restart run.			0
	KEYWORDS	A list of all the summary variable mnemonics written to the SUMMARY Data file at each time step.			Required
1-1	Format	KEYWORDS	NLIST	CHAR	
2-1	Data	Each mnemonic string should be enclosed in quotes.			

No.	Keyword Name	SUMMARY Index File Keywords (Global) Table F.42			Status Or Value
		Keyword	No. of Entries	Data Type	
	WGNAMES	A list of all well and group objects associated with summary variable mnemonics			Required
1-1	Format	WGNAMES	NLIST	CHAR	
2-1	Data	Each well and group name should be enclosed in quotes			
	NUMS	Integer grid cell or region number associated with the summary variable mnemonics.			Required
1-1	Format	NUMS	NLIST	INTE	
2-1	Data	Integer values.			
	LENGTHS	Horizontal well length.			Optional
1-1	Format	LENGTHS	NLIST	REAL	
2-1	Data	The distance from the bottom-hole reference depth to the completion depth for the associated summary variable mnemonics for horizontal wells			
	LENUNITS	Horizontal well length units.			Optional
1-1	Format	LENUNITS	1	CHAR	
2-1	Data	Horizontal well length units.			
	MEASRMNT	Measurements associated with each summary variable mnemonic.			Optional
1-1	Format	MEASRMNT	NBLOCK x NLIST	CHAR	
2-1	Data	NBLOCK is equal to the number of elements provided by this keyword divided by NLIST.			
	UNITS	Units			Required
1-1	Format	UNITS	NLIST	CHAR	
2-1	Data	Units for the associated summary variable mnemonics.			
	STARTDAT	Three integer values representing the date and four integers representing the time of the data being written out.			Required
1-1	Format	STARTDAT	3	INTE	
2-1	Data	Calendar day and should range between one and 31.			DAY
2-2	Data	Calendar month should range between one and 12.			MONTH
2-3	Data	Calendar year in four digits, that is 2020.			YEAR
2-4	Data	Hour.			HOUR
2-5	Data	Minutes.			MINUTE
2-6	Data	Seconds in micro seconds, 0-59:999:999			SECOND

No.	Keyword Name	SUMMARY Index File Keywords (Global) Table F.42			Status Or Value
		Keyword	No. of Entries	Data Type	
Notes: 1) Keywords up to and including the NUMS keyword are order dependent, after the NUMS keyword the keywords may be in any order. 1) Rows shaded in gray indicate the start of a keyword. 2) Cells colored orange in the No. column indicate that this item is not supported by OPM Flow and those colored red are either "Not Used" or "Undefined". 3) For formatted output all character variables, including the keyword, should be enclosed in single quotes, for example the INTEHEAD and DIMENS keywords should be written out as 'INTEHEAD ' and 'DIMENS '.					

Table F.42: SUMMARY Index File Keywords (Global)

In addition, Table F.43 summarizes the SUMMARY Index File keywords associated with Local Grid Refinements.

No.	Keyword Name	SUMMARY Index File Keywords (LGR) Table F.43			Status Or Value
		Keyword	No. of Entries	Data Type	
	LGRS	Local Grid Refinement names associated summary variable mnemonic for LGR grids.			Optional
1-1	Format	LGRS	NLIST	CHAR	
2-1	Data	Name of the LGRs.			
	NUMLX	This keyword defines the LGR grid index in the x direction (I direction) for the associated summary variable mnemonic for LGR grids.			Optional
1-1	Format	NUMLX	NLIST	INTE	
2-1	Data	LGR I location of local grid block data or the connection data.			
	NUMLY	This keyword defines the LGR grid index in the y direction (J direction) for the associated summary variable mnemonic for LGR grids.			Optional
1-1	Format	NUMLY	NLIST	INTE	
2-1	Data	LGR J location of local grid block data or the connection data.			
	NUMLZ	This keyword defines the LGR grid index in the z direction (K direction) for the associated summary variable mnemonic for LGR grids.			Optional
1-1	Format	NUMLX	NLIST	INTE	
2-1	Data	LGR K location of local grid block data or the connection data.			
	LGRNAMES	The names of all the LGRs in this case with the number of of LGRs in the case given by NLGR.			Optional
1-1	Format	LGRNAMES	NLGR	CHAR	
2-1	Data	List of LGR names.			
	LGRVEC	Integer list of the number of summary variable mnemonics for each LGR.			Optional

No.	Keyword Name	SUMMARY Index File Keywords (LGR) Table F.43			Status Or Value
		Keyword	No. of Entries	Data Type	
1-1	Format	LGRVEC	NLGR	INTE	
2-1	Data	Number of summary variable mnemonics for each LGR.			
	LGRTIMES	A list of the total number of local time steps for each LGR.			
1-1	Format	LGRTIMES	NLGR	INTE	
2-1	Data	Number of local time steps for each LGR.			

Notes:

1) The LGR keywords may be in any order but must come after the NUMS keyword in Table F.42.
2) Rows shaded in gray indicate the start of a keyword.
3) Cells colored orange in the No. column indicate that this item is not supported by OPM Flow
4) For formatted output all character variables, including the keyword, should be enclosed in single quotes, for example the LGRTIMES keyword should be written out as 'LGRTIMES'.

Table F.43: SUMMARY Index File Keywords (LGR)

Finally, Table F.44 summarizes the SUMMARY Index File keywords specific to the commercial simulator and are therefore not described in any detail.

No.	Keyword Name	SUMMARY Index File Keywords (Commercial Simulator) Table F.44	Status Or Value
	RUNTIMEI	Run time monitoring – not supported.	
	RUNTIMED	Run time monitoring – not supported.	
	STEPRESN	Run time monitoring – not supported.	
	XCOORD	OFM Data – not supported.	
	YCOORD	OFM – not supported.	
	TIMESTMP	Not supported.	

Notes:

1) Rows shaded in gray indicate the start of a keyword.
2) Cells colored orange in the No. column indicate that this item is not supported by OPM Flow.
3) For formatted output all character variables, including the keyword should be enclosed in single quotes.

Table F.44: SUMMARY Index File Keywords (Commercial Simulator)

Example: SUMMARY Index File Keywords

The following example shows a typical formatted SUMMARY Index file set of keywords.

```
 'INTEHEAD'           2 'INTE'
           1         100
 'RESTART '           9 'CHAR'
 '        ' '        ' '        ' '        ' '        ' '        '
 '        ' '        '
 'DIMENS  '           6 'INTE'
         164          10          10          10           0           0
 'KEYWORDS'         164 'CHAR'
 'TIME    ' 'YEARS   ' 'FGIP    ' 'FGIPG   ' 'FGIPL   ' 'FGIR    ' 'FGIT    '
 'FGOR    ' 'FGPR    ' 'FGPT    ' 'FOIP    ' 'FOIPG   ' 'FOIPL   ' 'FOIR    '
 'FOIT    ' 'FOPR    ' 'FOPT    ' 'FPR     ' 'FVIR    ' 'FVIT    ' 'FVPR    '
 'FVPT    ' 'FWCT    ' 'FWIP    ' 'FWIR    ' 'FWIT    ' 'FWPR    ' 'FWPT    '
 'GGIR    ' 'GGIT    ' 'GGOR    ' 'GGPR    ' 'GGPT    ' 'GOIR    ' 'GOIT    '
 'GOPR    ' 'GOPT    ' 'GVIR    ' 'GVIT    ' 'GVPR    ' 'GVPT    ' 'GWCT    '
 'GWIR    ' 'GWIT    ' 'GWPR    ' 'GWPT    ' 'SPR     ' 'SPR     ' 'SPR     '
 'SPR     ' 'SPR     ' 'SPR     ' 'SPR     ' 'SPR     ' 'SPR     ' 'SPR     '
  ……..……..
 'WWCT    ' 'WWCT    ' 'WWIR    ' 'WWIR    ' 'WWIT    ' 'WWIT    ' 'WWPR    '

 'WWPR    ' 'WWPT    ' 'WWPT    '
 'WGNAMES '         164 'CHAR'
 ':+:+:+:+' ':+:+:+:+' ':+:+:+:+' ':+:+:+:+' ':+:+:+:+' ':+:+:+:+' ':+:+:+:+'
 ':+:+:+:+' ':+:+:+:+' ':+:+:+:+' ':+:+:+:+' ':+:+:+:+' ':+:+:+:+' ':+:+:+:+'
 ':+:+:+:+' ':+:+:+:+' ':+:+:+:+' ':+:+:+:+' ':+:+:+:+' ':+:+:+:+' ':+:+:+:+'
 ':+:+:+:+' ':+:+:+:+' ':+:+:+:+' ':+:+:+:+' ':+:+:+:+' ':+:+:+:+' ':+:+:+:+'
 'PLATFORM' 'PLATFORM' 'PLATFORM' 'PLATFORM' 'PLATFORM' 'PLATFORM' 'PLATFORM'
 'PLATFORM' 'PLATFORM' 'PLATFORM' 'PLATFORM' 'PLATFORM' 'PLATFORM' 'PLATFORM'
 'PLATFORM' 'PLATFORM' 'PLATFORM' 'PLATFORM' 'OP01    ' 'OP01    ' 'OP01    '
 'OP01    ' 'OP01    ' 'OP01    ' 'OP01    ' 'OP01    ' 'OP01    ' 'OP01    '
 'OP01    ' 'OP01    ' 'OP01    ' 'OP01    ' 'OP01    ' 'OP01    ' 'OP01    '
 'OP01    ' 'OP01    ' 'OP01    ' 'OP01    ' 'OP01    ' 'OP01    ' 'OP01    '
 'OP01    ' 'OP01    ' 'OP01    ' 'OP01    ' 'OP01    ' 'OP01    ' 'OP01    '
 'OP01    ' 'OP01    ' 'OP01    ' 'OP01    ' 'OP01    ' 'OP01    ' 'OP01    '
……..……..

 'WI01    ' 'OP01    ' 'WI01    '
 'NUMS    '         164 'INTE'
           0           0           0           0           0           0
           0           0           0           0           0           0
           0           0           0           0           0           0
           0           0           0           0           0           0
           0           0           0           0           0           0
           0           0           0           0           0           0
           0           0           0           0           0           0
           0           0           0           0           1           2
           3           4           5           6           7           8
           9          10          11          12          13          14
……..……..
          0           0
 'UNITS   '         164 'CHAR'
 'DAYS    ' 'YEARS   ' 'SM3     ' 'SM3     ' 'SM3     ' 'SM3/DAY ' 'SM3     '
 'SM3/SM3 ' 'SM3/DAY ' 'SM3     ' 'SM3     ' 'SM3     ' 'SM3     ' 'SM3/DAY '
 'SM3     ' 'SM3/DAY ' 'SM3     ' 'BARSA   ' 'RM3/DAY ' 'RM3     ' 'RM3/DAY '
 'RM3     ' 'SM3/SM3 ' 'SM3     ' 'SM3/DAY ' 'SM3     ' 'SM3/DAY ' 'SM3     '
……..……..
 'SM3/DAY ' 'SM3     ' 'SM3     '
 'STARTDAT'           6 'INTE'
           1           1        2020           0           0           0
```

End of Example

F.9.3 SUMMARY DATA FILE KEYWORDS

The structure for this series of keywords is relatively simple compared to the SUMMARY Index file, in that it only consists of four keyword types that are written out at the requested time steps, as outlined in Table F.45.

No.	Keyword Name	SUMMARY Data File Keywords Table F.45			Status Or Value
		Keyword	No. of Entries	Data Type	
	SEQNUM	This keyword contains a sequence data value corresponding to an encoded integer representing the time the file was created. Although required the data is not used. Secondly, this keyword is written at each report step only and not for each inter leaving mini step, that is the file format is of the following format: SEQNUM — Report step n MINISTEP PARAMS MINISTEP PARAMS MINISTEP PARAMS INISTEP PARAMS SEQNUM — Report step n + 1 MINISTEP PARAMS MINISTEP PARAMS			Required
1-1	Format	SEQNUM	1	INTE	
2-1	Data	The value entered is not used by either OPM Flow or OPM ResInsight			0
	MINSTEP	The time step number starting from zero and incremented by one for each subsequent time step.			Required
1-1	Format	MINISTEP	1	INTE	
2-1	Data	Time step.			
	PARAMS	This keyword contains the actual data for all the vectors and objects specified in the index file.			Required
1-1	Format	PARAMS	NLIST	REAL	
2-1	Data	There should be NLIST values corresponding to the variable list declared on the corresponding SUMMARY Index file.			

Notes:

1) Rows shaded in gray indicate the start of a keyword.

Table F.45: SUMMARY Data File Keywords

For LGRs, the global cell data is written to the standard SUMMARY Data file containing an additional keyword that states the LGR name, as depicted in Table F.46.

No.	Keyword Name	SUMMARY Data File Keywords (LGR) Table F.46			Status Or Value
		Keyword	No. of Entries	Data Type	
	SEQHDR	As per Table F.45			Required
	LGRNAME	Name of Local Grid Refinement.			Required
	Format	LGRNAME	1	CHAR	
	Data	LGR name enclosed in single quotes.			
	MINSTEP	As per Table F.45.			Required
	PARAMS	As per Table F.45.			Required

Notes:
1) Rows shaded in gray indicate the start of a keyword.

Table F.46: SUMMARY Data File Keywords (LGR)

The data for the LGR time steps, as opposed to the global time step, are written out to a separate SUMMARY Data File, with the suffix of LGR or FLG for fixed format output. In this case, the MINSTEP keyword states the "local" time step and the keyword is repeated for each local grid in the model.

Example: SUMMARY Data Data File keyword

The following example shows a typical formatted SUMMARY Data File keyword data set for one complete report time step and part of a second report time step.

```
'SEQHDR  '           1 'INTE'
         1
'MINISTEP'           1 'INTE'
         0
'PARAMS  '         164 'REAL'
  0.50000000E+00   0.13689254E-02   0.18381236E+11   0.51459348E+10
  0.13235301E+11   0.00000000E+00   0.00000000E+00   0.37649313E+03
  0.95391960E+07   0.47695980E+07   0.10947282E+09   0.30000488E+06
  0.10917282E+09   0.00000000E+00   0.00000000E+00   0.25336973E+05
  0.12668486E+05   0.27468176E+03   0.15571577E+05   0.77857886E+04
  0.63867320E+05   0.31933660E+05   0.12643164E-02   0.60474832E+08
  0.15000000E+05   0.75000000E+04   0.32074501E+02   0.16037251E+02
  0.00000000E+00   0.00000000E+00   0.37649313E+03   0.95391960E+07
  0.47695980E+07   0.00000000E+00   0.00000000E+00   0.25336973E+05
  0.12668486E+05   0.15571577E+05   0.77857886E+04   0.63867320E+05
  0.31933660E+05   0.12643164E-02   0.15000000E+05   0.75000000E+04
  0.32074501E+02   0.16037251E+02   0.26000000E+03   0.26172531E+03
  0.26362421E+03   0.26596732E+03   0.26747589E+03   0.26896045E+03
  ..............................................
'SEQHDR  '           1 'INTE'
         2
'MINISTEP'           1 'INTE'
         1
'PARAMS  '         164 'REAL'
  0.10000000E+01   0.27378509E-02   0.18377710E+11   0.51453399E+10
  0.13232370E+11   0.00000000E+00   0.00000000E+00   0.32637939E+03
  ..............................................
```

End of Example

F.10 SAVE FILES - INITIALIZATION AND SOLUTION DATA

This file format is currently not supported by OPM Flow.

End of Document