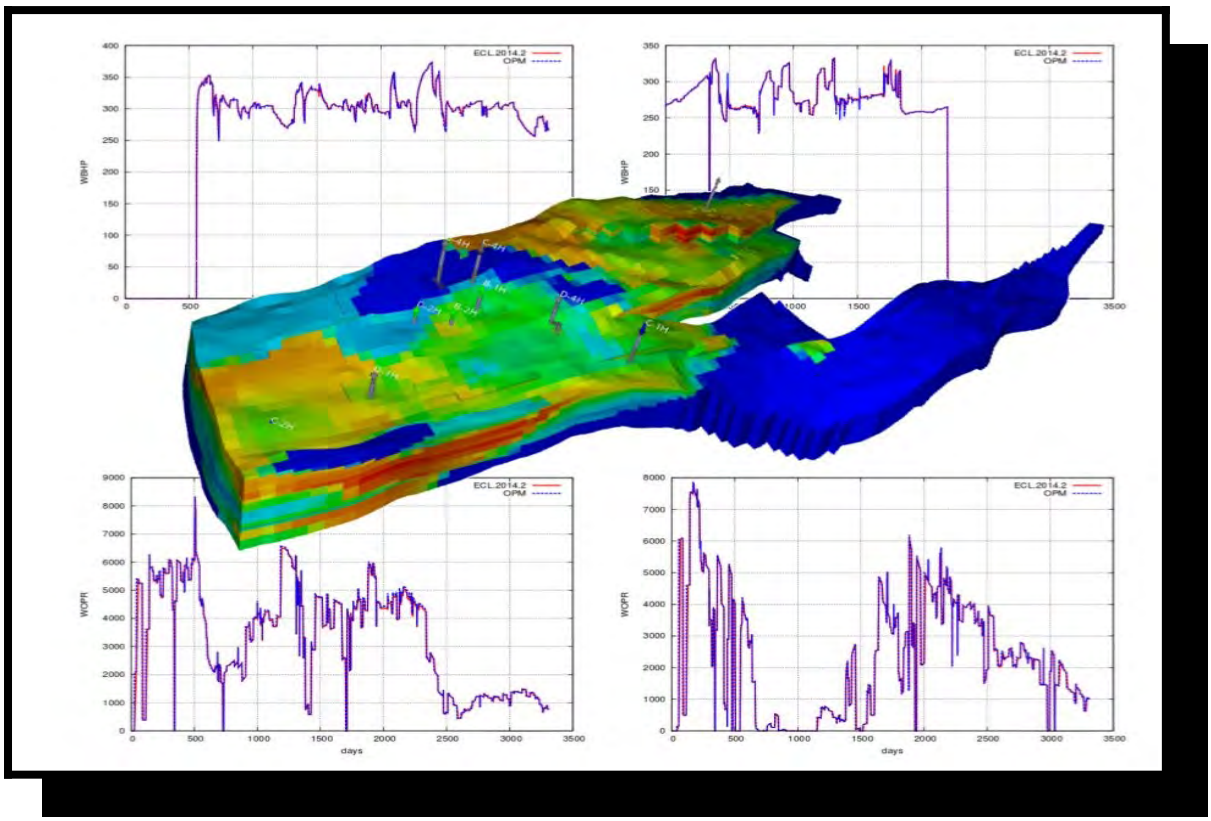
OPEN POROUS MEDIA

Flow Documentation Manual



OPM FLOW VERSION: 2019-10

MANUAL REVISION: Rev-1

OPEN POROUS MEDIA
Flow Documentation Manual
(2019-10)

<table>
<tr><th colspan="5">Document Control</th></tr>
<tr><td>Company</td><td colspan="4">Equinor ASA</td></tr>
<tr><td>Confidentiality, copyright and reproduction</td><td colspan="4">This document is Copyright 2019 by the OPEN POROUS MEDIA team.
<u>Documentation Contributors:</u>
David Baxendale | Kai Bao | Markus Blatt | Joakim Hove
Andreas Lauser | Arne Morten Kvarving | Atgeirr Rasmussen | Alf Birger Rustad
Tor Harald Sandve | Bård Skaflestad | Torbjørn Skille

One may distribute or modify the document under the terms of the Creative Commons Attribution-ShareALike 4.0 International License, version 4.0 or later (http://creativecommons.org/licenses/by/4.0/).
All trademarks within this guide belong to their legitimate owners.</td></tr>
<tr><td>Document Type</td><td colspan="4">OPEN POROUS MEDIA</td></tr>
<tr><td>Document Title</td><td colspan="4">Flow Documentation Manual (2019-10)</td></tr>
<tr><td>Document Ref.</td><td colspan="4">2019-10-RPT</td></tr>
<tr><td rowspan="10">Document Revision</td><td>Date</td><td colspan="2">Version and Revision</td><td>Status</td></tr>
<tr><td>December 5, 2019</td><td colspan="2">2019-10 Rev-1</td><td>Final</td></tr>
<tr><td>November 6, 2019</td><td colspan="2">2019-10 Rev-0</td><td>Final Draft</td></tr>
<tr><td>June 20, 2019</td><td colspan="2">2019-04 Rev-0</td><td>Final</td></tr>
<tr><td>June 7, 2019</td><td colspan="2">2019-04 Rev-A</td><td>Review Version</td></tr>
<tr><td>December 31, 2018</td><td colspan="2">2018-10 Rev-2</td><td>Fix Broken Links etc.</td></tr>
<tr><td>November 6, 2018</td><td colspan="2">2018-10 Rev-1</td><td>Final (DRSDTR & DRVDTR)</td></tr>
<tr><td>November 5, 2018</td><td colspan="2">2018-10 Rev-0</td><td>Final</td></tr>
<tr><td>October 4, 2017</td><td colspan="2">2017-04 Rev-0</td><td>Final</td></tr>
<tr><td rowspan="3">Author and Editor</td><td colspan="4">Equinox International Petroleum Consultants Pte. Ltd.
51 Goldhill Plaza, #07-10/11, Singapore 308900</td></tr>
<tr><td colspan="4">David Baxendale</td></tr>
<tr><td colspan="4">david.baxendale@eipc.co</td></tr>
<tr><td>Document Name</td><td colspan="4">D:\Projects\Equinor\Miscellaneous\M70 Technical Reports and Notes\OPM Flow Documentation 2019-10 Rev-1.odt</td></tr>
</table>

TABLE OF CONTENTS

INDEX OF TABLES

INDEX OF FIGURES

1 INTRODUCTION

OPM Flow is a fully-implicit, black-oil simulator capable of running industry-standard simulation models. The simulator is implemented using automatic differentiation to enable rapid development of new fluid models. The program is in active development with new features added in each bi-annual release. Currently the program has the following functionality:

Model Formulation:

- Back-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.

EOR Options:

- The Polymer Model in OPM Flow 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 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 Polymer model has now been incorporated into the main OPM Flow simulator and is no longer a separate simulator.
- The Solvent Model in OPM Flow extra component is again based on extending the black-oil oil formulation with a fourth component 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.
- 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.

Description of Geology:

- Rectilinear and fully-unstructured grid.
- Corner-point grids from Eclipse input, including fault and region multipliers, minpv/pinch, etc.

Well and Group Controls:

- Bottom-hole pressure and surface/reservoir rate.
- Group controls.
- Shut/stop/open individual completions.
- History-matching wells.
- Well lists.
- Action conditions and command processing.

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 Eclipse format.
- Logging to terminal and print file.

Simulation Technology:

- Fully-implicit in time.
- Two-point flux approximation in space with upstream-mobility weighting.
- Flexible assembly through the use of automatic differentiation.
- Block-structured linear solver with ILU0 pre-conditioner.
- Adaptive step-size controls.

2 INSTALLING AND RUNNING FLOW

2.1 INSTALLING 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 16.04 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 16.04 (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 Flow) and their dependencies,

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

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

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).

2) If an old versions of prerequisite libraries have already been installed (for example from installing a previous release of OPM) it may 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
```

2.1.3 USING A DOCKER CONTAINER

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

2.1.4 INSTALLING FROM SOURCE

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

2.2 RUNNING OPM FLOW 2019-10 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.

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

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

will send the output files to the foo directory. Alternatively one can use a parameter file to define the input deck and the various parameters via:

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

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:

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

will start OPM Flow on four nodes etc.

A list of command line options for this release is outlined in Table 2.1 and past releases are documented in section *APPENDIX C: RUNNING PREVIOUS RELEASES OF OPM FLOW*.

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.

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

OPM Flow release 2018-10 and beyond have switched to the eWoms/ebos[1] 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 depreciated but are still documented in section *APPENDIX C: RUNNING PREVIOUS RELEASES OF OPM FLOW* for backward compatibility with previous releases of the simulator. For the 2019-10 release a combination of OPM Flow and eWoms/ebos command line parameters are available as tabulated in Table 2.1 on this and subsequent pages. It is anticipated that future releases of OPM Flow will fully migrate to the eWoms/ebos command line parameter set.

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.

A complete list of command line options and their function is given in the table below.

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

[1] *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 2019-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
3	--cpr-ell-solvetype	A positive integer that defines the solver type of the elliptic pressure solver: 1) 0: bicgstab, 2) 1: cg, 3) 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: 1) 0: no output 2) 1: output summary of inner linear solver 3) 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 2019-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
17	--edge-weights-method	A defined positive integer value that defines the edge-weighing strategy: 1) 0: for uniform, 2) 1: for trans, 0r 3) 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 *5.2.79 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

OPM Flow 2019-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
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: 1) 0: no extra output 2) 1: output per solution iteration 3) >1: 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 pre-conditioner.	0
39	--ilu-redblack	A Boolean value set to true or false that instructs OPM Flow to use red-black partitioning for the ILU pre-conditioner. (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 pre-conditioner	0.9

OPM Flow 2019-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
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 pre-conditioner 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-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 pre-conditioner 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 *.DEBUG 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 vale 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: 1) 0: No output to the files. 2) 1: Output *.DBG file 3) 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: 1) 0: No output to the files. 2) 1: Output *.DBG file 3) 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[2]. 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

[2] *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 Newtonian 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.	1.0×10^{6}
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
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

OPM Flow 2019-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
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 pre-conditioner	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 pre-conditioner	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[3]		
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
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

[3] *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**
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
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

OPM Flow 2019-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
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
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

OPM Flow 2019-10 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.
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 pre-conditioner 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 2.1: OPM Flow 2019-04 Command Line Options

As per all UNIX and LINUX based system 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 CASNAME.DATA
```

or:

```
flow --EnableDryRun=false CASNAME.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 CASNAME.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.

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 Newtonian Solver iterations to 1 and Max to 15
#
flow-newton-min-iterations=1
flow-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 describes how to run the OPM Flow simulator using the Graphical User Interface ("GUI") program called OPMRUN. The program is written in Python 3 and has been tested under Unbuntu-Mate 18.04 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 2.1 shows the initial display.

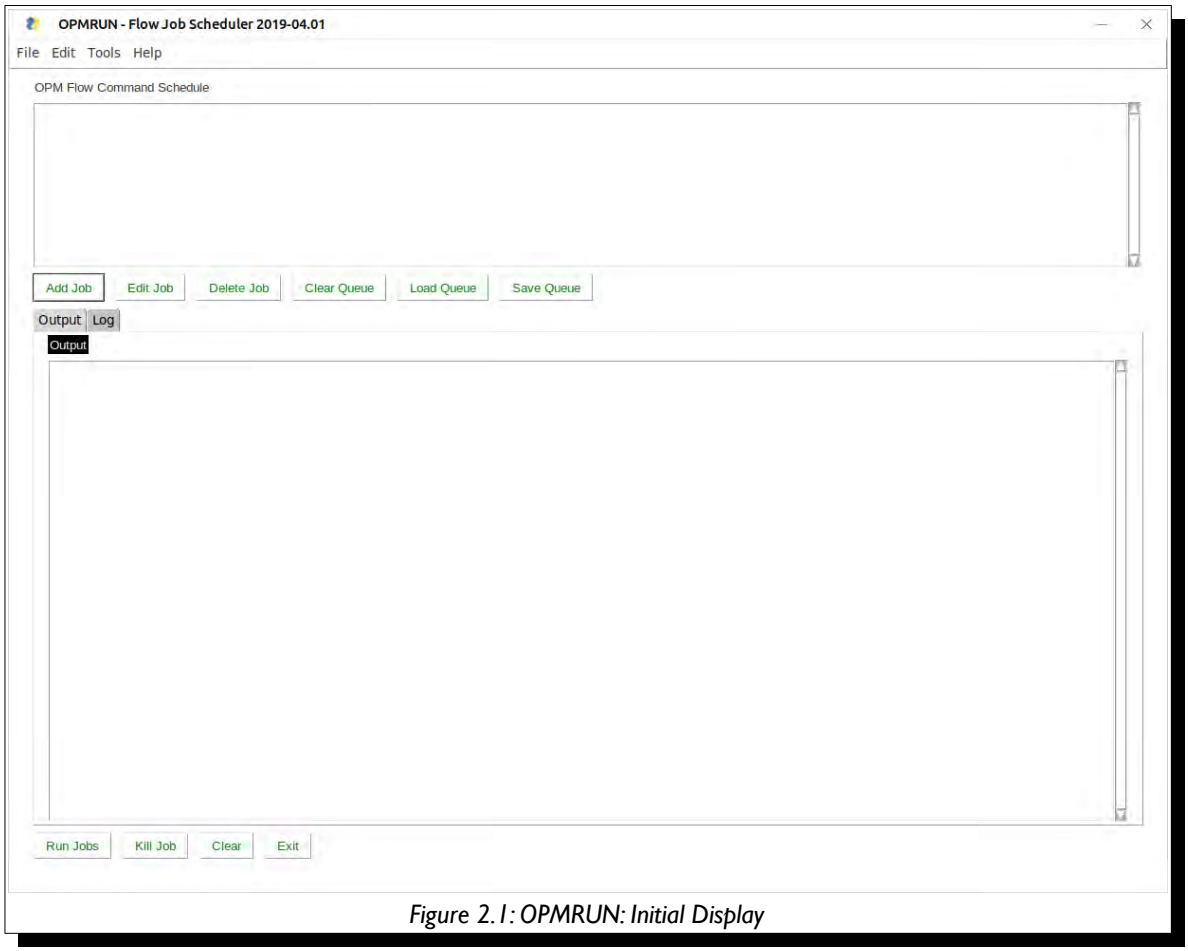


Figure 2.1: OPMRUN: Initial Display

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 2.2 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.

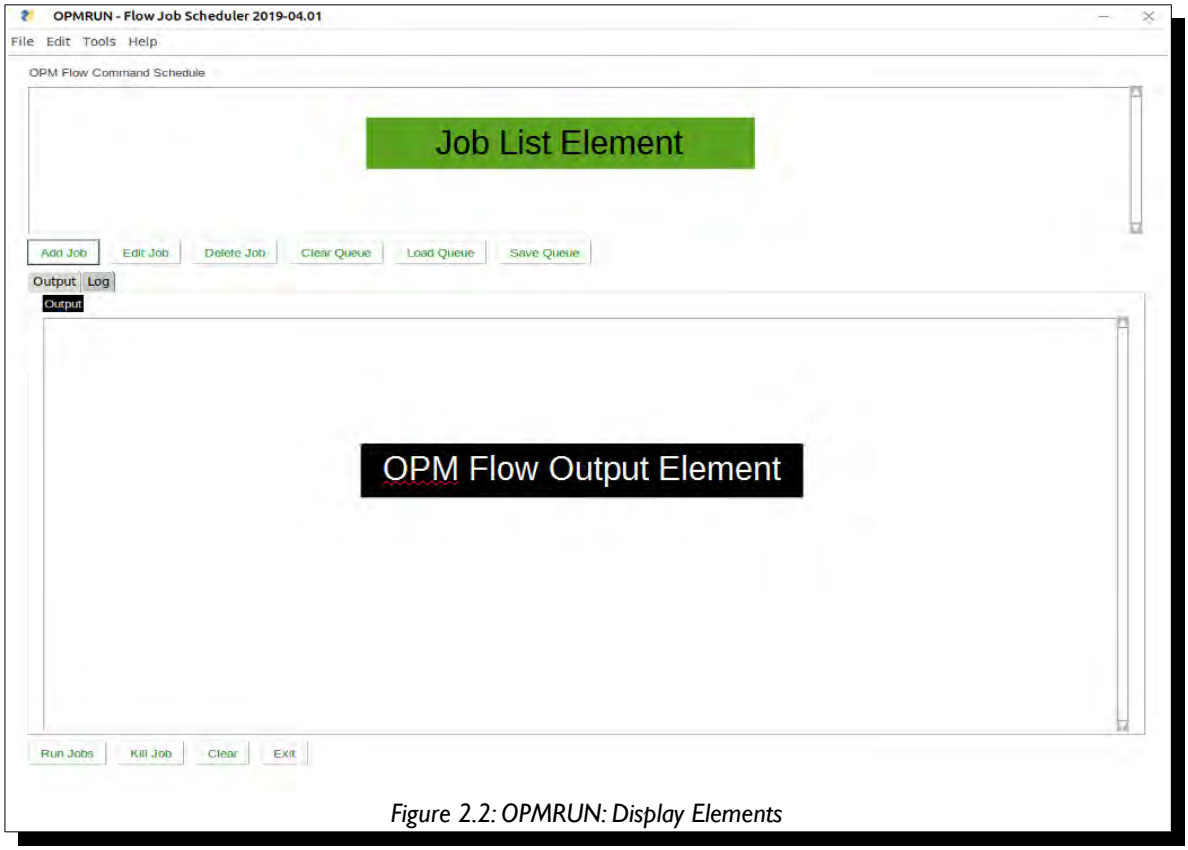


Figure 2.2: OPMRUN: Display Elements

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:

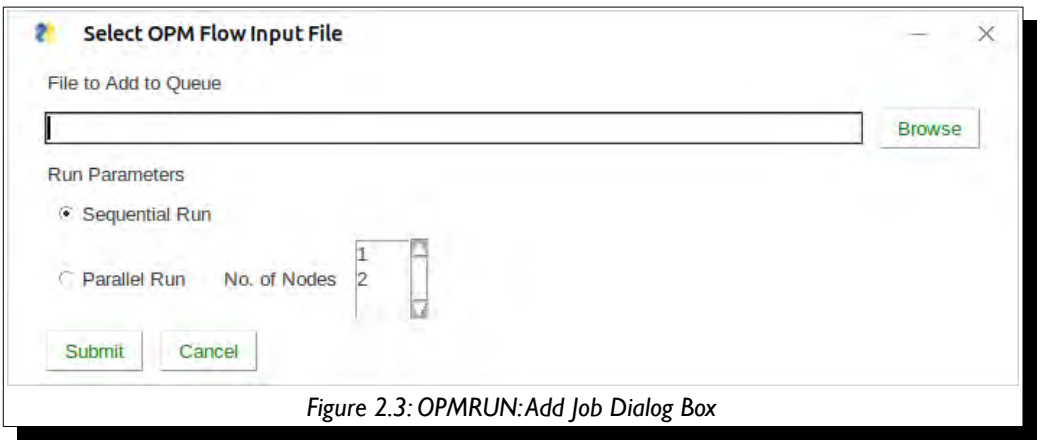


Figure 2.3: OPMRUN: Add Job Dialog Box

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.

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 top element as illustrated in Figure 2.4.

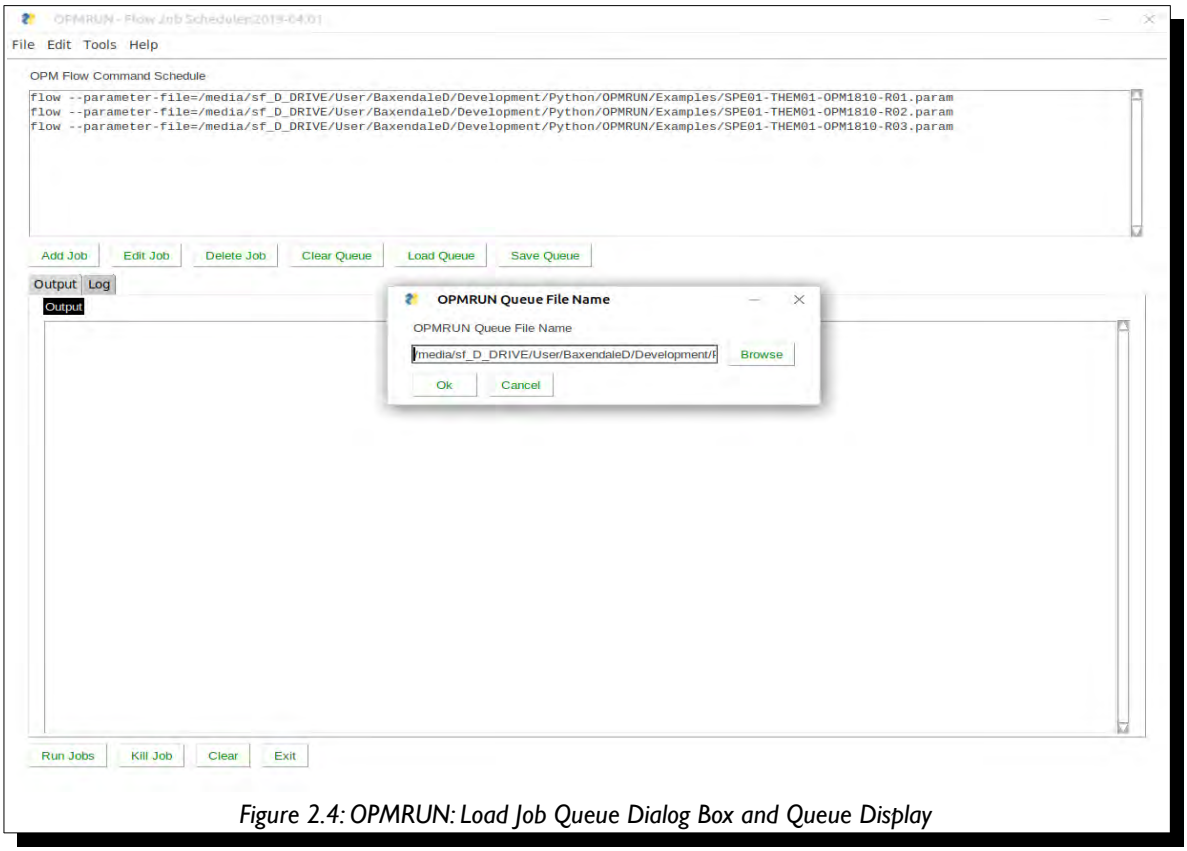


Figure 2.4: OPMRUN: Load Job Queue Dialog Box and Queue Display

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 2.5). 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).

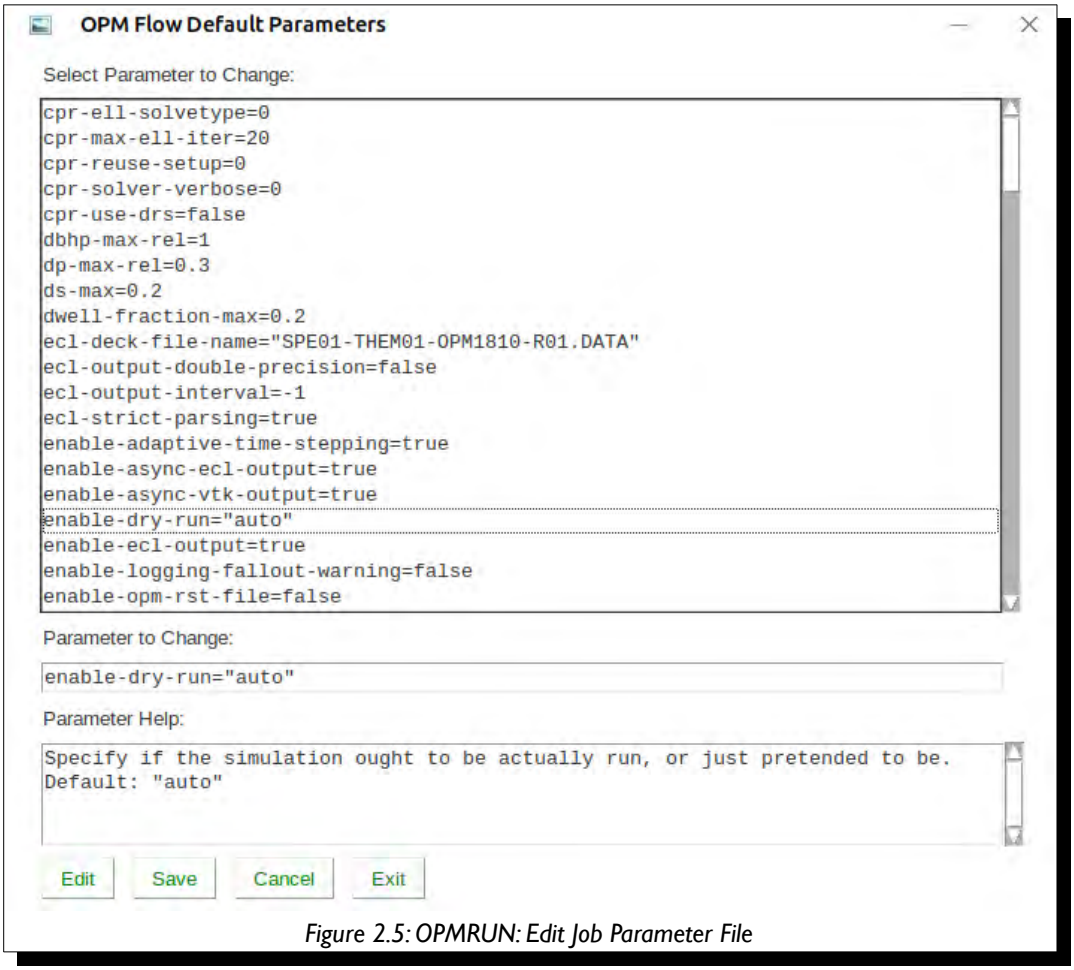


Figure 2.5: OPMRUN: Edit Job Parameter File

Selecting the Run Jobs button displays the Select Run Option dialog box shown in Figure 2.6.

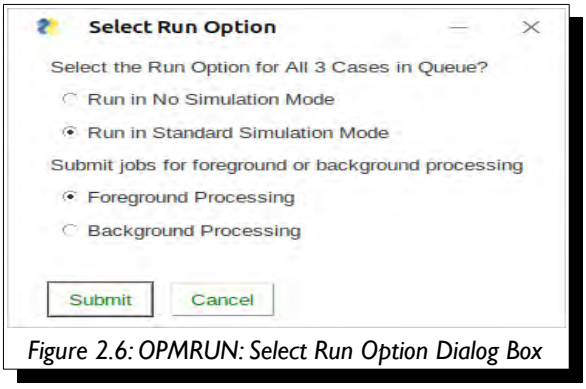


Figure 2.6: OPMRUN: Select Run Option Dialog Box

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 *5.2.79 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 2.7. The terminal output is also directed to a *.LOG file as well, similar to what the commercial simulator does.

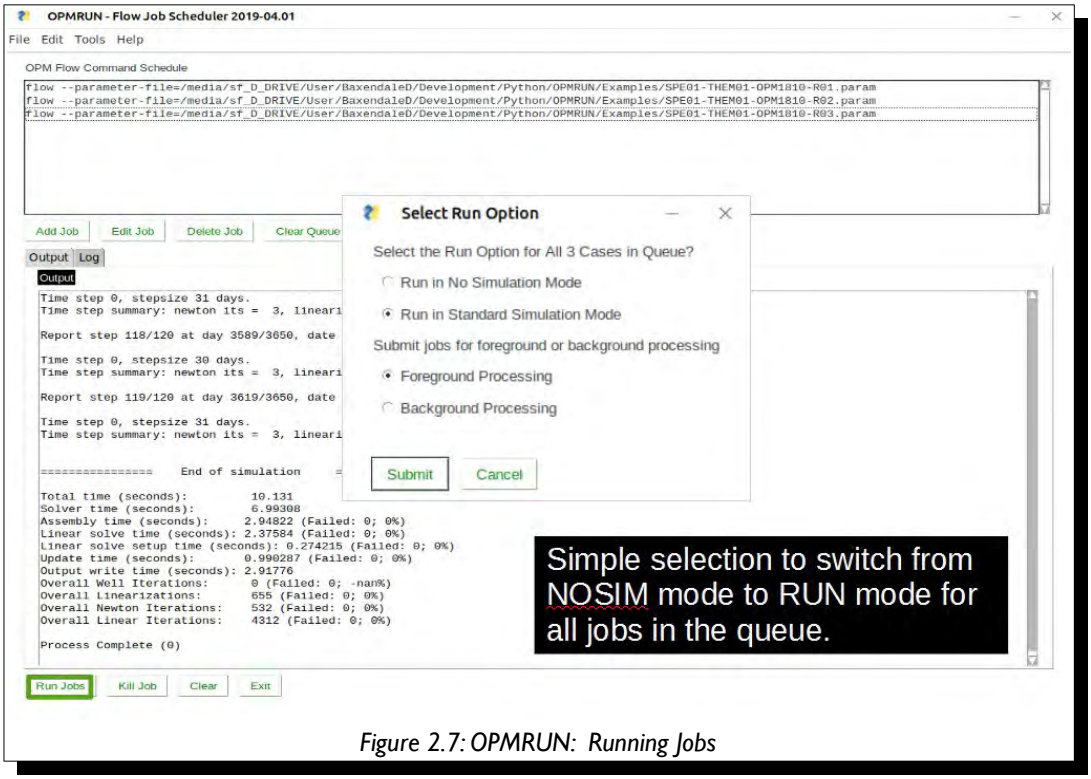


Figure 2.7: OPMRUN: Running Jobs

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.

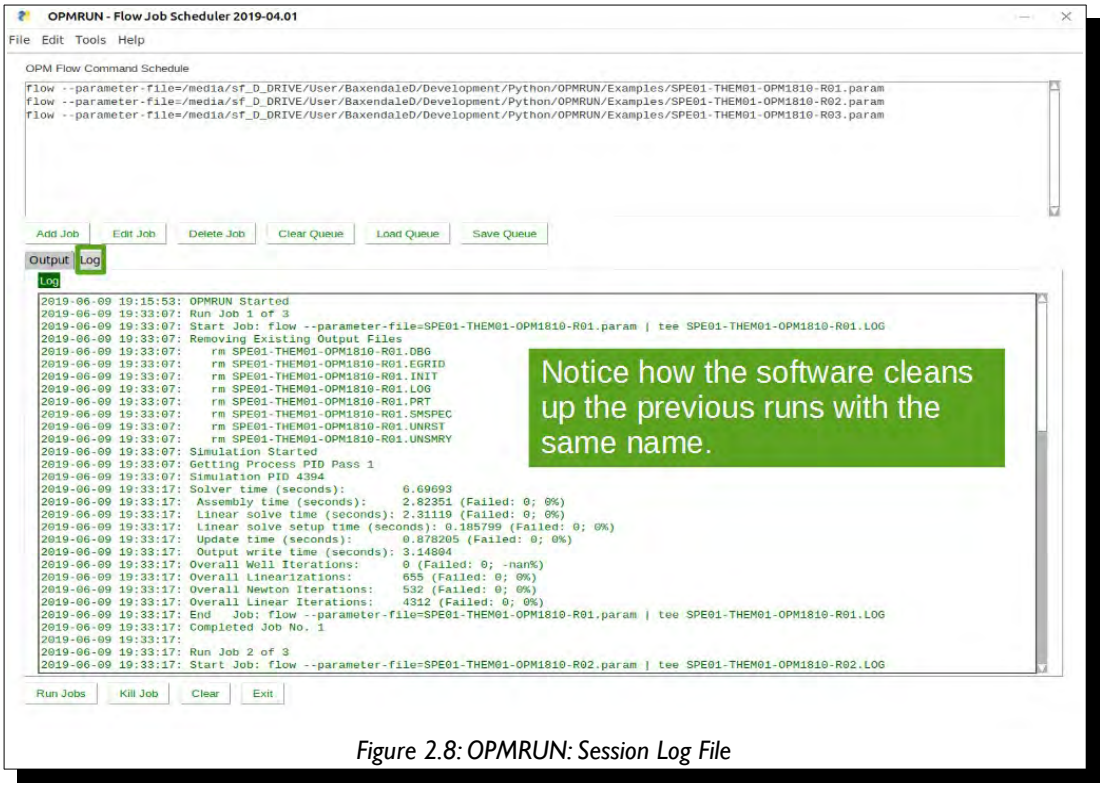


Figure 2.8: OPMRUN: Session Log File

Finally, an example *.param file generated by OPMRUN for the 2019-10 release is shown below for reference:

```
#
# OPMRUN Parameter File
#
# File Name   : "/media/Linux/OPM/Norne/NOR01-BASE01-OPM1910-R01.param"
# Created By  : OPMUSER
# Date Created: 2019-11-05 20:39:09
#
continue-on-convergence-error=false
cpr-ell-solvetype=0
cpr-max-ell-iter=20
cpr-reuse-setup=0
cpr-solver-verbose=0
cpr-use-drs=false
dbhp-max-rel=1
dp-max-rel=0.3
ds-max=0.2
dwell-fraction-max=0.2
ecl-deck-file-name="NOR01-BASE01-OPM1910-R01.DATA"
ecl-enable-drift-compensation=false
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-logging-fallout-warning=false
enable-opm-rst-file=false
enable-storage-cache=true
enable-terminal-output=true
enable-tracer-model=true
enable-tuning=false
enable-vtk-output=false
enable-well-operability-check=true
enable-write-all-solutions=false
flow-linear-solver-verbosity=0
flow-newton-max-iterations=20
flow-newton-min-iterations=1
force-disable-fluid-in-place-output=false
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-configuration-json-file="none"
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
matrix-add-well-contributions=false
max-inner-iter-ms-wells=100
max-pressure-change-ms-wells=1e+06
max-residual-allowed=1e+07
max-single-precision-days=20
max-strict-iter=8
max-welleq-iter=30
milu-variant="ILU"
```

```
newton-max-relax=0.5
newton-relaxation-type="dampen"
output-dir=""
output-interval=1
output-mode="all"
parameter-file=""
pri-var-oscilation-threshold=1e-05
print-parameters=2
print-properties=2
scale-linear-system=false
solve-welleq-initially=true
solver-growth-factor=2
solver-max-growth=3
solver-max-restarts=10
solver-max-time-step-in-days=365
solver-restart-factor=0.33
solver-verbosity=1
system-strategy="none"
threads-per-process=-1
time-step-after-event-in-days=-1
time-step-control="pid"
time-step-control-decay-rate=0.75
time-step-control-file-name="timesteps"
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=1e+09
tolerance-mb=1e-06
tolerance-pressure-ms-wells=1000
tolerance-well-control=1e-07
tolerance-wells=0.0001
update-equations-scaling=false
use-amg=false
use-cpr=false
use-gmres=false
use-inner-iterations-ms-wells=true
use-multisegment-well=true
use-update-stabilization=true
vtk-write-average-molar-masses=false
vtk-write-densities=true
vtk-write-dof-index=false
vtk-write-ecl-tracer-concentration=false
vtk-write-extrusion-factor=false
vtk-write-filter-velocities=false
vtk-write-fugacities=false
vtk-write-fugacity-coeffs=false
vtk-write-gas-dissolution-factor=false
vtk-write-gas-formation-volume-factor=false
vtk-write-gas-saturation-pressure=false
vtk-write-intrinsic-permeabilities=false
vtk-write-mass-fractions=false
vtk-write-mobilities=false
vtk-write-molarities=false
vtk-write-mole-fractions=true
vtk-write-oil-formation-volume-factor=false
vtk-write-oil-saturation-pressure=false
vtk-write-oil-vaporization-factor=false
vtk-write-porosity=true
vtk-write-potential-gradients=false
vtk-write-pressures=true
vtk-write-primary-vars=false
vtk-write-primary-vars-meaning=false
```

```
vtk-write-process-rank=false
vtk-write-relative-permeabilities=true
vtk-write-saturated-gas-oil-vaporization-factor=false
vtk-write-saturated-oil-gas-dissolution-factor=false
vtk-write-saturation-ratios=false
vtk-write-saturations=true
vtk-write-temperature=true
vtk-write-total-mass-fractions=false
vtk-write-total-mole-fractions=false
vtk-write-viscosities=false
vtk-write-water-formation-volume-factor=false
#
# End of Parameter File
```

In the above example, all the default parameters have be used except for the enable-tracer-model option that has been set to true to use tracers.

3 KEYWORD DOCUMENTATION STRUCTURE

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 imply by 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.

3.1 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 means 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.

3.2 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 section 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.

3.3 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 three types of keyword format types used by OPM Flow for data input. The description of the three types is given in the next three section together with some examples.

3.3.1 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
/
```

3.3.2 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.

3.3.3 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 "/".			

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.

Examples

```
--
--        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 the documenting the defaults 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 "/" at the end of the line, **there is no terminator for the keyword.**

Table 3.4: ROCK Keyword Description

Examples

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.

3.3.4 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 "/" 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.

3.3.5 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 "/".					

Table 3.3: PORO Keyword Description

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

Examples

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

3.4 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[4] 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

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

[4] *ECLIPSE Industry-Reference Reservoir Simulator – Reference Manual 2015.1, Schlumberger.*

4 GLOBAL SECTION KEYWORDS

Keywords used in this section can be used in all input file sections

4.1 KEYWORD DEFINITIONS

4.1.1 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 ignored by OPM Flow and has no effect on the simulation

4.1.2 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), it is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

4.1.3 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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.

4.1.4 END – DEFINE THE END OF THE INPUT FILE

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.

4.1.5 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.

4.1.6 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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.

4.1.7 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 "/".

Table 4.1: EXTRAPMS Keyword Description

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

Examples

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

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

4.1.8 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 "/".			

Table 4.2: FORMFEED Keyword Description

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

Examples

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

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

4.1.9 INCLUDE – LOAD ANOTHER DATA FILE AT THE CURRENT POSITION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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 "/".			

Table 4.3: 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 from where the data file is located.

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

4.1.10 MESSAGE – OUTPUT USER MESSAGE

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 MESSAGE**S** keyword, that defines OPM Flows message print limits and stop limits generated by the simulator.

OPM Flow does not recognize this keyword and the keyword may cause the program to terminate, but it is documented here for completeness.

4.1.11 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 4.4.

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
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

No.	Name	Description	Default
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 are 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 "/".		

Table 4.4: 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.

4.1.12 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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.

4.1.13 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

Example

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

The examples 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.

4.1.14 SKIP – ACTIVATE SKIPPING OF ALL KEYWORDS AND INPUT DATE

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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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.

4.1.15 SKIP100 – ACTIVATE SKIPPING OF "BLACK-OIL" KEYWORDS AND INPUT DATE

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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 examples 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.

4.1.16 SKIP300 – ACTIVATE SKIPPING OF "COMPOSITIONAL" KEYWORDS AND INPUT DATE

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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 examples 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.

4.1.17 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

Example

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

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

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

The examples 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.

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 OPM Flow.

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

5.2 KEYWORD DEFINITIONS

5.2.1 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 "/".			

Table 5.1:ACTDIMS 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.

Examples

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

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

5.2.2 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 1×10^{-4} means that if the difference between two real values is less than 1×10^{-4} then the values are considered equal.			
		fraction 1×10^{-4}	fraction 1×10^{-4}	fraction 1×10^{-4}	Defined

Notes:

1) The keyword is terminated by "/".

Table 5.2: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.

Examples

```
--
--       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 1.0×10^{-4}.

5.2.3 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 ignored by OPM Flow and has no effect on the simulation

5.2.4 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 ignored by OPM Flow and has no effect on the simulation.

5.2.5 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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.

5.2.6 API – ACTIVATE API TRACKING

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

Example

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

The above example switches on the API tracking facility.

5.2.7 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	MXAQN	A positive integer value that defines the AQUNUM keyword maximum number of lines associated with this keyword.	1
2	MXNAQC	A positive integer value that defines the AQUCON keyword maximum number of lines of connection data associated with this keyword.	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 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 AQUFET and AQUCT maximum number of analytical aquifers defined by these two 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 analytic aquifers in any single aquifer list as defined by (7).	0

Notes:

1) The keyword is terminated by "/".

Table 5.3:AQUDIMS Keyword Description

Examples

```
--
--        AQF      AQF      AQF      AQF      AQF      AQF      AQF      AQF
--        MXAQN    MXNAQC   NIFTBL   NRIFTB   NANAQ    NCAMAX   MXNALI   MXAAQL
AQUDIMS
          1*       1*       1*       1*       1*       1*       1*       1*              /
```

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

5.2.8 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 ignored by OPM Flow and has no effect on the simulation.

5.2.9 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.

5.2.10 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 DEADOIL and 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.

5.2.11 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

See section *2.2 Running OPM Flow 2019-10 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.

5.2.12 BPDIMS – 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

5.2.13 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.

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 — $MgSO_4$	None

Notes:

1) There is no data required for this keyword if the standard Brine Tracking option is being activated; however, if the Multi-Component Brine Tracking option has been invoked by the ECLMC keyword, a list of SALTS just be supplied and in this case the keyword is terminated by "/".

Table 5.4: BRINE Keyword Description

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    CACL    MGC03                                                /
```

The above example activates the Multi-Component Brine model with three different water salinities.

5.2.14 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 ignored by OPM Flow and has no effect on the simulation.

5.2.15 CBMOPTS – DEFINE COAL BED METHANE OPTIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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 ignored by OPM Flow and has no effect on the simulation.

5.2.16 COAL – ACTIVATE THE COAL PHASE (CBM MODEL)

Description

The COAL keyword actives the coal phase and the Coal Bed Methane ("CBM") model for the run.

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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

Example

```
--
--       ACTIVATE THE COAL PHASE (CBM MODEL)
--
COAL
```

The above example declares that the Coal phase is active in the run and activates the CBM model option.

5.2.17 CPR – ACTIVATE CONSTRAINED PRESSURE RESIDUAL ("CPR") LINEAR SOLVER

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

Turns on the Constrained Pressure Residual ("CPR") linear solver. There is no data required for this keyword.

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

See section *2.2 Running OPM Flow 2019-10 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 linear solver for the run.

5.2.18 DEADOIL – ACTIVATE THE DEAD OIL PHASE (NO FREE OR DISSOLVED GAS)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword deactivates free and dissolved gas in the model and therefore makes the oil phase dead oil[5] in the "black-oil" formulation, and is equivalent to setting the phases present in the model to be oil 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 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.

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

Example

The following example activates the "black-oil" phases in the model.

```
--
--        ACTIVATE DEAD-OIL PHASE
--
DEADOIL
```

Alternatively one could explicitly declare the phases using the following keywords in the RUNSPEC section.

```
--
--        OIL PHASE IS PRESENT IN THE RUN
--
OIL
--
--        WATER PHASE IS PRESENT IN THE RUN
--
WATER
```

The above example switches on the oil and water phases in the model.

[5] *"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.*

5.2.19 DIFFUSE – ACTIVATE MOLECULAR DIFFUSION OPTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The DIFFUSE keyword activates molecular diffusion option.

This keyword is ignored by OPM Flow and has no effect on the simulation.

5.2.20 DIMENS – DEFINE THE DIMENSION 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 "/".			

Table 5.5: 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.

Examples

```
--
--        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.

5.2.21 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[6] 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[7], 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.

[6] *"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.*

[7] *"Dead" oil is oil that it contains no dissolved gas or a relatively thick oil or residue that has lost its volatile components.*

5.2.22 DISPDIMS – DEFINE THE MAXIMUM NUMBER OF DISPERSION TABLES

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 ignored by OPM Flow and has no effect on the simulation.

5.2.23 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[8,9,10,11 and 12].

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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.

[8] *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.*

[9] *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.*

[10] *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*

[11] *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.*

[12] *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.*

5.2.24 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[13][14][15][16] and [17].

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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.

[13] *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.*

[14] *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.*

[15] *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*

[16] *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.*

[17] *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.*

5.2.25 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 ignored by OPM Flow and has no effect on the simulation.

5.2.26 ECLMC – ACTIVATE MULTI-COMPONENT BRINE MODEL

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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    CACL    MGC03                                          /
```

The above example activates the Multi-Component Brine model with three different water salinities.

5.2.27 ENDSCALE – ACTIVATE RELATIVE PERMEABILITY END-POINT SCALING OPTION

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 tion of depth as oppose to being a grid property array. This is accomplished via the ENKRVD and ENPVTD 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 switch off and the same saturation function is used in the x, y and z directions (unless activated otherwise by the SATOPS keyword in the RUNSPEC section). In this case the SWL, SWCR, SWU, SGL, SGCR, SGU, SOWCR and SOGCR saturation grid arrays and the KRG, KROG, KROW and KRW relative permeability grid cell arrays should be use to enter the grid block end-point data. If DIRECT is to DIRECT then directional end-point scaling is switch on and the same saturation function is used in the x, y and z directions (unless activated otherwise by the SATOPS 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, KROG, KROW 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.. Activates or deactivates directional end-point scaling. 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 REVER 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 xl to xl + 1 direction as for the flow from the xl to the xl – 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 xl to xl + 1 direction and the xl to the xl – 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 oppose 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. Only the default option is supported by OPM Flow.	20
5	MODE	A positive integer that activates the options for temperature dependent saturation end-point scaling. Only the default value, MODE equal to zero, is implemented that 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 "/".			

Table 5.6: ENDSCALE Keyword Description

Examples

```
--        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.

5.2.28 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 the 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. This option is ignored by OPM Flow.	1
5	NSTRVD	A positive integer that defines the maximum number of depth entries in the TVDP tables as described in (4). This option is ignored by OPM Flow.	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 "/".

Table 5.7: 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.

5.2.29 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 and should be defaulted with 1* on the keyword.	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 ignored by OPM Flow.	None
3	THRPRES	A character string that activates the inter-region equilibration flow option. This option allows for a threshold pressure variable entered via the THRPRES keyword to define a pressure which prevents flow between regions until the THRPRES value between regions is exceeded. If the THRPRES command is stated then this option is activated.	None
4	IRREVER	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 IRREVER command is stated then this option is activated. This option is not supported and should be defaulted with 1* on the keyword.	None
Notes: 1) The keyword is terminated by "/".			

Table 5.8: EQLOPTS Keyword Description

Examples

```
--
--       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.

5.2.30 FAULTDIM – DEFINE THE NUMBER OF FAULT SEGMENTS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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 "/".			

Table 5.9: FAULTDIM Keyword Description

Examples

```
--
--        FAULT
--        SEGMS
--
FAULTDIM
          10000                                                    /
```

The above example defines the maximum number of records that can be entered using the FAULT keyword to be 10,0000 segments.

5.2.31 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.

5.2.32 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 ignored by OPM Flow and has no effect on the simulation.

5.2.33 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 5.10.

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 *.DEBUG 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 5.10: 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.

5.2.34 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 5.11.

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 *.DEBUG 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 5.11: 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.

5.2.35 FOAM – ACTIVATE THE FOAM PHASE AND MODEL

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 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 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

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

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

5.2.36 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/".			

Table 5.12: 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.

5.2.37 FULLIMP – ACTIVATE FULLY IMPLICIT SOLUTION OPTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The FULLIMP keyword activates the Fully Implicit Solution formulation and solution options. OPM Flow users 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 2019-10 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.

5.2.38 GAS – ACTIVATE THE GAS PHASE IN THE MODEL

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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.

5.2.39 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 ignored by OPM Flow and has no effect on the simulation.

5.2.40 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 ignored by OPM Flow and has no effect on the simulation.

5.2.41 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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.[18] 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.

[18] *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.*

5.2.42 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 ignored by OPM Flow and has no effect on the simulation, but is documented here for completeness.

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.

5.2.43 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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.

5.2.44 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/".			

Table 5.13: 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.

5.2.45 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 users 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. This option is not supported and should be defaulted with 1* or NO on the keyword.	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

Table 5.14: GRIDOPTS Keyword Description

See also the MULTNUM, MULTREGPV, 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.

5.2.46 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 ignored by OPM Flow and has no effect on the simulation.

5.2.47 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 ignored by OPM Flow and has no effect on the simulation.

5.2.48 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 users 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 2019-10 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.

5.2.49 IMPLICIT – ACTIVATE FULLY IMPLICIT SOLUTION OPTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The IMPLICIT keyword activates the Fully Implicit Solution formulation and solution options. OPM Flow users 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 2019-10 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.

5.2.50 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 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 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 RUNSPEC 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.

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

This keyword is ignored by OPM Flow and has no effect on the simulation.

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.

5.2.51 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.

5.2.52 LICENSES – DEFINE REQUIRED LICENSES FOR RUN

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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

5.2.53 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[19] 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

Example

The following example activates the "black-oil" phases in the model.

```
--
--       ACTIVATE DEAD-OIL PHASE
--
LIVEDOIL
```

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.

[19] *"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.*

5.2.54 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[20] 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 users a different numerical scheme which makes this parameter redundant, see section *2.2 Running OPM Flow 2019-10 From The Command Line* on how to run OPM Flow in parallel mode.	0

Notes:

1) The keyword is terminated by "/".

Table 5.15: LGR Keyword Description

[20] *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.

5.2.55 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
```

5.2.56 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 ignored by OPM Flow and has no effect on the simulation.

5.2.57 LOWSALT – ACTIVATE THE LOW SALT BRINE PHASE IN THE BRINE MODEL

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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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.

5.2.58 MEMORY – DEFINE ALLOCATED MEMORY (RETIRED)

Description

This keyword defines the memory allocation for the run.

OPM Flow and now the commercial simulator users dynamic memory allocation and therefore the keyword has no effect and is ignored by both simulators.

5.2.59 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 ignored by OPM Flow and has no effect on the simulation.

5.2.60 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.

5.2.61 MISCIBLE – DEFINE MISCIBILITY TODD-LONGSTAFF PARAMETERS

Description

The MISCIBLE keyword defines the options associated with the Todd-Longstaff[21] mixing parameters used for when polymer flooding or CO_2 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 "/".			

Table 5.16: 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.

[21] *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.*

5.2.62 MONITOR – ACTIVATE OUTPUT OF THE MONITORING DATA AND FILE

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.

5.2.63 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/".

Table 5.17: MSGFILE Keyword Description

Example

```
--
--        OUTPUT
--        OPTN
MSGFILE
          0                                                                    /
```

The above example deactivates the message file output, but the keyword is ignored by OPM Flow.

5.2.64 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 5.18.

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 *.DEBUG 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 5.18: 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.

5.2.65 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 5.19.

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 *.DEBUG 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 5.19: 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.

5.2.66 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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.

5.2.67 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

5.2.68 NETWORK – ACTIVATE THE EXTENDED NETWORK OPTION AND DEFINE PARAMETERS

Description

This keyword activates the Extended Network option and defines the maxixum number on 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.

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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	Not Used	1*

Notes:

1) The keyword is terminated by "/".

Table 5.20: 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.

5.2.69 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[22].

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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.

[22] *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.*

5.2.70 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 ignored by OPM Flow and has no effect on the simulation.

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 "/".

Table 5.21: 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.

5.2.71 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/".

Table 5.22: NNEWTF Keyword Description

Examples

```
--
--        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.

5.2.72 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 users 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.

5.2.73 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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.

5.2.74 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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.

5.2.75 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.

5.2.76 NOMONITO – DEACTIVATE OUTPUT OF THE MONITORING DATA AND FILE

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.

5.2.77 NONNC – DEACTIVATE NON-NEIGHBOR CONNECTIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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.

5.2.78 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.

5.2.79 NOSIM – ACTIVATE THE NO SIMULATION MODE FOR DATA FILE CHECKING

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

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.

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 queue in NOSIM mode and have software re-run jobs in simulation mode if there are no errors

5.2.80 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/".

Table 5.23: 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.

5.2.81 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 users a different numerical scheme which makes this keyword redundant; see section *2.2 Running OPM Flow 2019-10 From The Command Line* on how to invoke various numerical schemes via the OPM Flow command line interface.

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/".

Table 5.24: 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.

5.2.82 NUMRES – DEFINE THE NUMBER OF RESERVOIR GRIDS

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 "/".			

Table 5.25: 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.

5.2.83 NUPCOL – DEFINE THE NUMBER OF NEWTONIAN ITERATIONS USED TO UPDATE WELL TARGETS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The NUPOL keyword defines the maximum number of Newtonian 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 Newtonian 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 to greater than the default value of three, will improve the accuracy of the group targets and limits at the expense of computational efficiency.

See also section *2.2 Running OPM Flow 2019-10 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 Newtonian iterations used to update well targets within a time step. Note that default value of 12 is different to the commercial simulator's default value of three.	12
Notes: 1) The keyword is terminated by "/".			

Table 5.26: NUPCOL Keyword Description

Example

```
--
--       DEFINE THE NUMBER OF ITERATIONS TO UPDATE WELL FLOW TARGETS
--
NUPCOL
         4                                                                  /
```

The above example sets the default NUPCOL value to four

5.2.84 OIL – ACTIVATE THE OIL PHASE IN THE MODEL

Description

This keyword indicate 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.

5.2.85 OPTIONS – ACTIVATE VARIOUS PROGRAM OPTIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The OPTIONS keyword activates various OPM Flow program options; however only the options known to be available and tested are documented out of the over 200 activation operations available in the commercial simulator.

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

No.	Description	Default
1 - 273	Undefined.	0

Notes:

1) The keyword is terminated by "/".

Table 5.27: 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.

5.2.86 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 2019-10 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 "/".			

Table 5.28: PARALLEL Keyword Description

OPM Flow users 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.

5.2.87 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 ignored by OPM Flow and has no effect on the simulation.

5.2.88 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	NPROCS	A character string enclosed in quotes defining the alias.	None
2	RTYPE	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 "/".

Table 5.29: 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.

5.2.89 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 ignored by OPM Flow and has no effect on the simulation.

5.2.90 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 ignored by OPM Flow and has no effect on the simulation.

5.2.91 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, and the PIMULTAB keyword defined in the SCHEDULE section of the input deck.

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 "/".			

Table 5.30: 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.

5.2.92 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.

5.2.93 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 ignored by OPM Flow and has no effect on the simulation.

5.2.94 RADIAL – RADIAL GRID ACTIVATION OPTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

RADIAL[23] activates the radial grid geometry option for the model, if this keyword is omitted then Cartesian geometry is assumed by OPM Flow.

Although this keyword is read by OPM Flow, radial grids have not been fully implemented and therefore this type of grid should not be used.

[23] *Radial grids are not currently implemented in this version of OPM Flow but are expected to be incorporated in a future release.*

5.2.95 REGDIMS – DEFINE THE MAXIMUM NUMBER OF REGIONS FOR A REGION ARRAY

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. 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 regions. The FIPNUM regions are defined by (1). if additional sets of fluid in-place regions are required, as per the FIPxxx series of fluid in-place region keywords, then these are to be defined here by adding to the value in (1). So for example, if NTFIP equals 5 and the number of distinct FIPxx 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.	1
7	NOPREG	A positive integer defining the maximum number of regions in the OPERNUM region array.	0
8	NWKDREG	A positive integer defining the maximum maximum of real double-precision work arrays for use with the OPERATE and OPERATER keywords	0
9	NWKIREG	A positive integer defining the maximum number of integer work arrays for use with the OPERATE and OPERATER keywords	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 "/".

Table 5.31: 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.

5.2.96 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 ignored by OPM Flow and has no effect on the simulation.

5.2.97 ROCKCOMP – ACTIVATE ROCK COMPACTION

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. Note that this is the default value and is the only option currently supported by OPM Flow. 2) IRREVERS: Rock compaction is irreversible, that is the rock expansion does not occur when the pressure subsequently decreases. 3) HYSTER: Invokes the hysteresis rock compaction option. 4) BOBERG: Rock compaction hysteresis is modeled using the Boberg formulation[24]. 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[25] and [26] 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 default option is 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 the either the ROCKTABW or the ROCK2D and ROCKWNOD keywords should be entered in the PROPS section Only the YES option is currently supported by OPM Flow.	NO

[24] *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.*

[25] *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.*

[26] *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 the porosity dependence transmissibility dependence for when transmissibility is dependent on porosity, should be set to either: 1) EXP: An exponential porosity-transmissibility relationship should be used. 2) CZ: The Carmen-Kozeny[27, 28 and 29] porosity-transmissibility relationship should be used. This option is currently ignored by OPM Flow.	1*
Notes: 1) The keyword is terminated by "/".			

Table 5.32: 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.

[27] *J. Kozeny, "Ueber kapillare Leitung des Wassers im Boden." Sitzungsber Akad. Wiss., Wien, 136(2a): 271-306, 1927.*

[28] *P.C. Carman, "Fluid flow through granular beds." Transactions, Institution of Chemical Engineers, London, 15: 150-166, 1937.*

[29] *P.C. Carman, "Flow of gases through porous media." Butterworths, London, 1956*

5.2.98 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 ignored by OPM Flow and has no effect on the simulation, but is documented here for completeness.

Example

```
--
--       ACTIVATE COUPLE SIMULATION REPORTING
--
RPTCPL
/
```

The above example switches on couple simulation reporting; however, this has no effect in OPM Flow input decks.

5.2.99 RPTHMD - DEFINE WELL HISTORY MATCH GRADIENT REPORTING OPTIONS

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 ignored by OPM Flow and has no effect on the simulation.

5.2.100 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 ignored by OPM Flow and has no effect on the simulation, but is documented here for completeness.

Example

```
--
--       ACTIVATE RUNSPEC SECTION REPORTING
--
RPTRUNSP
```

The above example switches on RUNSPEC reporting; however, this has no effect in OPM Flow input decks.

5.2.101 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.

5.2.102 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.

5.2.103 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.

OPM Flow does not recognize this keyword and the keyword may cause the program to terminate, but it is documented here for completeness.

See section *2.2 Running OPM Flow 2019-10 From The Command Line* on how to invoke various numerical schemes via the OPM Flow command line interface.

5.2.104 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, KRNUMX 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 x_i to x_{i-1} and the x_i to x_{i+1} flow directions. Similarly for the y direction the same curves are used for the y_i to y_{i-1} and the y_i to y_{i+1} flow directions. And again for the z direction, the same relative permeability function is used for flow in the z_i to z_{i-1} and the z_i to z_{i+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 x_i to x_{i+1}, y_i to y_{i+1}, z_i to z_{i+1}, flow directions, respectively. For Radial grids the KRNUMR, KRNUMT and KRNUMZ keywords should be used. For flow in the x_i to x_{i-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[30] is used to describe relative permeability hysteresis and the Killough[31] 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 5.34 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.

[30] *Carlson, F. M. (1981) SPE 10157, presented at the 56th Annual SPE Fall Meeting, San Antonio, 1981*

[31] *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. Only the default option is supported by OPM Flow.	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 x_I to x_{I+1} direction and the x_I to the x_{I-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 x_i to x_{i+1} flow directions etc.. For Radial grids the KRNUMR, KRNUMT and KRNUMZ keywords should be used. For flow in the x_i to x_{i-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.	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 IREVERS 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 x_i to x_{i+1} flow direction etc., the x_i to x_{i-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. Only the default option is supported by OPM Flow.	None
4	SURFTENS	A character string that activates the capillary pressure surface tension pressure dependency option. Only the default option is supported by OPM Flow.	None

No.	Name	Description	Default
Notes: 1) Note that the IRREVERS command can only been activated if the DIRECT command is activated at the same time. See Table 5.34 for the various relative permeability table allocation keywords. 2) The keyword is terminated by "/".			

Table 5.33: 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.				

Table 5.34: 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.

5.2.105 SAVE – ACTIVATE OUTPUT OF A SAVE FILE FOR FAST RESTARTS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword activates output of a SAVE file for fast restarts. There is no data required for this keyword.

This keyword is ignored by OPM Flow and has no effect on the simulation, but is documented here for completeness.

Example

```
--
--       WRITE OUT SAVE FILE FOR FAST RESTARTS
--
SAVE
```

The above example requests that a SAVE file be written out; however, this has no effect in OPM Flow input decks.

5.2.106 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/".

Table 5.35:SCDPDIMS Keyword Description

Examples

```
--
--        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.

5.2.107 SMRYDIMS – DEFINE MAXIMUM NUMBER OF SUMMARY VECTORS TO BE WRITTEN

Description

The SMRYDIMS keyword defines the maximum number of summary vectors to be written out to the SUMMARY file (*.SUMMARY).

OPM Flow users 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 "/".			

Table 5.36: 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.

5.2.108 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")[32] [and] [33] grid nested factorization solver dimensions. This keyword is generated by an external pre-processing program for generating simulation grids.

This keyword is ignored by OPM Flow and has no effect on the simulation.

[32] *Heinemann, Z.E. and Brand, C.W. 1988. Gridding Techniques in Reservoir Simulation. Proc., First Intl. Forum on Reservoir Simulation, Alpbach, Austria, 339.*

[33] *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*

5.2.109 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.

5.2.110 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 "/".			

Table 5.37: 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.

5.2.111 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 SURFACT 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.

Example

```
--
--       ACTIVATE THE SURFACTANT PHASE IN THE MODEL
--
SURFACT
```

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

5.2.112 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.

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.

5.2.113 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 maximum 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 maximum 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 notr 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 compositional commercial simulator.	1*
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 compositional commercial simulator.	1
10	NMEOSS	A positive integer that defines the maximum number of separator or surface equations of states for the compositional commercial 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
12	MXNTHR	A positive integer that defines the maximum number of thermal regions for the compositional commercial simulator.	1

No.	Name	Description	Default
13	NTROCC	A positive integer that defines the maximum number of rock compressibility entries enter by the ROCK keyword defined in the input deck. The tables are allocated to different parts of the grid by the PVTNUM keyword.	1*
14	MXNPMR	A positive integer that defines the maximum number of pressure maintenance regions for the compositional commercial 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 compositional commercial simulator.	0
16	NTALPHA	A positive integer that defines the maximum number of transport coefficient tables for the compositional commercial simulator.	0
17	NASPKA	A positive integer that defines the maximum number of maximum number of entries in the ASPKDAM keyword tables for the compositional commercial simulator.	0
18	MXRAWG	A positive integer that defines the maximum number of maximum number of entries in the ASPREWG keyword tables for the compositional commercial simulator.	0
19	MXRASO	A positive integer that defines the maximum number of pressure maintenance regions for the compositional commercial simulator.	0
20		Not Used	1*
21	MCASPP	A positive integer that defines the maximum number of column entries in the ASPPW2D keyword tables for the compositional commercial simulator.	0
22	MRASPP	A positive integer that defines the maximum number of row entries in the ASPPW2D keyword tables for the compositional commercial simulator.	0
23	MXRATF	A positive integer that defines the maximum number of entries in the ASPWETF table for the compositional commercial simulator.	0
24	MXNKVT	A positive integer that defines the maximum number of composition dependent K-value tables for the compositional commercial simulator.	0
25	RESVED	Not Used	1*

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 "/".

Table 5.38:TABDIMS Keyword Description

Examples

```
--
--        MAX      MAX      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 maximum number of relative permeability tables to be 15 with a maximum number of rows for each table set to 40, and the maximum number of PVT tables to be nine with a maximum number of 30 rows per table.

5.2.114 TEMP – ACTIVATE THE TEMPERATURE MODELING OPTION

Description

This keyword activates the temperature modeling option. 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.

The reservoir temperature can be set to a constant or to be varying with depth utilizing the keywords in the following table:

Input Section	Constant Temperature	Temperature Variation With Depth
PROPS	RTEMP and RTEMPA	
SOLUTION	RTEMP and RTEMPA	RTEMPVD and TEMPVD
Notes: 1) The TEMP option is not implemented in OPM Flow; however, some of the above keywords can be used with OPM Flow's THERMAL option.		

Table 5.39: Reservoir Temperature Keywords

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness. See the THERMAL keyword in the RUNSPEC section to activate OPM Flow's THERMAL option instead.

Example

```
--
--        ACTIVATE THE TEMPERATURE MODELING OPTION
--
TEMP
```

The above example activates the temperature modeling option, although the keyword is ignored by OPM Flow and has no effect on the simulation.

5.2.115 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 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 5.40 for easy of reference.

Section	Keyword	Function	OPM Flow	Commercial	
			THERMAL	TEMP	THERMAL
GRID	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.			
PROPS	GASDENT	Gas Density Temperature Coefficients (OPM Flow keyword).			
	GASVISCT	Gas Viscosity versus Temperature Functions (OPM Flow keyword).			
	OILDENT	Oil Density Temperature Coefficients (OPM Flow keyword).			
	OILVISCT	Oil Viscosity versus Temperature Functions (OPM Flow keyword).			
	RTEMP	Constant Initial Reservoir Temperature.			
	RTEMPA	Constant Initial Reservoir Temperature.			
	RTEMPVD	Initial Reservoir Temperature versus Depth.			
	TEMPVD	Initial Reservoir Temperature versus Depth.			
	SPECHEAT	Specific Heat of Oil, Water and Gas			
	SPECROCK	Specific Heat of the Reservoir Rock			
	WATDENT	Oil Density Temperature Coefficients.			
	WATVISCT	Oil Viscosity versus Temperature Function.			

Section	Keyword	Function	OPM Flow	Commercial	
			THERMAL	TEMP	THERMAL
REGION	THERMNUM	Thermal Region Numbers.			
SOLUTION	RTEMP	Constant Initial Reservoir Temperature.			
	RTEMPA	Constant Initial Reservoir Temperature.			
	TEMPI	Initial Reservoir Temperature for All Cells.			
SCHEDULE	WTEMP	Set An Injection Well's Fluid Temperature			
	WINJTEMP	Define Injection Fluid Thermal Properties			

Notes:

1) 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 keywords.

Table 5.40:THERMAL Option Associated Keywords

Example

```
--
--       ACTIVATE THE THERMAL MODELING OPTION
--
THERMAL
```

The above example activates the thermal modeling option.

5.2.116 TITLE – DEFINE THE TITLE FOR THE INPUT DECK

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 5.41: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.

Examples

```
--
--       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*".

5.2.117 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 ignored by OPM Flow and has no effect on the simulation.

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 "/".			

Table 5.42: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.

5.2.118 TRACERS – ACTIVATE TRACER OPTIONS AND SET TRACER ARRAY DIMENSIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The TRACER keyword defines the number of tracers in the model and the various passive tracer tracking options.

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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.	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.	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).	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

Notes:

1) The keyword is terminated by "/".

Table 5.43:TRACERS Keyword Description

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.

5.2.119 UDADIMS – DEFINE THE DIMENSIONS OF THE USER DEFINED ARGUMENTS

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 augments in a keyword that are replaced by numeric UDQ values.	0
2	NULL	Not Used	1*
3	MXUDA	MXUDA is a positive integer that defines the maximum number of unique augments in a keyword that are replaced numeric UDQ values. Note that MXUDA differs from NMUDA, for example: 3) If only the oil rate argument of, say the WCONPROD keyword is specified by a UDQ, then both NMUDA and NMUDA equal one. 4) However, if a second WCONPROD uses a different UDQ, then NMUDA equals two, but MXUDA would can still be one. 5) Finally, if the same UDQ is 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 UDQ is used as a UDA more than 100 times.	100
Notes: 1) The keyword is terminated by "/".			

Table 5.44: 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.

5.2.120 UDQDIMS – DEFINE THE DIMENSIONS OF THE USER DEFINED UDQ FEATURE

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 "/".			

Table 5.45: 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).

5.2.121 UDQPARAM – DEFINE PARAMETERS FOR THE USER DEFINED QUANTITY FEATURE

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 areal positive value greater than or equal to one and less than or equal to 1.0 x 10^{20}, that sets the absolute range for user define quantities. The default value of 1 x 10^{20} sets the range from -1 x 10^{20} to +1 x 10^{20}.	1 x 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 "/".			

Table 5.46: 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 1×10^{20}, the undefined UDQ variables to zero, and the comparison tolerance to 1.0×10^{-4}.

5.2.122 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/".

Table 5.47: 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.

5.2.123 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

Example

```
--
--       ACTIVATE HISTORY MATCH GRADIENT UNENCODED OUTPUT
--
UNCODHMD
```

The above example switches on the unified output file option.

5.2.124 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 5.48.

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 *.DEBUG 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 5.48: 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.

5.2.125 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 5.49.

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 *.DEBUG 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 5.49: 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.

5.2.126 UNIFOUTS – ACTIVATE THE UNIFIED OUTPUT SUMMARY FILE OPTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

Example

```
--
--       ACTIVATE THE UNIFIED OUTPUT SUMMARY FILE OPTION
--
UNIFOUTS
```

The above example switches on the unified SUMMARY output files.

5.2.127 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

Example

```
--
--      ACTIVATE THE UNIFIED OUTPUT SAVE FILE OPTION
--
UNIFSAVE
```

The above example switches on the unified SUMMARY output files.

5.2.128 VAPOIL – ACTIVATE THE VAPORIZE 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[34] 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[35], 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.

[34] *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.*

[35] *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.*

5.2.129 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 ignored by OPM Flow and has no effect on the simulation.

5.2.130 VFPIDIMS – INJECTION VERTICAL FLOW PERFORMANCE TABLE DIMENSIONS

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 "/".			

Table 5.50: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 maximum number of BHP entries is 12.

5.2.131 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.) entries for the VFPPROD keyword.	0
4	MXMGFR	A positive integer that defines the maximum number of gas fraction entries (GOR, GLR, OGR etc.) entries 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 "/".			

Table 5.51: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.

5.2.132 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 ignored by OPM Flow and has no effect on the simulation.

5.2.133 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[36]. 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

Example

```
--
--       ACTIVATE DUAL POROSITY VISCOUS DISPLACEMENT OPTION
--
VISCD
```

The above example activates the dual porosity viscous displacement option.

[36] *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.*

5.2.134 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-gas, 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 oil phase is active in the model.

5.2.135 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.	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
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

No.	Name	Description	Default
14	MXNGPP	A positive integer defining the maximum number of 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 "/".			

Table 5.52: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.

5.2.136 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

5.2.137 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 branh groups for this model.	1
4	MXLINKS	A positive integer defining the maximum number of segment links per multi-segment well.	0
Notes: 1) The keyword is terminated by "/".			

Table 5.53:WESEGDIMS 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.

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 transmissiblities (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[37]). 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.

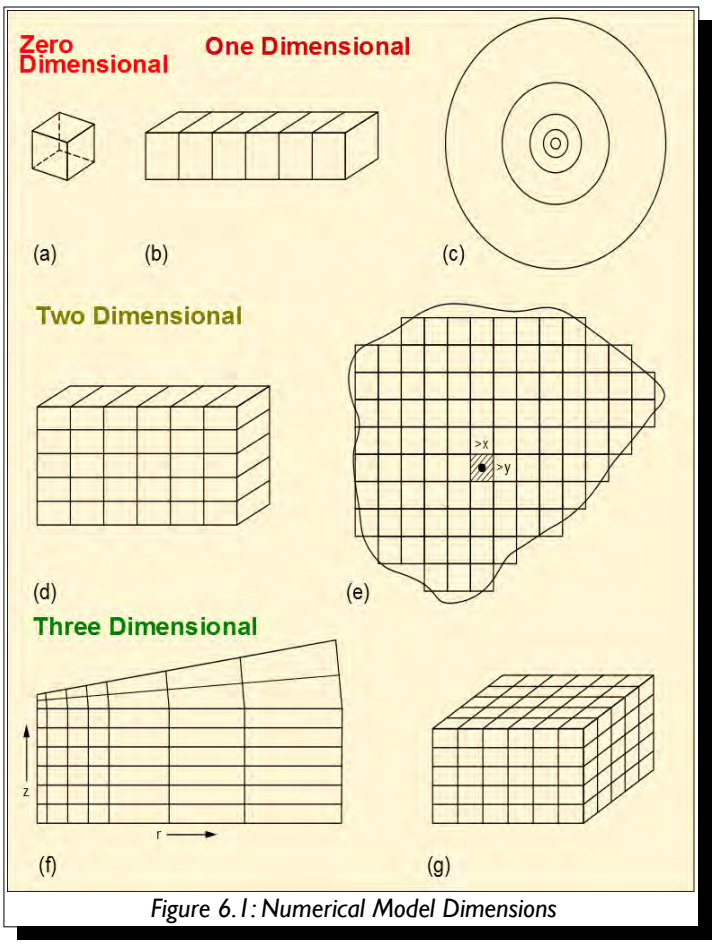


Figure 6.1: Numerical Model Dimensions

OPM Flow enables the user to define 1D, 2D and 3D models using three types of grids: Cartesian Regular Grid, Radial Grid[38], 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.

[37] *Mattax, C.C. and Dalton R.L. 1990. Reservoir Simulation. Society of Petroleum Engineers, Henry L. Doherty Series, Monograph Vol. 13*

[38] *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 GRID

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[39].

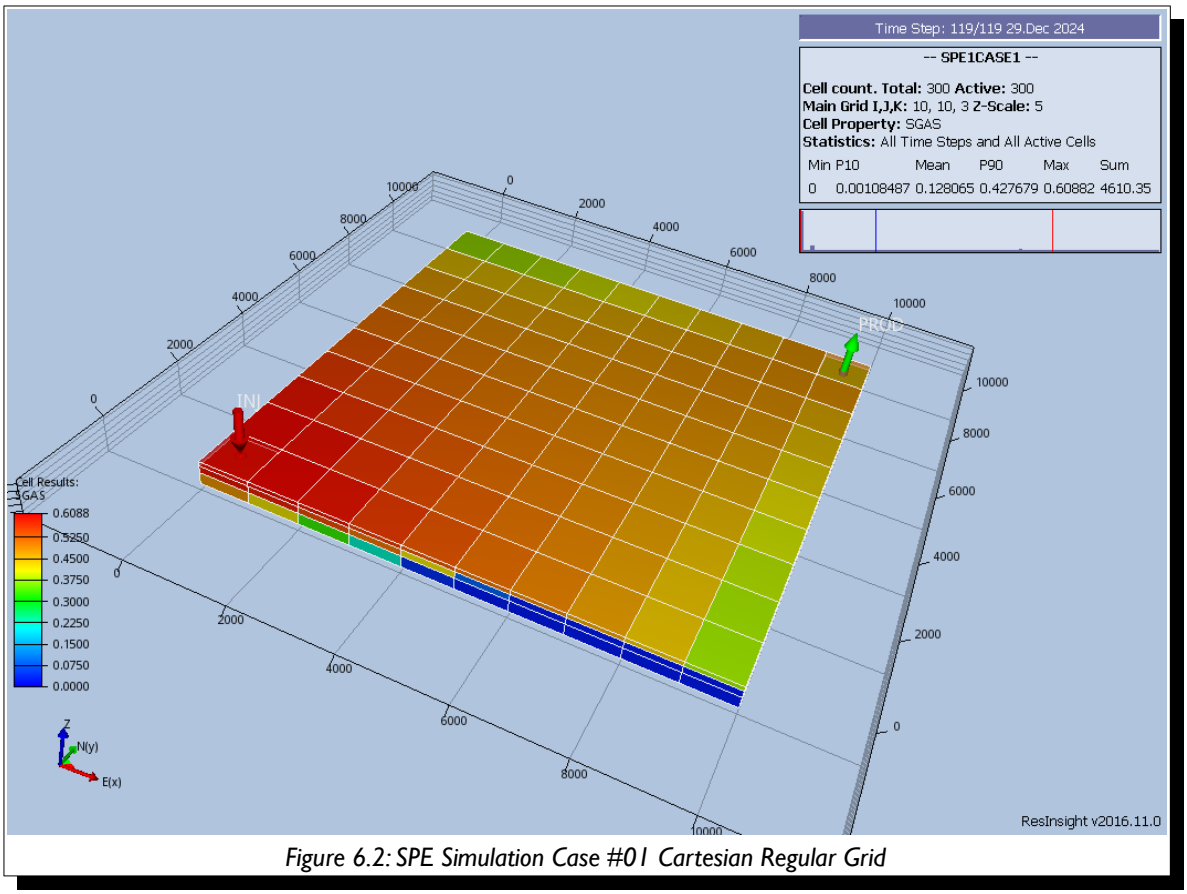


Figure 6.2: SPE Simulation Case #01 Cartesian Regular Grid

The model consists of a simple 10 x 10 x 3 (NX, NY, NY) grid and is defined using the following GRID section keywords to define the grid geometry:

```
--
--  DEFINE GRID BLOCK X DIRECTION CELL SIZE (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 Y DIRECTION CELL SIZE (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 Z DIRECTION CELL SIZE (BASED ON NX x NY x NZ = 300)
-- (The Layers Are 20, 30 And 50 Ft Thick, In Each Layer There Are 100 Cells)
DZ
  100*20.0   100*30.0   100*50.0                                          /
```

[39] *Odeh, A. "Comparison of Solutions to a Three Dimensional Black-Oil Reservoir Simulation Problem." JPT 33 (1981):13-25.*

```
--
--  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 GRID BLOCK POROSITY DATA FOR ALL CELLS (BASED ON NX x NY x NZ = 300)
--
PORO
  300*0.300                                                                 /
--
--  DEFINE GRID BLOCK PERMY DATA FOR ALL CELLS (BASED ON NX x NY x NZ = 300)
--
PERMX
  100*500.0   100%50.0   100*200.0                                          /
--
--  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                                          /
--
--  DEFINE GRID BLOCK 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 RADIAL GRID

To be written in a future release of the manual.

6.2.3 IRREGULAR CORNER-POINT GRIDS

This type of grid is an industry standard grid used to formulate the structure of complex reservoirs. 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.

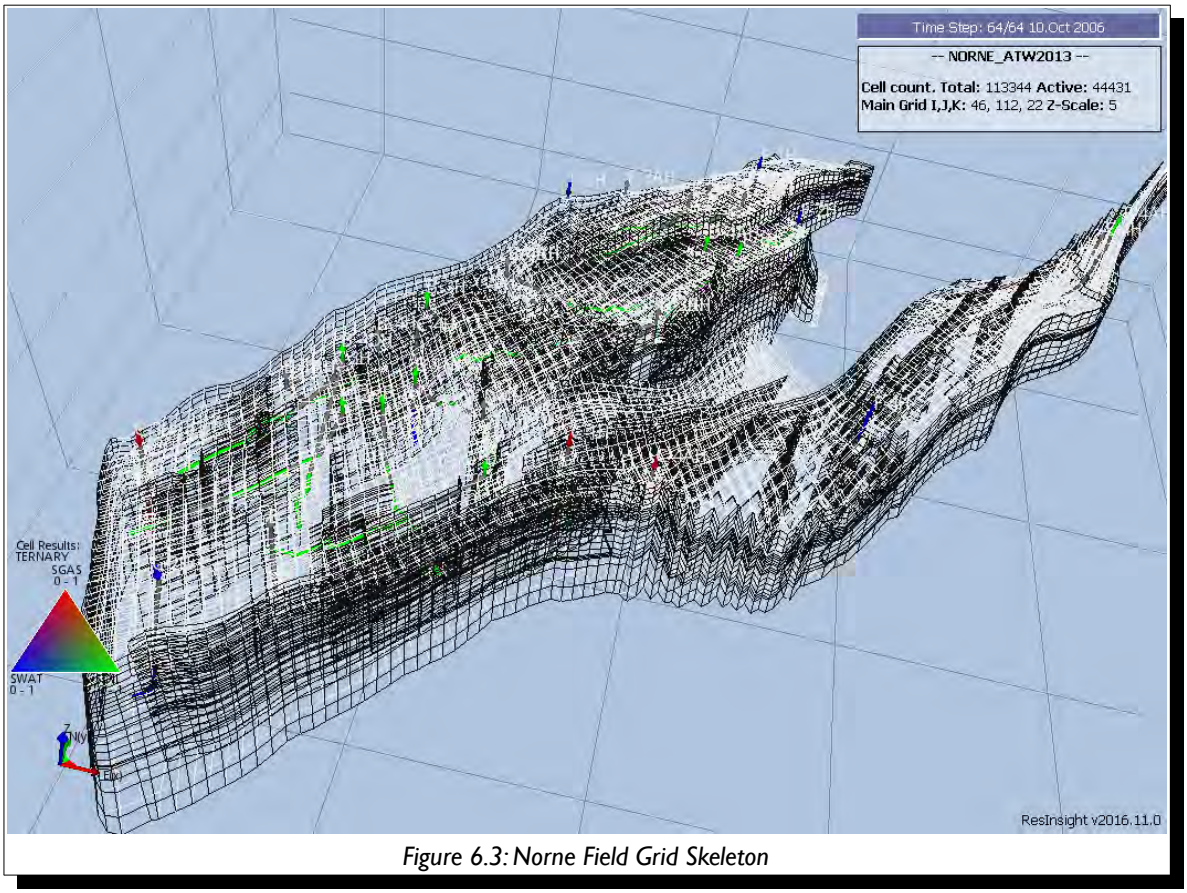


Figure 6.3: Norne Field Grid Skeleton

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                                  /
```

A portion of the coordinate line data defined by the COORD keyword from the Norne model is shown on the next page.

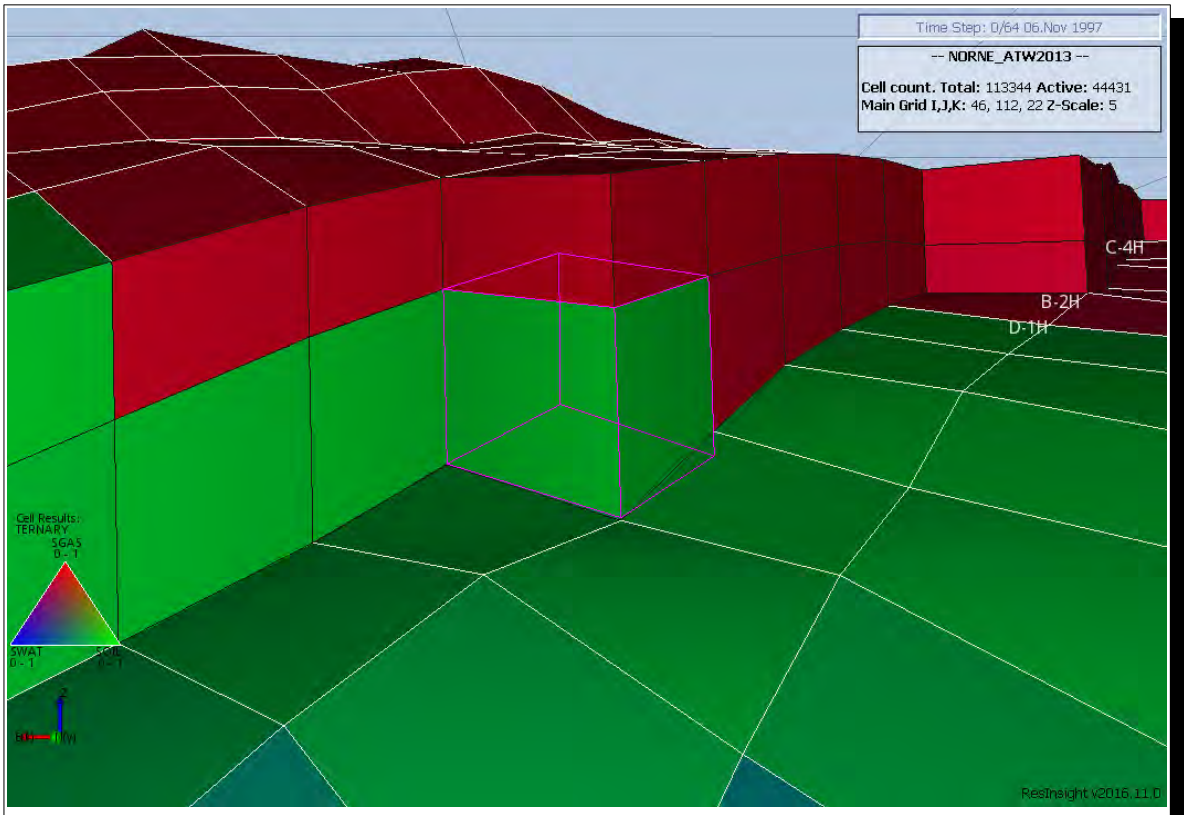


Figure 6.4: Norne Field Corner-Point Geometry Example

```
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 Grid* and the data is defined using the same keywords. The resulting Norne model showing the ternary solution variable is displayed in Figure 6.5.

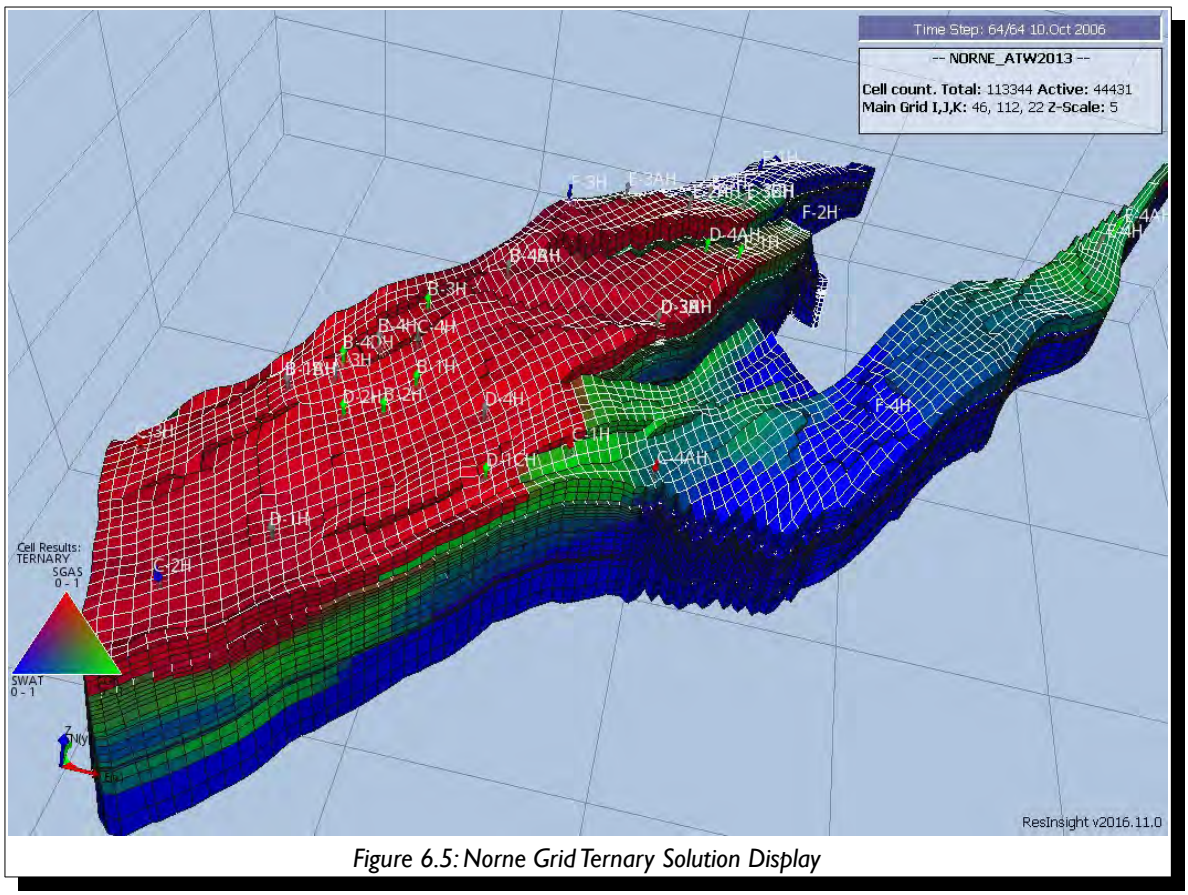


Figure 6.5: Norne Grid Ternary Solution Display

6.2.4 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 keyword 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 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. 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 as 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 keyword 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.

Note that modifications to grid property data can only be done in the GRID section, thereafter only the calculated pore volumes and transmissibilities are available for adjustment.

6.3 KEYWORD DEFINITIONS

6.3.1 ACTNUM – SET THE STATUS OF A GRID BLOCK TO ACTIVE OR INACTIVE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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 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) A total of NX x NY x NZ integer values, as defined by the DIMENS in the RUNSPEC section, must be specified for the array.

2) The keyword is terminated by "/".

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 "/".

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

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 "/".

Table 6.5:ADDREG Keyword Description

The applicable arrays for each section are defined in Table 6.6 on the following page.

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		
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

Examples

```
--
-- 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 ignored by OPM Flow and has no effect on the simulation.

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.

OPM Flow does not recognize this keyword and the keyword may cause the program to terminate, but it is documented here for completeness.

6.3.1 AQUANCON – DEFINE ANALYTICAL CONNECTIONS TO THE GRID

Description

The AQUANCON keyword defines how analytical aquifers are connected to the simulation grid, this includes Carter-Tracy and Fetkovich analytical aquifers, although the Carter-Tracy aquifer is the only aquifer currently implemented in OPM Flow.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	AQUNUM	AQUNUM is a positive integer greater than or equal to one and less than the maximum number of analytical aquifers as defined by the NANAQ 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 for each cell is applied and if a values is declared then then this values is applied to all cells declared by this record.			
		ft^2	m^2	cm^2	1*
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

No.	Name	Description			Default
		Field	Metric	Laboratory	
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 "/".					

Table 6.7: AQUANCON Keyword Description

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.

6.3.2 AQUCON – DEFINE NUMERICAL AQUIFER CONNECTIONS TO THE GRID

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

AQUCON keyword defines how numerical aquifers are connected to the simulation grid.

This keyword is ignored by OPM Flow and has no effect on the simulation.

6.3.3 AQUCT – DEFINE CARTER-TRACY ANALYTICAL AQUIFERS

Description

The AQUCT keyword defines Carter-Tracy 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 NANAQ 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
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 complete surrounds the hydrocarbon reservoir.			
		degrees	degrees	degrees	360.0

No.	Name	Description			Default
		Field	Metric	Laboratory	
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.			1
12	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
13	TEMP	TEMP is a real positive number that defines the initial temperature of the aquifer at DATUM. This variable is ignored by OPM Flow.			
		°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 6.8: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 functions via the AQUTAB keyword in the PROPS section, if the default infinite acting table is not being employed, and 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  NANAQ   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:

```
--
--                        CARTER-TRACY AQUIFER DESCRIPTION
--
--       ID   DATUM   AQF    AQF    AQF    AQF     AQF   AQF   INFL   PVT  AQU
--       NUM  DEPTH   PRESS  PERM   PORO   RCOMP   RET   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.

6.3.4 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 ignored by OPM Flow and has no effect on the simulation.

6.3.5 AQUNUM – DEFINE NUMERICAL AQUIFER PROPERTIES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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.

This keyword is ignored by OPM Flow and has no effect on the simulation.

6.3.6 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 ignored by OPM Flow and has no effect on the simulation.

6.3.7 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

6.3.8 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. 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 "/".

Table 6.9: 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 data elements 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'     0.1000       1   50   1*  1*   20  20 / PERMX TO 100. IN LAYER 20
   'PERMX'     0.1000       50 100   1*  1*   20  20 / PERMX TO 75.0 IN LAYER 20
   'NTG'       0.0500       1*  1*   1*  1*   20  20 / NRT   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.

6.3.9 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 BTOBALFVV keyword in the GRID section that applies a multipliers to individual cells.

This keyword is ignored by OPM Flow and has no effect on the simulation.

6.3.10 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 ignored by OPM Flow and has no effect on the simulation.

6.3.11 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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
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

No.	Name	Description	Default
Notes: 1) The keyword is terminated by "/".			

Table 6.10: 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.

6.3.12 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.

OPM Flow does not recognize this keyword and the keyword may cause the program to terminate.

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 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 "/".

Table 6.11: COALNUM Keyword Description

Example

The example below sets three COALNUM regions for a 4 x 5 x 2 model.

```
COALNUM

2 2 1 1 # layer 1
2 2 1 1
1 1 1 1
1 1 1 1
1 1 1 1

3 3 1 1 # layer 2
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.

6.3.13 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 ignored by OPM Flow and has no effect on the simulation.

6.3.14 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 ignored by OPM Flow and has no effect on the simulation

6.3.15 COORD – DEFINE A SET OF COORDINATES LINES FOR A RESERVOIR GRID

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) lines 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) 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 "/".

Table 6.12: 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.

6.3.16 COORDSYS – DEFINE COORDINATE GRID OPTIONS

Description

This keyword sets various options for when multiple grid systems are being used.

This keyword is ignored by OPM Flow and has no effect on the simulation.

6.3.17 COPY – COPY ARRAY DATA TO ANOTHER ARRAY

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 "/".

Table 6.13: COPY Keyword Description

The applicable arrays for each section are defined in Table 6.14 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 6.14: 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.

6.3.18 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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
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

No.	Name	Description	Default
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 "/".

Table 6.15: COPYBOX Keyword Description

Note that he SOURCE and DESTINATION arrays must be of the same size in all dimensions and the applicable arrays for each section are defined in Table 6.16.

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 6.16: 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.

6.3.19 COPYREG – COPY AN ARRAY TO ANOTHER ARRAY BASED ON A REGION NUMBER

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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.

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/".			

Table 6.17: COPYREG Keyword Description

The applicable arrays for each section are defined in Table 6.18 on the following page.

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		
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: 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
     'PERMZ'     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.

6.3.20 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 ignored by OPM Flow and has no effect on the simulation.

6.3.21 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 ignored by OPM Flow and has no effect on the simulation.

6.3.22 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 ignored by OPM Flow and has no effect on the simulation.

6.3.23 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 ignored by OPM Flow and has no effect on the simulation.

6.3.24 DIFFMTH- – DEFINE GRID BLOCK NEGATIVE THETA DIRECTION DIFFUSIVITY MULTIPLIERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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 ignored by OPM Flow and has no effect on the simulation.

6.3.25 DIFFMX – DEFINE GRID BLOCK X-DIRECTION DIFFUSIVITY MULTIPLIERS

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 ignored by OPM Flow and has no effect on the simulation.

6.3.26 DIFFMX- – DEFINE GRID BLOCK NEGATIVE X-DIRECTION DIFFUSIVITY MULTIPLIERS

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 ignored by OPM Flow and has no effect on the simulation.

6.3.27 DIFFMY – DEFINE GRID BLOCK Y-DIRECTION DIFFUSIVITY MULTIPLIERS

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 ignored by OPM Flow and has no effect on the simulation.

6.3.28 DIFFMY- – DEFINE GRID BLOCK NEGATIVE Y-DIRECTION DIFFUSIVITY MULTIPLIERS

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 ignored by OPM Flow and has no effect on the simulation.

6.3.29 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 ignored by OPM Flow and has no effect on the simulation.

6.3.30 DIFFMZ- – DEFINE GRID BLOCK NEGATIVE Z-DIRECTION DIFFUSIVITY MULTIPLIERS

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 ignored by OPM Flow and has no effect on the simulation.

6.3.31 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 ignored by OPM Flow and has no effect on the simulation.

See section *2.2 Running OPM Flow 2019-10 From The Command Line* on how to run OPM Flow in parallel mode.

6.3.32 DPGRID – ACTIVATE THE MATRIX CELL TO FRACTURE CELL OPTION

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 ignored by OPM Flow and has no effect on the simulation.

6.3.33 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 ignored by OPM Flow and has no effect on the simulation.

6.3.34 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 a Radial Grid model.

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 keyword is terminated by "/".

Table 6.19: DR Keyword Description

See also the DRV, DTHETAV, DZ and TOPS keywords in the GRID section to fully define a radial grid model.

Example

Given the dimensions of the grid in the RUNSPEC section to be 10, 1, 8 for NX, NY and NZ respectively, then DR should be defined as:

```
--
--       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
         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 the INRAD keyword to define the inner radius of the radial grid.

6.3.35 DRV - DEFINE THE SIZE OF GRID BLOCKS IN THE R DIRECTION VIA A VECTOR

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

DRV[40] 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 Grid. The RADIAL keyword in the RUNSPEC section should be activated to indicate that radial geometry is being used.

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/".

Table 6.20: DRV Keyword Description

See also the DR, DTHETAV, DZ and TOPS keywords in the GRID section to fully define a radial 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  1101.0  /
```

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.

[40] *Radial grids are not currently implemented in this version of OPM Flow, but is expected to be incorporated in a future release.*

6.3.36 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 R direction via an array for each cell in a Radial Grid model.

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	DR	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 "/".

Table 6.21: DTHETA Keyword Description

See also the DRV, DTHETAV, DZ and TOPS keywords in the GRID section to fully define a radial 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.

6.3.37 DTHETAV - SETS THE SIZE OF GRID BLOCKS IN THETA DIRECTION VIA A VECTOR

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

DTHETAV[41] 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 keyword in the RUNSPEC should be activated to indicate that radial geometry is being used.

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/".

Table 6.22: DTHETA Keyword Description

See also the DRV, DZV and TOPS keywords to fully define a radial 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.

[41] *Radial grids are not currently implemented in this version of OPM Flow, but is expected to be incorporated in a future release.*

6.3.38 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.

6.3.39 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 "/".

Table 6.23: 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.

6.3.40 DXV - DEFINE THE SIZE OF GRID BLOCKS IN THE X DIRECTION VIA A VECTOR

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 "/".

Table 6.24: 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.

6.3.41 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 "/".

Table 6.25: 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.

6.3.42 DYV - DEFINE THE SIZE OF GRID BLOCKS IN THE Y DIRECTION VIA A VECTOR

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 "/".

Table 6.26: 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.

6.3.43 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 "/".

Table 6.27: 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.

6.3.44 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 ignored by OPM Flow and has no effect on the simulation.

6.3.45 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 ignored by OPM Flow and has no effect on the simulation.

6.3.46 DZMTRXV - 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 DZMATRIX keyword in the GRID section.

This keyword is ignored by OPM Flow and has no effect on the simulation.

6.3.47 DZNET – DEFINE GRID BLOCK NET THICKNESS FOR ALL CELLS

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 "/".

Table 6.28: 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.

6.3.48 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) he keyword is terminated by "/".

Table 6.29: 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.

6.3.49 ENDBOX – DEFINE THE END OF THE BOX DEFINED GRID

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.

6.3.50 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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.

6.3.51 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 ignored by OPM Flow and has no effect on the simulation.

6.3.52 EQUALREG – SETS AN ARRAY TO A CONSTANT BY REGION NUMBER

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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/".

Table 6.30: EQUALREG Keyword Description

The applicable arrays for each section are defined in Table 6.31 on the following page.

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		
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 6.31: EQUALREG Keyword Applicable Arrays by Section

Examples

```
-- 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.

6.3.53 EQUALS – SETS A SPECIFIED ARRAY TO A CONSTANT

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 "/".

Table 6.32: EQUALS Keyword Description

The applicable arrays for each section are defined in Table 6.33 on the following page.

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		
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 6.33: 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.

6.3.54 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 ignored by OPM Flow and has no effect on the simulation.

6.3.55 EXTHOST - DEFINE HOST CELLS FOR EXTERNAL LGRS

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 ignored by OPM Flow and has no effect on the simulation.

6.3.56 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 ignored by OPM Flow and has no effect on the simulation.

6.3.57 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 FAULT 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 "/".

Table 6.34: 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'                          /
……..
/
```

6.3.58 FILEUNIT – ACTIVATE UNIT CONSISTENCY CHECKING

Description

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.

No.	Name	Description	Default
1	FILEUNIT	A character string that defines the units of the coordinates stated on the MAPAXES keyword, 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 "/".			

Table 6.35: 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.

6.3.59 FLUXNUM – DEFINE THE FLUX REGIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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 "/".

Table 6.36: FLUXNUM Keyword Description

Examples

The example below sets three FLUXNUM regions for a 4 x 5 x 2 model.

```
FLUXNUM

2 2 1 1 # layer 1
2 2 1 1
1 1 1 1
1 1 1 1
1 1 1 1

3 3 1 1 # layer 2
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
/
```

6.3.60 FLUXREG – DEFINE ACTIVE FLUX REGIONS

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 ignored by OPM Flow and has no effect on the simulation.

6.3.61 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

6.3.62 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. This type of file is operating system dependent, that is a Microsoft Windows generated binary file cannot be read in by a Linux based system unlike formatted files. If the variable FMTOPT is omitted then the default is for binary file input.	U
Notes: 1) The keyword is terminated by "/".			

Table 6.37: 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'  /
```

6.3.63 GDORIENT - DEFINE GRID ORIENTATION PARAMETERS

Description

This keyword defines the grid orientation parameters for post-processing applications.

This keyword is ignored by OPM Flow and has no effect on the simulation.

6.3.64 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 ignored by OPM Flow and has no effect on the simulation.

6.3.65 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.

6.3.66 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 of the GRID geometry file: 1) - for no GRID file to be written out. 2) - for the standard GRID file to be written out. 3) - 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 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 "/".			

Table 6.38: 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

6.3.67 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) FIELD for field units, 2) METRES for metric units, or 3) LAB 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 "/".

Table 6.39: GRIDUNIT Keyword Description

Example

```
--
--       SET THE GRID UNITS FOR THE GRID
--
GRIDUNIT
         METRES                                                    /
```

The above example defines that the GRID units to be metric.

6.3.68 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 ignored by OPM Flow and has no effect on the simulation.

6.3.69 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 users 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/ft^3/°R	kJ/m^3/K	J/cm^3/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 "/".

Table 6.40: 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)
--
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.

6.3.70 HEATCRT – DEFINE RESERVOIR ROCK HEAT CAPACITY TEMPERATURE DEPENDENCE FOR ALL CELLS

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 users 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/ft^3/°R^2	kJ/m^3/K^2	J/cm^3/K^2	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 "/".					

Table 6.41: 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 use to calculate the reservoir rock volumetric heat capacity temperature dependence using the following relationship:

$$\text{Heat Capacity of Rock} = \text{HEATCR}\left(Temp - Temp_{ref}\right) + \frac{\text{HEATCRT}\left(Temp - Temp_{ref}\right)^2}{2} \quad \textbf{(6.3)}$$

Example

```
--
--       DEFINE RESERVOIR ROCK HEAT CAPACITY TEMPERATURE DEPENDENCE
--       FOR ALL CELLS (BASED ON NX x NY x NZ = 300)
--
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.

6.3.71 HMFAULTS – HISTORY MATCH FAULT GRADIENT PARAMETERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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 ignored by OPM Flow and has no effect on the simulation.

6.3.72 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 ignored by OPM Flow and has no effect on the simulation.

6.3.73 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 ignored by OPM Flow and has no effect on the simulation.

6.3.74 HMMLT – HISTORY MATCH GRID PERMEABILITY GRADIENT CUMULATIVE MULTIPLIERS

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 6.42, that determines the permeability direction, for example, HMMLTPX.

This keyword is ignored by OPM Flow and has no effect on the simulation.

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 6.42: HMMLT Keyword List

See also the HMMULT keyword in the EDIT section

6.3.75 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 ignored by OPM Flow and has no effect on the simulation.

6.3.76 HMMULRGT – HISTORY MATCH REGION TRANSMISSIBILITY PARAMETERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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 ignored by OPM Flow and has no effect on the simulation.

6.3.77 HMMULTFT – HISTORY MATCH FAULT TRANSMISSIBILITY GRADIENT CUMULATIVE MULTIPLIERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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 ignored by OPM Flow and has no effect on the simulation.

6.3.78 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 ignored by OPM Flow and has no effect on the simulation.

6.3.79 HRFIN - DEFINE THE RATIO OF LGR GRID BLOCKS IN THE R-DIRECTION

Description

HRFIN[42] 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/".

Table 6.43: 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.

[42] *Radial grids are not currently implemented in this version of OPM Flow, but is expected to be incorporated in a future release.*

6.3.80 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/".

Table 6.44: 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.

6.3.81 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/".

Table 6.45: 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.

6.3.82 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/".

Table 6.46: 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.

6.3.83 IHOST – ASSIGN LGRS TO PARALLEL PROCESS NUMBER

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 ignored by OPM Flow and has no effect on the simulation.

6.3.84 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 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-processoring software and the keyword allows for both formatted and unformatted (binary) files to be loaded.

This keyword is ignored by OPM Flow and has no effect on the simulation.

6.3.85 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 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.

6.3.86 INRAD – DEFINE THE INNER RADIUS OF A RADIAL GRID

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

INRAD[43] defines the inner radius of the reservoir model for a radial grid geometry. The RADIAL 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 "/".					

Table 6.47: INRAD Keyword Description

See also the DR, DRV, DTHETA, DTHETAV and TOPS keywords to fully define a Radial Grid.

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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.

[43] *Radial grids are not currently implemented in this version of OPM Flow, but is expected to be incorporated in a future release.*

6.3.87 IONROCK – DEFINE THE ION EXCHANGE CAPACITY FOR ALL THE CELLS

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 ignored by OPM Flow and has no effect on the simulation.

6.3.88 ISOLNUM – DEFINE THE INDEPENDENT RESERVOIR REGIONS

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 "/".

Table 6.48: ISOLNUM Keyword Description

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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.

6.3.89 JFUNC - ACTIVATE THE LEVERETT J-FUNCTION OPTION

Description

JFUNC keyword activates Leverett-J-Function[44] 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. 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.

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

[44] *Leverett, M. C.; "Capillary Behaviour in Porous Solids", Trans. AIME (1941) 142, 152-168.*

No.	Name	Description	Default
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 "/".			

Table 6.49: 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[45], 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:

$$J\left(S_w\right) = P_{c,res}\left(S_w\right)\sqrt{\frac{k}{\varphi}} \tag{6.6}$$

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:

[45] *Leverett, M. C.; "Capillary Behaviour in Porous Solids", Trans. AIME (1941) 142, 152-168.*

$$J\left(S_w\right) = \frac{P_{c,res}\left(S_w\right)\left(\frac{k^{\beta}}{\varphi^{\alpha}}\right)}{\sigma} \quad \textbf{(6.7)}$$

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 capillary pressure data entered on the SWFN, SGFN or the SWOF, SGOF, SLGOF keywords, should be replaced by dimensionless J-function values.

See also the JFUNCR keyword in the GRID section that performs similar calculations based on the saturation tables.

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.

6.3.90 JFUNCR - ACTIVATE THE LEVERETT J-FUNCTION SATURATION TABLE OPTION

Description

JFUNCR keyword activates Leverett-J-Function[46] 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 cells 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.

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness. See the JFUNC keyword in the GRID section to activate J-Function scaling based on global parameters.

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

[46] *Leverett, M. C.; "Capillary Behaviour in Porous Solids", Trans. AIME (1941) 142, 152-168.*

No.	Name	Description	Default
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 "/" and there is no "/" terminator for the keyword.

Table 6.50: 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[47], 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 actually ignore the denominator as it is a constant, and we can therefore just use:

$$J\left(S_w\right) = P_{c,res}\left(S_w\right)\sqrt{\frac{k}{\varphi}} \tag{6.10}$$

[47] *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:

$$J\left(S_w\right) = \frac{P_{c,res}\left(S_w\right)\left(\frac{k^{\beta}}{\varphi^{\alpha}}\right)}{\sigma} \qquad (6.11)$$

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 capillary pressure data entered on the SWFN, SGFN or the SWOF, SGOF, SLGOF keywords, should be replaced by dimensionless J-function values.

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.

6.3.91 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.

6.3.92 LINKPERM - ASSIGN CELL PERMEABILITIES TO CELL FACES

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 ignored by OPM Flow and has no effect on the simulation.

6.3.93 LTOSIGMA - DUAL POROSITY VISCOUS DISPLACEMENT SIGMA PARAMETERS

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 ignored by OPM Flow and has no effect on the simulation.

6.3.94 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/".

Table 6.51: 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.

6.3.95 LXFIN – DEFINE LOGARITHMIC LGR GRID BLOCK SPACING IN THE X-DIRECTION

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 ignored by OPM Flow and has no effect on the simulation.

6.3.96 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/".

Table 6.52: 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.

6.3.97 LYFIN – DEFINE LOGARITHMIC LGR GRID BLOCK SPACING IN THE Y-DIRECTION

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 ignored by OPM Flow and has no effect on the simulation.

6.3.98 LZ - DUAL POROSITY VISCOUS DISPLACEMENT Z DIRECTION MATRIX SIZE FOR ALL CELLS

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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/".

Table 6.53: 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 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.

6.3.99 LZFIN – DEFINE LOGARITHMIC LGR GRID BLOCK SPACING IN THE Z-DIRECTION

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 ignored by OPM Flow and has no effect on the simulation.

6.3.100 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 "/".					

Table 6.54: 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.

6.3.101 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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) FIELD for field units 2) METRES for metric units, or 3) LAB for laboratory units	METRES
Notes: 1) Note the alternative spelling of METRES, that is METERS is not recognized. 2) The keyword is terminated by "/".			

Table 6.55: 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.

6.3.102 MAXVALUE – SETS A MAXIMUM VALUE FOR AN ARRAY ELEMENT

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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.

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 grater 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 "/".

Table 6.56: MAXVALUE Keyword Description

The applicable arrays for each section are defined in Table 6.57 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 6.57: MAXVALUE Keyword Applicable Arrays by Section

Example

```
--
--        -- ARRAY     CONSTANT --  ---------- BOX ---------
--                                   I1  I2   J1  J2   K1  K2
MAXVALUE
         'PERMX'      1.0E2        1*  1*   1*  1*   5   5  / MAXIMUM PERMX
         'PERMY'      1.0E2        1*  1*   1*  1*   5   5  / MAXIMUM PERMY
         'PERMZ'      1.0E1        1*  1*   1*  1*   5   5  / MAXIMUM 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 in layer five.

6.3.103 MINNNCT – SET A MINIMUM NON-NEIGHBOR CONNECTION TRANSMISSIBILITY

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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 ignored by OPM Flow and has no effect on the simulation.

6.3.104 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 mode (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 ignored by OPM Flow and has no effect on the simulation, but see the MINPV keyword in the GRID section that 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	rm^3	rcc	
		1.0e-6	1.0e-6	1.0e-6	Defined

Notes:

1) The keyword is terminated by "/".

Table 6.58: MINPORV Keyword Description

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 m^3) as the minimum pore volume for a cell to be active in the model.

6.3.105 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 mode (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.

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	rm^3	rcc	
		1.0e-6	1.0e-6	1.0e-6	Defined
Notes: 1) The keyword is terminated by "/".					

Table 6.59: MINPV Keyword Description

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 m^3) as the minimum pore volume for a cell to be active in the model.

6.3.106 MINPVV – SET A MINIMUM GRID BLOCK PORE VOLUME THRESHOLD FOR INDIVIDUAL CELLS

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 keyword in the GRID section that sets 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 "/".

Table 6.60: MINPVV Keyword Description

See also the PINCH keyword for the treatment of inactive grid cells and pinch-outs.

Example

```
--
--       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 LAYER
--
--       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
```

The above example defines 500 rb (or m^3) as the minimum pore volume for all cells in layer 19 to be active in the model, and 750 rb (or m^3) as the minimum pore volume for all cells in layer 20.

6.3.107 MINVALUE – SET A MINIMUM VALUE FOR AN ARRAY ELEMENT

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.

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/".

Table 6.61: MINVALUE Keyword Description

The applicable arrays for each section are defined in Table 6.62 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 6.62: MINVALUE Keyword Applicable Arrays by Section

Example

```
--
--        -- ARRAY     CONSTANT --  ---------- BOX ---------
--                                   I1  I2   J1  J2   K1  K2
MINVALUE
         'PERMX'      1.0E1        1*  1*   1*  1*   5   5  / MAXIMUM PERMX
         'PERMY'      1.0E1        1*  1*   1*  1*   5   5  / MAXIMUM PERMY
         'PERMZ'      1.0E-1       1*  1*   1*  1*   5   5  / MAXIMUM 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 in layer five.

6.3.108 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 ignored by OPM Flow and has no effect on the simulation.

6.3.109 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.

OPM Flow does not recognize this keyword and the keyword may cause the program to terminate, but it is documented here for completeness.

6.3.110 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 the 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 "/".

Table 6.63: 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.

6.3.111 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 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 "/".

Table 6.64: MULTIPLY Keyword Description

The applicable arrays for each section are defined in Table 6.65 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 6.65: 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.

6.3.112 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 "/" .			

Table 6.66: MULTIREG Keyword Description

The applicable arrays for each section are defined in Table 6.67 as shown on the following page.

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		
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 6.67: 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.

6.3.113 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 "/".

Table 6.68: MULTNUM Keyword Description

Examples

The example below sets three MULTNUM regions for a 4 x 5 x 2 model.

```
MULTNUM

2 2 1 1 # layer 1
2 2 1 1
1 1 1 1
1 1 1 1
1 1 1 1

3 3 1 1 # layer 2
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               /
/
```

6.3.114 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 "/".

Table 6.69: MULTPV Keyword Description

See also the MULTREGP 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.

6.3.115 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 grids, as declared by the RADIAL keyword in the RUNSPEC section.

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/".

Table 6.70: MULTR Keyword Description

See also the MULTR-, MULTTHT, MULTTHT-, MULTZ and MULTZ- keywords for scaling transmissible between radial grid cells.

Examples

```
--
--       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.

6.3.116 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 grids, as declared by the RADIAL keyword in the RUNSPEC section.

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/".

Table 6.71: MULTR- Keyword Description

See also the MULTR, MULTTHT, MULTTHT-, MULTZ and MULTZ- keywords for scaling transmissible between radial grid cells.

Examples

```
--
--       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.

6.3.117 MULTREGD – MULTIPLY DIFFUSIVITIES BETWEEN REGIONS

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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/".

Table 6.72: 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.

6.3.118 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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: 4) NNC – Only apply the thermal conductivity multiplier between REGION1 and REGION2 to non-neighbor connections. 5) NONNC – Do not apply the thermal conductivity multiplier between REGION1 and REGION2 to non-neighbor connections. 6) 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 "/".

Table 6.73: 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.

6.3.119 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 "/".			

Table 6.74: 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.

6.3.120 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 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: 7) NNC – Only apply the transmissibility multiplier between REGION1 and REGION2 to non-neighbor connections. 8) NONNC – Do not apply the transmissibility multiplier between REGION1 and REGION2 to non-neighbor connections. 9) 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 "/".

Table 6.75: 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.

6.3.121 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 grids, as declared by the RADIAL keyword in the RUNSPEC section.

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/".

Table 6.76: MULTTHT Keyword Description

See also the MULTTHT-, MULTR, MULTR-, MULTZ and MULTZ- keywords for scaling transmissible between radial grid cells.

Examples

```
--
--       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.

6.3.122 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 grids, as declared by the RADIAL keyword in the RUNSPEC section.

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/".

Table 6.77: MULTTHT- Keyword Description

See also the MULTTHT, MULTR, MULTR-, MULTZ and MULTZ- keywords for scaling transmissible between radial grid cells.

Examples

```
--
--       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.

6.3.123 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 "/".			

Table 6.78: MULTX Keyword Description

See also the MULTX-, MULTY, MULTY-, MULTZ and MULTZ- keywords for scaling transmissible between grid cells.

Examples

```
--
--       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.

6.3.124 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 "/".

Table 6.79: MULTX- Keyword Description

See also the MULTX, MULTY, MULTY-, MULTZ and MULTZ- keywords for scaling transmissible between grid cells.

Examples

```
--
--       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.

6.3.125 MULTY - MULTIPLY CELL TRANSMISSIBILITY IN THE +Y DIRECTION

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 "/".

Table 6.80: MULTY Keyword Description

See also the MULTY-, MULTX, MULTX-, MULTZ and MULTZ- keywords for scaling transmissible between grid cells.

Examples

```
--
--       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.

6.3.126 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 "/".

Table 6.81: MULTY- Keyword Description

See also the MULTY, MULTX, MULTX-, MULTZ and MULTZ- keywords for scaling transmissible between grid cells.

Examples

```
--
--       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.

6.3.127 MULTZ - MULTIPLY CELL TRANSMISSIBILITY IN THE +Z DIRECTION

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 "/".

Table 6.82: MULTZ Keyword Description

See also the MULTZ-, MULTX, MULTX-, MULTY and MULTY- keywords for scaling transmissible between grid cells.

Examples

```
--
--       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.

6.3.128 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 "/".

Table 6.83: MULTZ- Keyword Description

See also the MULTZ, MULTX, MULTX-, MULTY and MULTY- keywords for scaling transmissible between grid cells.

Examples

```
--
--       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.

6.3.129 NEWTRAN – ACTIVATE IRREGULAR CORNER-POINT GRID TRANSMISSIBILITIES

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.

6.3.130 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/".

Table 6.84: 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 users nine-point discretization, if NINEPOIN is present in the RUNSPEC section, or five-point if NINEPOIN is absent.

Examples

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.

6.3.131 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 ignored by OPM Flow and has no effect on the simulation.

6.3.132 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 to be joined 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 to be joined 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 to be joined 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 to be joined 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 to be joined 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 to be joined 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 vaules 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 "/".

Table 6.85: 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 -----------    -- TRANSNCC --
--        I1   J1   K1    I2   J2   K2
NCC
          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.

6.3.133 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.

6.3.134 NOGGF – DEACTIVATE OUTPUT OF GRID GEOMETRY FILE

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 ignored by OPM Flow and has no effect on the simulation.

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.

6.3.135 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 "/".

Table 6.86: 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.

6.3.136 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/".

Table 6.87: 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 BLOCK 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.

6.3.137 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/".

Table 6.88: 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 BLOCK 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.

6.3.138 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/".

Table 6.89: 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 BLOCK 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.

6.3.139 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.

6.3.140 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.

6.3.141 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 "/".

Table 6.90: OPERATE Keyword Description

The applicable arrays for each section are defined in Table 6.91 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 6.91: 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
--
--        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.

6.3.142 OPERATER – DEFINE MATHEMATICAL OPERATIONS ON ARRAYS BY REGION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

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.

This keyword is ignored by OPM Flow and has no effect on the simulation.

6.3.143 OPERNUM – DEFINE REGIONS FOR MATHEMATICAL OPERATIONS ON ARRAYS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

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 the MULTNUM region array.

This keyword is ignored by OPM Flow and has no effect on the simulation.

6.3.144 OUTRAD - DEFINE THE OUTER RADIUS OF A RADIAL GRID

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

OUTRAD[48] defines the OUTER radius of the reservoir model for a radial grid geometry. The RADIAL keyword in the RUNSPEC should be activated to indicate that radial geometry is being used.

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 "/".					

Table 6.92: OUTRAD Keyword Description

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.
R_{i-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 6.93 shows the grid size calculations.

[48] *Radial grids are not currently implemented in this version of OPM Flow, but is expected to be incorporated in the next release.*

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 6.93: OUTRAD Radial Grid Example

See also the DR, DRV, DTHETA, DTHETAV and TOPS keywords to fully define a Radial 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.

6.3.145 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.

OPM Flow does not recognize this keyword and the keyword may cause the program to terminate, but it is documented here for completeness.

See section *2.2 Running OPM Flow 2019-10 From The Command Line* on how to run OPM Flow in parallel mode.

6.3.146 PEBI – ACTIVATE AND DEFINED PEBI GRID OPTIONS

Description

PEBI activates the unstructured Perpendicular Bisector ("PEBI")[49] and [50] loading of grid data generated by an external pre-processing program for generating simulation grids.

This keyword is ignored by OPM Flow and has no effect on the simulation, but is documented here for completeness.

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 6.94: PEBI Keyword Description

Example

```
--
--      OPTION1  OPTION2
--      CHECK    CALCULATE
GRIDOPTS
        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.

[49] *Heinemann, Z.E. and Brand, C.W. 1988. Gridding Techniques in Reservoir Simulation. Proc., First Intl. Forum on Reservoir Simulation, Alpbach, Austria, 339.*

[50] *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*

6.3.147 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 ignored by OPM Flow and has no effect on the simulation.

6.3.148 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[51] 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.

OPM Flow does not recognize this keyword and the keyword may cause the program to terminate, but it is documented here for completeness.

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 "/".

Table 6.95: 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.

[51] *Leverett, M. C.; "Capillary Behaviour in Porous Solids", Trans. AIME (1941) 142, 152-168.*

6.3.149 PERMR – DEFINE THE PERMEABILITY FOR EACH CELL IN THE R DIRECTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

PERMR[52] sets the permeability for each cell in the R direction in a radial geometry grid. The RADIAL keyword in the RUNSPEC should be activated to indicate that radial geometry is being used.

Although this keyword is read by OPM Flow, radial grids have not been fully implemented and therefore this type of grid should not be used.

[52] *Radial grids are not currently implemented in this version of OPM Flow but are expected to be incorporated in a future release.*

6.3.150 PERMTHT – DEFINE THE PERMEABILITY FOR EACH CELL IN THE THETA DIRECTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

PERMTHT[53] sets the permeability for each cell in the THETA direction in a radial geometry grid. The RADIAL keyword in the RUNSPEC should be activated to indicate that radial geometry is being used.

Although this keyword is read by OPM Flow, radial grids have not been fully implemented and therefore this type of grid should not be used.

[53] *Radial grids are not currently implemented in this version of OPM Flow but are expected to be incorporated in a future release.*

6.3.151 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 "/".

Table 6.96: 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.

6.3.152 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 "/".

Table 6.97: 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.

6.3.153 PERMZ - DEFINE THE PERMEABILITY IN THE Z DIRECTION FOR ALL THE CELLS

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 "/".

Table 6.98: PERMZ Keyword Description

Note

Although PERMX and PERMZ 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

```
--
--       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 above example 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.

6.3.154 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 ignored by OPM Flow and has no effect on the simulation.

6.3.155 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 deactive 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 "/".					

Table 6.99: 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
--       THICKNESS   NO GAP   GAP     CALC
         0.01        1*       1*      1*                                  /
```

For the second example, the MINPV keyword is used to set the minimum pore volume to 500 m^3 (metric units) and then the PINCH keyword is invoked with PINCHGAP set equal to GAP, as follows:

```
--
--       MINIMUM PORE VOLUME FOR ACTIVE CELLS
--
MINPV
         1000.0                                                          /
--
--       SET PINCH-OUT CRITERIA FOR THE MODEL
--
PINCH
--       THRESHOLD   GAP      EMPTY   TRANS
--       THICKNESS   NO GAP   GAP     CALC
         0.1         GAP       1*     1*                                  /
```

In the above example the MINPV keyword will deactivate all cells with pore volumes less than 500 m^3. 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.

6.3.156 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/".

Table 6.100: PINCHNUM Keyword Description

Examples

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.

6.3.157 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

Examples

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
```

6.3.158 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 6.101: 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.

6.3.159 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/".					

Table 6.102: PINCHXY Keyword Description

Examples

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                                    /
```

6.3.160 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 "/".

Table 6.103: 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.

6.3.161 QMOBIL ACTIVATE OR DEACTIVATE LGR END-POINT MOBILITY CORRECTION

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 ignored by OPM Flow and has no effect on the simulation.

6.3.162 RADFIN – DEFINE A RADIAL LOCAL GRID REFINEMENT WITH ONE COLUMN

Description

This keyword defines a radial local grid refinement using one columns Local grid refinement is currently not supported by OFM Flow.

OPM Flow does not recognize this keyword and the keyword may cause the program to terminate, but it is documented here for completeness.

6.3.163 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

6.3.164 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 ignored by OPM Flow and has no effect on the simulation.

6.3.165 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.

See the NUMRES keyword in the RUNSPEC section that defines he number of reservoir grids (COORD data sets) that the simulator should process.

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/".			

Table 6.104: RESVNUM Keyword Description

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
/
```

6.3.166 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/".

Table 6.105: 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.

6.3.167 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 ignored by OPM Flow and has no effect on the simulation, but is documented here for completeness.

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 "/".

Table 6.106: 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                  /
```

6.3.168 RPTGRIDL – DEFINE GRID SECTION REPORTING FOR LGRS

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 ignored by OPM Flow and has no effect on the simulation, but is documented here for completeness.

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")[54] and [55] LGRs.	
….		….	N/A

Notes:

1) The keyword is terminated by "/".

Table 6.107: RPTGRIDL Keyword Description

[54] *Heinemann, Z.E. and Brand, C.W. 1988. Gridding Techniques in Reservoir Simulation. Proc., First Intl. Forum on Reservoir Simulation, Alpbach, Austria, 339.*

[55] *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*

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                  /
```

6.3.169 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 ignored by OPM Flow and has no effect on the simulation.

6.3.170 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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).

6.3.171 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[56] 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 l_x, l_y and l_z 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

[56] *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.*

6.3.172 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[57] 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 l_x, l_y and l_z 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

[57] *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.*

6.3.173 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[58] 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 l_x, l_y and l_z 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

[58] *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.*

6.3.174 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[59] 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 l_x, l_y and l_z 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

[59] *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.*

6.3.175 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 (I_{host}, J_{host}, K_{host}) in the host base grid, multiplies the transmissibility all the cells (I_{auto}, J_{auto}, K_{auto}) and ($I+I_{auto}$, J_{auto}, K_{auto}) 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/".

Table 6.108: SMULTX Keyword Description

See also the MULTX, MULTY and MULTZ keywords for scaling transmissible between grid cells.

Examples

```
--
--       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.

6.3.176 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 (I_{host}, J_{host}, K_{host}) in the host base grid, multiplies the transmissibility all the cells (I_{auto}, J_{auto}, K_{auto}) and (I_{auto}, $J+I_{auto}$, K_{auto}) 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/".

Table 6.109: SMULTY Keyword Description

See also the MULTX, MULTY and MULTZ keywords for scaling transmissible between grid cells.

Examples

```
--
--       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.

6.3.177 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 (I_{host}, J_{host}, K_{host}) in the host base grid, multiplies the transmissibility all the cells (I_{auto}, J_{auto}, K_{auto}) and (I_{auto}, J_{auto}, $K+I_{auto}$) 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/".

Table 6.110: SMULTX Keyword Description

See also the MULTX, MULTY and MULTZ keywords for scaling transmissible between grid cells.

Examples

```
--
--       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.

6.3.178 SOLVDIRS – DEFINE LINEAR SOLVER PRINCIPAL DIRECTIONS

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 ignored by OPM Flow and has no effect on the simulation.

See section *2.2 Running OPM Flow 2019-10 From The Command Line* on how to invoke various numerical schemes via the OPM Flow command line interface.

6.3.179 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")[60] and [61] 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 ignored by OPM Flow and has no effect on the simulation.

[60] *Heinemann, Z.E. and Brand, C.W. 1988. Gridding Techniques in Reservoir Simulation. Proc., First Intl. Forum on Reservoir Simulation, Alpbach, Austria, 339.*

[61] *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*

6.3.180 SPECGRID- DEFINE THE DIMENSIONS OF A CORNER-POINT GRID

Description

SPECGRID defines the dimensions of a corner-point grid in the x, y, and z planes as well as the number of reservoirs, where each reservoir has it's own set of corner-point geometry data.

The keyword can only be used with Irregular Corner-Point 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	F
Notes: 1) The keyword is terminated by "/". 2) The dimensions are also entered on the DIMENS section in the RUNSPEC section and the two sets of numbers should be consistent.			

Table 6.111: 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.

6.3.181 SWATINIT – DEFINE THE INITIAL WATER SATURATION ARRAY FOR CAPILLARY PRESSURE SCALING

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 PPCMAX 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 "/".

Table 6.112: SWATINIT Keyword Description

See also the PPCMAX 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.

6.3.182 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 users 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 "/".

Table 6.113:THCGAS Keyword Description

The THCGAS data is used in conjunction with the other thermal conductivity arrays to calculate the porosity weighted thermal conductivity of a grid block using:

$$\text{Average Thermal Conductivity} = \frac{PORO \times (THCOIL + THCGAS + THCWATER + THCSOLID)}{\text{NUMBER OF PHASES IN THE MODEL}} \times (1 - PORO) \times THCROCK \quad \textbf{(6.19)}$$

See also the THCOIL, THCWATER, THROCK and THCSOLID keywords in the GRID section

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.

6.3.183 THCOIL – DEFINE OIL PHASE THERMAL CONDUCTIVITY FOR ALL CELLS

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 users 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 "/".

Table 6.114:THCOIL Keyword Description

The THCOIL data is used in conjunction with the other thermal conductivity arrays to calculate the porosity weighted thermal conductivity of a grid block using:

$$\text{Average Thermal Conductivity} = \frac{PORO \times (THCOIL + THCGAS + THCWATER + THCSOLID)}{\text{NUMBER OF PHASES IN THE MODEL}} \times (1 - \text{PORO}) \times \text{THCROCK} \quad \textbf{(6.20)}$$

See also the THCGAS, THCWATER, THROCK and THCSOLID keywords in the GRID section

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.

6.3.184 THCONR – DEFINE ROCK AND FLUID THERMAL CONDUCTIVITY FOR ALL CELLS

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.

Note that the THCROCK and THCONR keywords are mutually exclusive.

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 users 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 "/".

Table 6.115:THCONR Keyword Description

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.

6.3.185 THCONSF – DEFINE GAS SATURATION DEPENDENT THERMAL CONDUCTIVITY SCALING FACTOR FOR ALL CELLS

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.

Note that the THCROCK and THCONR keywords are mutually exclusive.

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 users 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 "/".

Table 6.116:THCROCK Keyword Description

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 THCORNR 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 THCSOLID 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 THCSOLID keywords. Secondly, the solid phase is not supported by OPM Flow and therefore neither is the THCSOLID keyword.

Example

```
--
--       DEFINE GRID SGAS DEPENDENT SCALING FACTOR FOR THE THCONR ARRAY
–        FOR ALL CELLS (BASED ON NX x NY x NZ = 300)
--
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.

6.3.186 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.

Note that the THCROCK and THCONR keywords are mutually exclusive.

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 users 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 "/".					

Table 6.117:THCROCK Keyword Description

The THCROCK data is used in conjunction with the other thermal conductivity arrays to calculate the porosity weighted thermal conductivity of a grid block using:

$$\text{Average Thermal Conductivity} = \frac{PORO \times (THCOIL + THCGAS + THCWATER + THCSOLID)}{\text{NUMBER OF PHASES IN THE MODEL}} \times (1 - PORO) \times THCROCK \quad \textbf{(6.22)}$$

See also the THCGAS, THCOIL, THCWATER and THCSOLID keywords in the GRID section

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.

6.3.187 THCSOLID – DEFINE SOLID PHASE THERMAL CONDUCTIVITY FOR ALL CELLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The THCSOLID keyword defines the solid phase thermal conductivity for when the thermal calculation is activated by the THERMAL keyword in the RUNSPEC section in the commercial simulator. THCSOLID 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 in the commercial simulator.

OPM Flow does not recognize this keyword and the keyword may cause the program to terminate, but it is documented here for completeness.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	THCSOLID	THCSOLID is an array of real positive numbers that define the thermal conductivity of the solid 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 "/".

Table 6.118:THCSOLID Keyword Description

The THSOLID data is used in conjunction with the other thermal conductivity arrays to calculate the porosity weighted thermal conductivity of a grid block using:

$$\text{Average Thermal Conductivity} = \frac{PORO \times (THCOIL + THCGAS + THCWATER + THCSOLID)}{\text{NUMBER OF PHASES IN THE MODEL}} \times (1 - PORO) \times THCROCK \qquad \textbf{(6.23)}$$

See also the THCGAS, THCOIL, THCWATER and THCROCK keywords in the GRID section.

Example

```
--
--       DEFINE GRID BLOCK RESERVOIR SOLID PHASE THERMAL CONDUCTIVITY
-        FOR ALL CELLS (BASED ON NX x NY x NZ = 300)
--
THSOLID
         300*20.0                                                    /
```

The above example defines the solid 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.

6.3.188 THCWATER – DEFINE WATER PHASE THERMAL CONDUCTIVITY FOR ALL CELLS

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 users 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 "/".

Table 6.119:THCWATER Keyword Description

The THCWATER data is used in conjunction with the other thermal conductivity arrays to calculate the porosity weighted thermal conductivity of a grid block using:

$$\text{Average Thermal Conductivity} = \frac{PORO \times (THCOIL + THCGAS + THCWATER + THCSOLID)}{\text{NUMBER OF PHASES IN THE MODEL}} \times (1 - PORO) \times THCROCK \quad \textbf{(6.24)}$$

See also the THCGAS, THCOIL, THROCK and THCSOLID keywords in the GRID section

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.

6.3.189 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 6.120:THPRESFT Keyword Description

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness, as it is expected to included in a future release.

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                                                 /
/
```

6.3.190 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 "/".					

Table 6.121: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 ALL LAYERS (BASED ON NX = 5, NY = 5, 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                                                                     /
```

The next 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 (BASED ON NX = 5, NY = 5, NZ = 3)
--
TOPS
 25*3100                                                                    /
--
--  DEFINE GRID BLOCK X DIRECTION CELL SIZE (BASED ON NX = 5)
--
DZV
  3*5.0                                                                     /
--
--  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                                                                     /
```

6.3.191 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	I1	A positive integer that defines the LGR grid block in the I-direction to be joined 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 to be joined 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 to be joined 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 to be joined 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 to be joined 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 to be joined 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 "/".

Table 6.122:TRANGL Keyword Description

Example

```
--
--       MANUALLY DEFINE LGR-GLOBAL GRID NON-NEIGHBOR CONNECTIONS
--
--       ----LGR-----    ---GLOBAL----    -- TRANSNCC --
--       I1   J1   K1     I2   J2   K2
TANGL
         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.

6.3.192 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 ignored by OPM Flow and has no effect on the simulation.

6.3.193 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 ignored by OPM Flow and has no effect on the simulation.

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

Example

```
--
--       ACTIVATE FLUX BOUNDARY MODEL WITHOUT A FLUX FILE
--
USEFLUX
/

USENOFLO
```

The above example activates the Flux Boundary model without a FLUX file.

6.3.194 VEDEBUG – VERTICAL EQUILIBRIUM DEBUG DATA OUTPUT

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.

The keyword is ignored by OPM Flow and has no effect on the simulation.

6.3.195 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 ignored by OPM Flow and has no effect on the simulation.

6.3.196 ZCORN – DEFINE THE DEPTH OF EACH CORNER-POINT OF A GRID BLOCK

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 "/".

Table 6.123: 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
  1450   1500   1500   1550   1550   1600 # top of layer 1
  1500   1550   1550   1600   1600   1650
  1500   1550   1550   1600   1600   1650
  1550   1600   1600   1650   1650   1700
  1460   1510   1510   1560   1560   1610 # bottom of layer 1
  1510   1560   1560   1610   1610   1660
  1510   1560   1560   1610   1610   1660
  1560   1610   1610   1660   1660   1710
  1460   1510   1510   1560   1560   1610 # top of layer 2
  1510   1560   1560   1610   1610   1660
  1510   1560   1560   1610   1610   1660
  1560   1610   1610   1660   1660   1710
  1470   1520   1520   1570   1570   1620 # bottom of layer 2
  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.

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 indicitive 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[62] field is shown in Figure 7.1 and Figure 7.2 illustrates the model's transmissibility in the x-direction (TRANX).

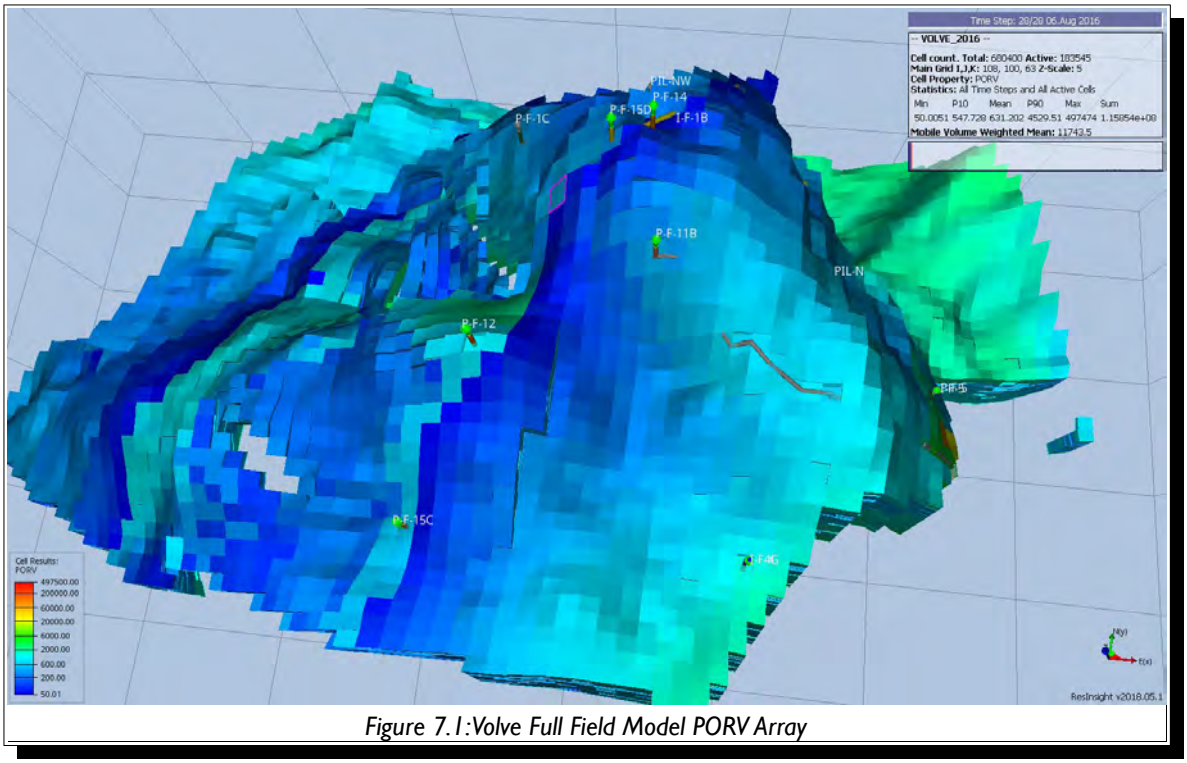


Figure 7.1:Volve Full Field Model PORV Array

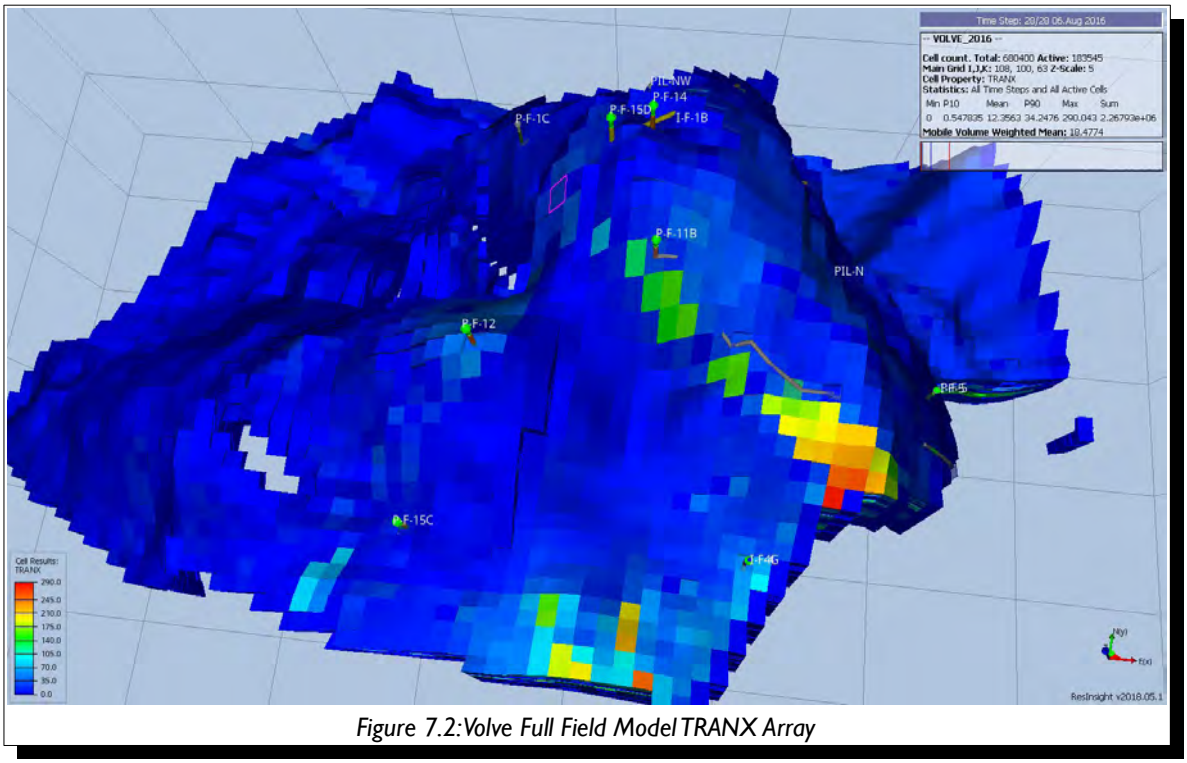


Figure 7.2:Volve Full Field Model TRANX Array

[62] *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 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 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.

7.3.6 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.

7.3.7 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 "/".					

Table 7.2: Depth Keyword Description

See also the TOPS keyword to define the top structural depth for the cells.

Examples

The example below defines the DEPTH 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.

```
--    ---------- BOX ---------
--    I1  I2    J1  J2    K1  K2
BOX
      1  10    11  11    20  20 / SET BOX AREA TO BE MODIFIED
/
--  DEFINE GRID BLOCK X DIRECTION CELL SIZE (BASED ON NX = 5)
--
DEPTH
  10*3500.0                                                              /
--
--    ---------- BOX ---------
--    I1  I2    J1  J2    K1  K2
BOX
      1*  1*    1*  1*    1*  1* / RESET BOX DEFAULTS
/
```

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
/
```

7.3.8 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 ignored by OPM Flow and has no effect on the simulation.

7.3.9 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 ignored by OPM Flow and has no effect on the simulation.

7.3.10 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 ignored by OPM Flow and has no effect on the simulation.

7.3.11 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 ignored by OPM Flow and has no effect on the simulation.

7.3.12 DIFFZ – DEFINE GRID BLOCK Z-DIRECTION DIFFUSIVITY VALUES

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 ignored by OPM Flow and has no effect on the simulation.

7.3.13 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.

7.3.14 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 T_{old} and the multiplier is C, then the resulting transmissibility, T_{new}, 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 an error will occur.

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	I1	A positive integer that defines the first grid block in the I-direction to be joined in a non-neighbor connection, must be greater than or equal to one and less than or equal to NX on the DIMENS in the RUNSPEC section.			None
2	J1	A positive integer that defines the first grid block in the J-direction to be joined in a non-neighbor connection, must be greater than or equal to one and less than or equal to NY on the DIMENS in the RUNSPEC section.			None
3	K1	A positive integer that defines the first grid block in the K-direction to be joined in a non-neighbor connection, must be greater than or equal to one and less than or equal to NZ on the DIMENS in the RUNSPEC section.			None
4	I2	A positive integer that defines the second grid block in the I-direction to be joined in a non-neighbor connection, must be greater than or equal to one and less than or equal to NX on the DIMENS in the RUNSPEC section.			None
5	J2	A positive integer that defines the second grid block in the J-direction to be joined in a non-neighbor connection, must be greater than or equal to one and less than or equal to NY on the DIMENS in the RUNSPEC section.			None
6	K2	A positive integer that defines the second grid block in the K-direction to be joined in a non-neighbor connection, must be greater than or equal to one and less than or equal to NZ on the DIMENS 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 vale 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
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).			
		dimensionless	dimensionless	dimensionless	0.0
Notes: 1) Each record must be terminated by a "/" and the keyword is terminated by "/".					

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 multipliers 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.

7.3.15 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 an error will occur. See also the EDITNNC keyword in the EDIT section that scales an existing NNC.

OPM Flow does not recognize this keyword and the keyword may cause the program to terminate.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	I1	A positive integer that defines the first grid block in the I-direction to be joined in a non-neighbor connection, must be greater than or equal to one and less than or equal to NX on the DIMENS in the RUNSPEC section.			None
2	J1	A positive integer that defines the first grid block in the J-direction to be joined in a non-neighbor connection, must be greater than or equal to one and less than or equal to NY on the DIMENS in the RUNSPEC section.			None
3	K1	A positive integer that defines the first grid block in the K-direction to be joined in a non-neighbor connection, must be greater than or equal to one and less than or equal to NZ on the DIMENS in the RUNSPEC section.			None
4	I2	A positive integer that defines the second grid block in the I-direction to be joined in a non-neighbor connection, must be greater than or equal to one and less than or equal to NX on the DIMENS in the RUNSPEC section.			None
5	J2	A positive integer that defines the second grid block in the J-direction to be joined in a non-neighbor connection, must be greater than or equal to one and less than or equal to NY on the DIMENS in the RUNSPEC section.			None
6	K2	A positive integer that defines the second grid block in the K-direction to be joined in a non-neighbor connection, must be greater than or equal to one and less than or equal to NZ on the DIMENS 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 "/".					

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     / 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 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.

7.3.16 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.

7.3.17 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.

7.3.18 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.

7.3.19 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.

7.3.20 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.

7.3.21 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

7.3.22 HMMULT – HISTORY MATCH GRID TRANSMISSIBILITY & PORE VOLUME GRADIENT CUMULATIVE MULTIPLIERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The HMUMLT 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 "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 ignored by OPM Flow and has no effect on the simulation.

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	HMUMLTZ
PV	MULTPV	HMULTPV	MULTPV	HMULTPV

Table 7.5: HMMULT Keyword List

See also the HMMULT keyword in the GRID section.

7.3.23 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-processoring 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.

7.3.24 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.

7.3.25 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.

7.3.26 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.

7.3.27 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.

7.3.28 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.

7.3.29 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.

7.3.30 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.

7.3.31 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.

7.3.32 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.

7.3.33 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.

7.3.34 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.

7.3.35 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.

7.3.36 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.

7.3.37 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.

7.3.38 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.

7.3.39 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+I, J, K).

See *MULTX - Multiply Cell Transmissibility in the +X Direction* in the GRID section for a full description.

7.3.40 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.

7.3.41 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.

7.3.42 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.

7.3.43 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.

7.3.44 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 (1, J, K-1) and (I, J, K).

See *MULTZ- - Multiply Cell Transmissibility in the -Z Direction* in the GRID section for a full description.

7.3.45 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.

7.3.46 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.

7.3.47 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 "/".

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.

7.3.48 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.

7.3.49 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/".

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.

7.3.50 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/".

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 TRANR+ TRANSMISSIBILITY
--
TRANR
         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.

7.3.51 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 "/".

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.

7.3.52 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.			None
		cP.rb/day/psia	cP.rm^3/day/bars	cP.rcc/hr/atm	

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 "/".

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.

7.3.53 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 "/".

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.

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.

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.

8.2.1 FLUID PROPERTY TABLES

Table 8.1 outlines the various fluid types that can be active in the model, together with the associated RUNSPEC section keywords that activate the phases, versus the PVT keywords that can be used to define the PVT behavior.

Fluid Property Keywords Versus Fluid Type							
Item	**Oil**		**Gas**		**Water**	**Polymer** **Solvent**	
Fluid Type	Dead Oil	Live Oil	Dry Gas	Wet Gas	Water	Polymer	Solvent
RUNSPEC Keywords	OIL	OIL DISGAS	GAS	GAS VAPOIL	WATER	POLYMER	SOLVENT
Pressure Dependent PVT	PVCDO PVDO	PVCO PVTO	PVDG PVZG	PVTG	PVTW		PVDS
Pressure Independent PVT	RSCONST RSCONSTT		RVCONST RVCONSTT				
Surface Density	DENSITY GRAVITY						SDENSITY
Polymer						PLYADS PLYROCK PLYVISC PLMIXPAR PLYMAX PLYSHLOG	

Fluid Property Keywords Versus Fluid Type							
Item	**Oil**		**Gas**		**Water**	**Polymer** **Solvent**	
Fluid Type	Dead Oil	Live Oil	Dry Gas	Wet Gas	Water	Polymer	Solvent

Notes:

1) Currently the GRAVITY, PVZG, RSCONST, RSCONSTT, RVCONST, and RVCONSTT keywords are not supported in OPM Flow.
2) When two keywords are stated for a given fluid type then either one can be used to define the PVT behavior for the given phase.
3) 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.
4) 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.

Table 8.1: Fluid Property Keywords versus Fluid Type

In addition to the above the ROCK keyword should be used to define the rock compressibility.

Typical live oil and dry gas PVT data is from the Volve[63] field is shown in Figure 8.1 and Figure 8.2, respectively.

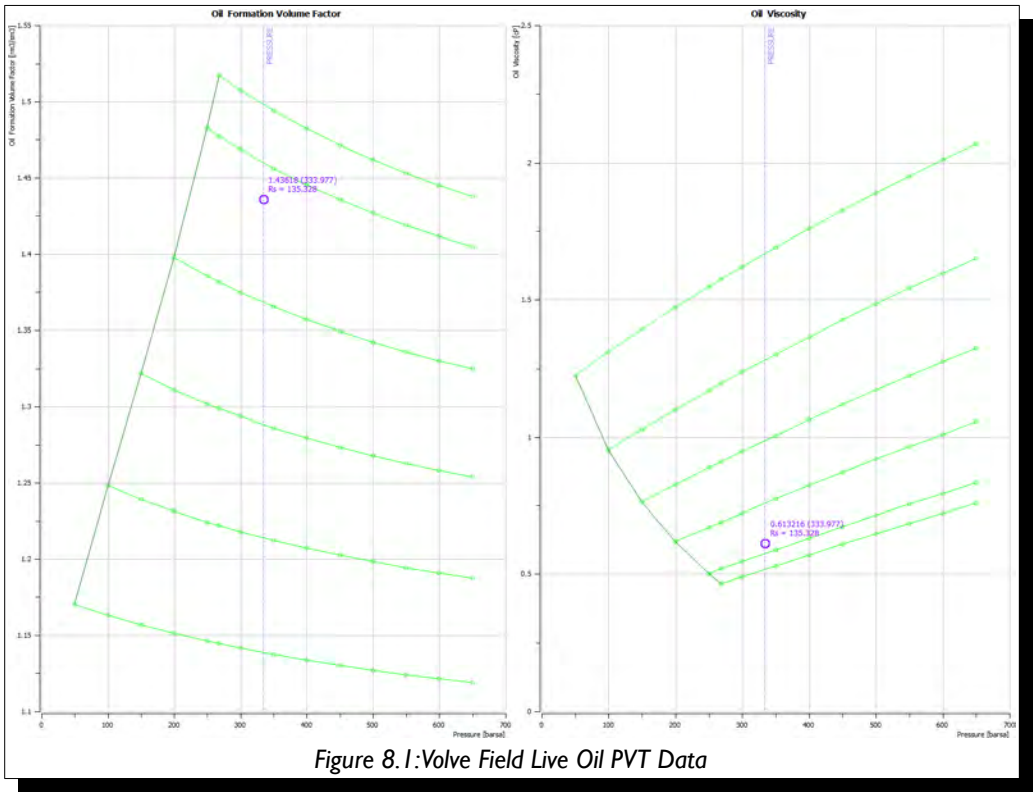


Figure 8.1: Volve Field Live Oil PVT Data

[63] *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.*

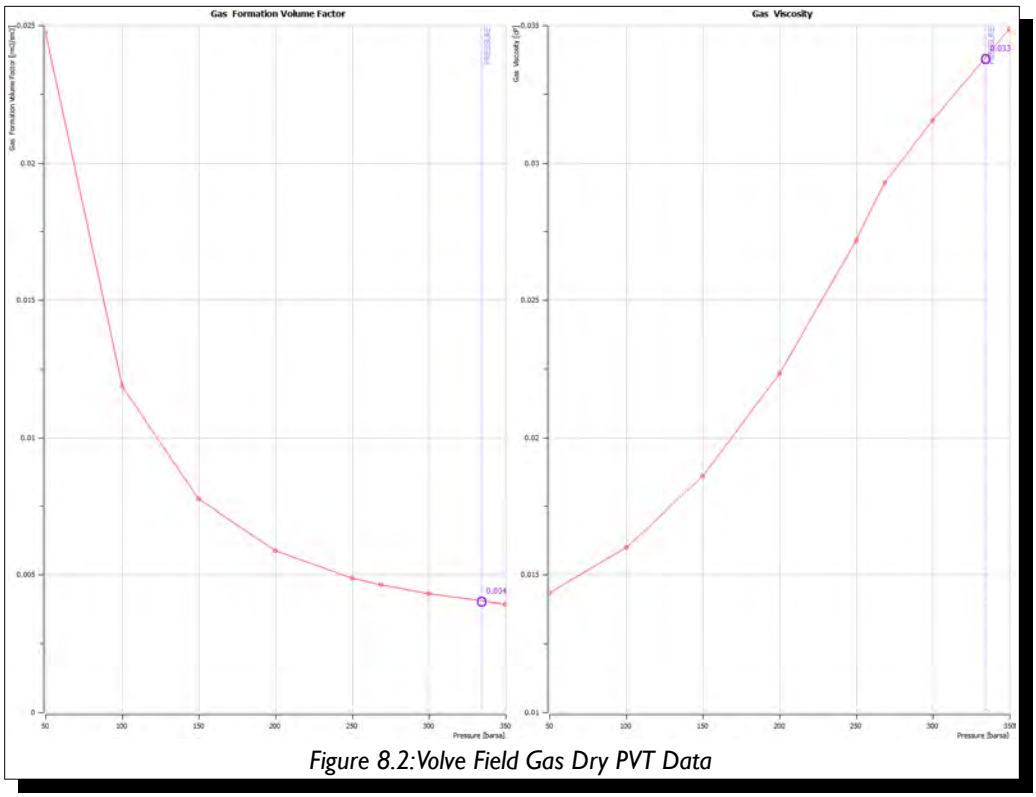


Figure 8.2:Volve Field Gas Dry PVT Data

8.2.2 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.

A typical oil-water relative permeability set of curves is shown in Figure 8.3 indicating the oil end-point data (KRO, KRORW and (1 – SOWCR)) and the water end-point data (KRWR, KRW. SWL and SWCR).

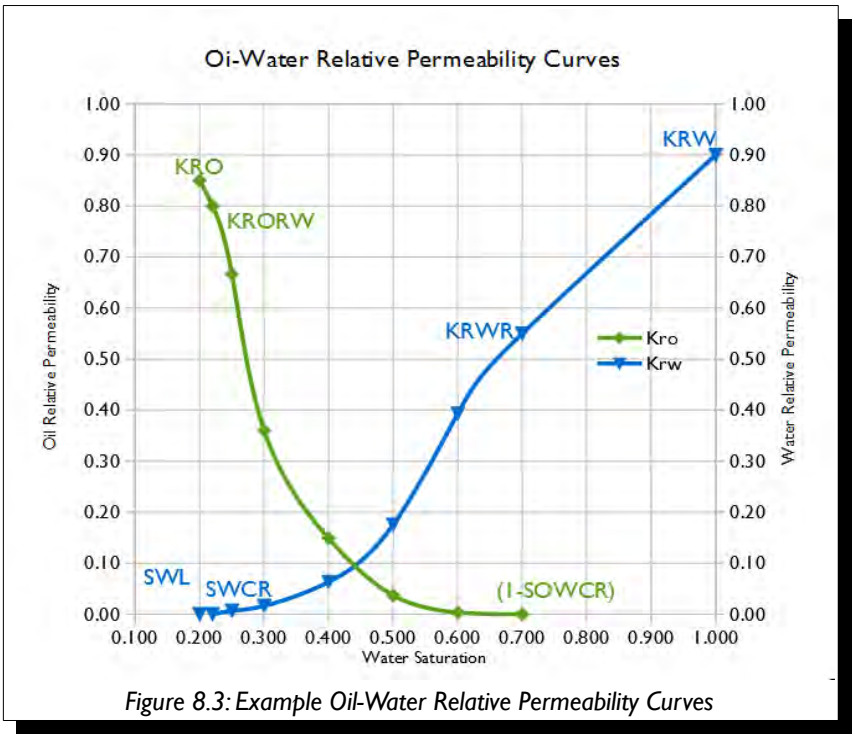


Figure 8.3: Example Oil-Water Relative Permeability Curves

The associated oil-water end-point definitions are outlined in the following table:

Type	End-Point Keyword	Oil-Water End-Point Definitions
Saturation	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.

Table 8.2: Oil-Water Relative Permeability End-Point Data Definitions

Similarly for gas-oil systems, Figure 8.4 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).

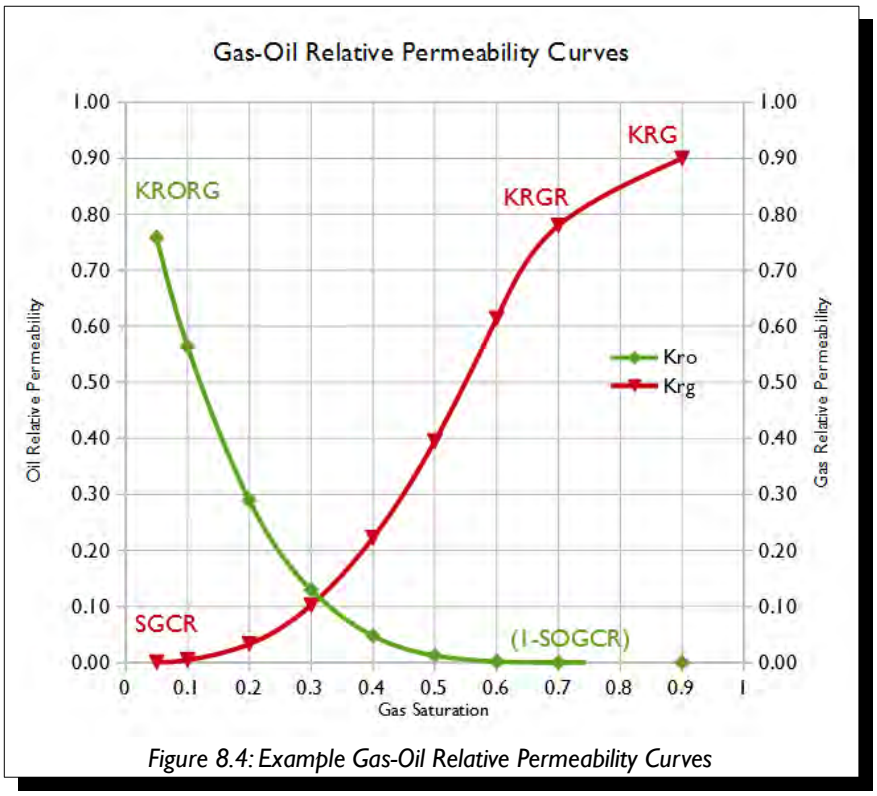


Figure 8.4: Example Gas-Oil Relative Permeability Curves

The gas-oil end-point definitions are outlined in the following table:

Type	End-Point Keyword	Gas-Oil End-Point Definitions
Saturation	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.
Capillary Pressure	SGLPC	Capillary pressure connate gas saturation, that is the smallest gas saturation in a gas saturation function table.

Table 8.3: Gas-Oil 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 8.2 and Table 8.3 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.

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			SGFN[1]		Pcog	
SLGOF	Pcog			SGWFN		Pcgw	
SWOF	Pcwo		Pcwo	SOF2[2]	No P_c		
				SOF3[3]	No P_c		
				SOF32D	No P_c		
				SWFN			Pcwo

Notes:

1) 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.
2) The SOF2 defines the relative permeability in oil-gas and oil-water runs only, and the miscible hydrocarbon in SOVENT runs. This keyword should not be used to define the oil relative permeability when oil, gas and water are present.
3) Defines oil in relative with respect to water and oil relative permeability with respect gas.

Table 8.4: 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.

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/sm^3	gm/scc	None
Notes: 1) Each data set must be terminated by a "/" including the last data set.					

Table 8.5: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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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[64] 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.

OPM Flow does not recognize this keyword and the keyword may cause the program to terminate, but it is documented here for completeness.

[64] *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*

8.3.1 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

8.3.2 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

8.3.3 ALPOLDS – 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

8.3.4 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

8.3.5 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

8.3.6 APIGROUP – DEFINE API TRACKING NUMBER OF GROUPED OIL PVT TABLES

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 ignored by OPM Flow and has no effect on the simulation

8.3.7 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

8.3.8 AQUTAB – DEFINE CARTER-TRACY AQUIFER INFLUENCE FUNCTIONS

Description

The AQUTAB keyword defines additional Carter-Tracy[65] aquifer functions to be used in the model. Carter-Tracy representation of the aquifer influx is via a q_w term in the nonlinear 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[66], 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 formulas 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 "/" and there is no "/" terminator for the keyword.

Table 8.6: 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 8.7 to Table 8.10 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[67] table 38-6 on page 38-12. In the tables r_D is defined as the ratio of the aquifer external radius divided by hydrocarbon radius, that is: $r_D = \frac{r_e}{r_o}$.

[65] *Carter, R. D., and Tracy, G.W. "An Improved Method for Calculating Water Influx." Transactions of AIME, Vol. 219 (1060), pp 415-417.*

[66] *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.*

[67] *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 8.7: Carter-Tracy Aquifer Influence Functions (R_D =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 8.8: Carter-Tracy Aquifer Influence Functions (R_D =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 8.9: Carter-Tracy Aquifer Influence Functions (R_D =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 8.10: Carter-Tracy Aquifer Influence Functions (R_D =10 and Finite Linear)

For the finite linear Carter-Tracy influence function in Table 8.10 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 8.11 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 and is taken from Bradely[68] table 38-3 on page 38-6.

Carter-Tracy Infinite Radial Aquifer Influence Function (Default)					
No.	**Infinite Dimensionless**		**No.**	**Infinite Dimensionless**	
	t_D	p_D		t_D	p_D
1	1.0×10^{-2}	0.112	19	4.0	1.275
2	5.0×10^{-2}	0.229	20	5.0	1.362
3	1.0×10^{-1}	0.315	21	6.0	1.436
4	1.5×10^{-1}	0.376	22	7.0	1.500
5	2.0×10^{-1}	0.424	23	8.0	1.556
6	2.5×10^{-1}	0.469	24	9.0	1.604
7	3.0×10^{-1}	0.503	25	10.0	1.651
8	4.0×10^{-1}	0.564	26	15.0	1.829
9	5.0×10^{-1}	0.616	27	20.0	1.960
10	6.0×10^{-1}	0.659	28	25.0	2.067
11	7.0×10^{-1}	0.702	29	30.0	2.147
12	8.0×10^{-1}	0.735	30	40.0	2.282
13	9.0×10^{-1}	0.772	31	50.0	2.388
14	1.0	0.802	32	60.0	2.476
15	1.5	0.927	33	70.0	2.550
16	2.0	1.020	34	80.0	2.615
17	2.5	1.101	35	90.0	2.672
18	3.0	1.169	36	100.0	2.723

Table 8.11: Carter-Tracy Infinite Radial Aquifer Influence Function (Default)

For an overview of analytical aquifers see Dake[69].

[68] *Bradley Howard B., et. al,, Petroleum Engineering Handbook, Society of Petroleum Engineers (June 1989), ISBN:9781555630102.*

[69] *Dake, L.P. Fundamentals of Reservoir Engineering, Amsterdam, The Netherlands, Elsevier Science BV (1978) Chapter 9.*

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
          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.

8.3.9 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.

The keyword is used in conjunction with the PVTSALT 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.

OPM Flow does not recognize this keyword and the keyword may cause the program to terminate, but it is documented here for completeness.

8.3.10 BGGI - DEFINE GI GAS FORMATION VOLUME FACTOR PRESSURE TABLES

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.

OPM Flow does not recognize this keyword and the keyword may cause the program to terminate, but it is documented here for completeness.

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.[70] 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.

[70] *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.*

8.3.11 BOGI - DEFINE GI OIL FORMATION VOLUME FACTOR PRESSURE TABLES

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.

OPM Flow does not recognize this keyword and the keyword may cause the program to terminate, but it is documented here for completeness.

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.[71] 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.

[71] *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.*

8.3.12 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.

8.3.13 COALADS – DEFINE GAS AND SOLVENT RELATIVE ADSORPTION TABLES

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 ignored by OPM Flow and has no effect on the simulation

8.3.14 COALPP – DEFINE GAS AND SOLVENT PARTIAL PRESSURE ADSORPTION TABLES

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 ignored by OPM Flow and has no effect on the simulation

8.3.15 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.

8.3.16 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.

8.3.17 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.

8.3.18 DENSITY – DEFINE THE SURFACE OIL, WATER GAS DENSITIES FOR THE FLUIDS

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 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	OILDEN	OILDEN is a real number defining the density of oil at surface conditions.			
		lb/ft^3	kg/m^3	gm/cc	
		600	600	600	Defined
2	WATDEN	WATDEN is a real number defining the density of water at surface conditions.			
		lb/ft^3	kg/m^3	gm/cc	
		999.014	999.014	999.014	Defined
3	GASDEN	GASDEN is a real number defining the density of gas at surface conditions.			
		lb/ft^3	kg/m^3	gm/cc	
		1.000	1.000	1.000	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 "/" and there is no "/" terminator for the keyword.					

Table 8.12: DENSITY Keyword Description

According to the SPE SI standard[72], ***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.

[72] *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
```

There is no terminating "/" for this keyword.

8.3.19 DEPTHTAB – RIVER TIME AND DEPTH TABLES

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 ignored by OPM Flow and has no effect on the simulation.

8.3.20 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 ignored by OPM Flow and has no effect on the simulation.

8.3.21 DIFFC – DEFINE PVT REGION MOLECULAR DIFFUSION TABLES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The DIFFC keyword defines the molecular diffusion tables for each PVT region for when the Diffusion option has been activated by the DIFFUSE keyword in the RUNSPEC section.

This keyword is ignored by OPM Flow and has no effect on the simulation.

8.3.22 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 ignored by OPM Flow and has no effect on the simulation.

8.3.23 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 ignored by OPM Flow and has no effect on the simulation.

8.3.24 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 ignored by OPM Flow and has no effect on the simulation.

8.3.25 DISPERSE– DEFINE DISPERSION TABLES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, DISPERSE, defines the dispersion tables for when the Dispersion option has been activated via declaring the dimensions of the DISPERSE tables using the DISPDIMS keyword and activating the Tracer option via the TRACERS keyword.

This keyword is ignored by OPM Flow and has no effect on the simulation.

8.3.26 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 ignored by OPM Flow and has no effect on the simulation.

8.3.27 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 ignored by OPM Flow and has no effect on the simulation.

8.3.28 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[73] and Killough[74] models are available.

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. This option is ignored by OPM Flow.	0.1
2	HYSTMOD	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:	0

Water Wet Hysteresis Models		
HYSMOD	Non-Wetting Phases	Wetting Phase
-1	Equilibration option for equilibrating the model with the SATNUM (drainage curves) and running the model with imbibition curves (IMBNUM). This option implies no hysteresis.	
0	Carlson Hysteresis Model	SATNUM
1	Carlson Hysteresis Model	IMBNUM
2	Killough Hysteresis Model	SATNUM
3	Killough Hysteresis Model	IMBNUM
4	Killough Hysteresis Model	Killough Hysteresis Model
Oil Wet to Water Wet Water Wet		
5	Carlson Non-Wetting Modeling for Gas and Water	SATNUM
6	Killough Non-Wetting Modeling for Gas and Water	SATNUM
7	Killough Non-Wetting Modeling for Gas and Water	Killough Non-Wetting Modeling for the Wetting Oil Phase
Note only the default value of zero is supported by OPM Flow.		

[73] *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).*

[74] *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 users the Killough capillary pressure model. Note only the default value of BOTH is 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
9		Not used	
10		Not used	

No.	Name	Description	Default
11		Not used	
12		Not used	
13		Not used	

Notes:

1) The keyword is terminated by "/".

Table 8.13: 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.

8.3.29 EHYSTRR – DEFINE HYSTERESIS MODEL AND PARAMETERS VIA SATNUM

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[75] model is available for this keyword and the keyword is optional.

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness. 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 8.14: 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.

[75] *Killough, J. E. "Reservoir Simulation with History-dependent Saturation Functions," paper SPE 5106, Society of Petroleum Engineers Journal (1976) 16, No. 1, 37-48.*

8.3.30 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.

8.3.31 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.

8.3.32 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 ignored by OPM Flow and has no effect on the simulation.

8.3.33 ENPCVD – DEFINE MAXIMUM CAPILLARY PRESSURE VERSUS DEPTH FUNCTIONS

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 ignored by OPM Flow and has no effect on the simulation.

8.3.34 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 ignored by OPM Flow and has no effect on the simulation.

8.3.35 ENSPCVD – DEFINE CAPILLARY PRESSURE END-POINTS VERSUS DEPTH

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 ignored by OPM Flow and has no effect on the simulation.

8.3.36 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 ignored by OPM Flow and has no effect on the simulation.

8.3.37 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 ignored by OPM Flow and has no effect on the simulation.

8.3.38 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.

8.3.39 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.

8.3.40 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 ignored by OPM Flow and has no effect on the simulation.

8.3.41 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 ignored by OPM Flow and has no effect on the simulation.

8.3.42 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.

8.3.43 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.

8.3.44 FOAMADS - DEFINE FOAM ROCK ADSORPTION TABLES

Description

The FOAMPADS keyword defines the foam rock adsorption tables 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	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. 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 increasing down the column values that defines the mass of adsorbed foam per unit mass of rock of the saturated concentration of foam adsorbed by the rock for a given FOAMCON. The first table data set entry should be zero to define a no foam concentration data set. Each FOAMCON/FOAMRATI data set 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 "/" and there is no "/" terminator for the keyword.

Table 8.15: 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.
There is no terminating "/" for this keyword.

8.3.45 FOAMDCYO – DEFINE FOAM DECAY VERSUS OIL SATURATION TABLES

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 ignored by OPM Flow and has no effect on the simulation.

8.3.46 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 ignored by OPM Flow and has no effect on the simulation.

8.3.47 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 ignored by OPM Flow and has no effect on the simulation.

8.3.48 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 ignored by OPM Flow and has no effect on the simulation.

8.3.49 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.

The keyword is recognized by the input deck parser but is ignored by OPM Flow and has no effect on the simulation.

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 (e_s in equation (8.1). Note if e_s is less than one then the slope of F_s in equation (8.1) 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 C_s^r is also small. If this is the case use MINSURF one 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 1×10^{-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 for which foam has no effect. The default value of 1×10^{-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.			Defined

No.	Name	Description			Default
		Field	Metric	Laboratory	
		dimensionless 1 x 10^{-6}	dimensionless 1 x 10^{-6}	dimensionless 1 x 10^{-6}	

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 "/" and there is no "/" terminator for the keyword.

Table 8.16: FOAMFSC Keyword Description

The gas mobility reduction as a function of surfactant concentration is of the form:

$$F_s = \left(\frac{C_s}{C_s^r} \right)^{e_s} \tag{8.1}$$

Where:

F_s = the resulting gas mobility reduction factor as a function of surfactant concentration,
C_s = surfactant concentration,
C^r_s = reference surfactant concentration, that is $C_s < C_s^r$ defines a weak foam and $C_s > C_s^r$ defines a strong foam (FOAMCON), and
e_s = 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 (M_{rf}) is:

$$M_{rf} = \frac{1}{1 + \left(M_r \times F_s \times F_w \times F_o \times F_c \right)} \tag{8.2}$$

Where:

M_{rf} = the reference mobility reduction factor, see the FOAMFRM keyword in the PROPS section,
F_s = gas mobility reduction factor as a function of surfactant concentration, see the FOAMFSC keyword in the PROPS section,
F_w = gas mobility reduction factor as a function of water saturation, see the FOAMFSW keyword in the PROPS section,
F_o = gas mobility reduction factor as a function of oil saturation, see the FOAMFSO keyword in the PROPS section, and
F_c = 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.

8.3.50 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 ignored by OPM Flow and has no effect on the simulation.

8.3.51 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 ignored by OPM Flow and has no effect on the simulation.

8.3.52 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 ignored by OPM Flow and has no effect on the simulation.

8.3.53 FOAMMOB - DEFINE FOAM GAS MOBILITY VERSUS FOAM CONCENTRATION TABLES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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 "/" and there is no "/" terminator for the keyword.

Table 8.17: 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      0.00000
            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.

8.3.54 FOAMMOBP – DEFINE FOAM MOBILITY REDUCTION VERSUS OIL PRESSURE

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 ignored by OPM Flow and has no effect on the simulation.

8.3.55 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 ignored by OPM Flow and has no effect on the simulation.

8.3.56 FOAMOPTS - DEFINE FOAM MODEL OPTIONS

Description

The FOAMOPTS keyword defines the transport phase for the foam (gas or water) and how gas mobility reduction should be calculated 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
1	FOAMOPT1	A defined character that defines the transport phase for the foam, and should be set to one of the following character strings: 1) GAS: for the foam to be transport in the gas phase., or 2) WATER: for the foam to be transported in the water phase. Only the default value of GAS is currently supported by OPM Flow.	GAS
2	FOAMOPT2	A defined character that defines the method to be used to calculated the reduction in gas mobility, and should be set to one of the following character strings: 1) TAB: Sets the reduction in gas mobility to be calculated based on tables using the FOAMMOD keyword as a function of foam concentration, the FOAMMOBS as a function for 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. Note this option is not supported by OPM Flow. Only the default value of TAB is currently supported by OPM Flow.	TAB
Notes: 1) The keyword is terminated by "/".			

Table 8.18: FOAMOPTS Keyword Description

Example

```
--
--        FOAMOPT1  FOAMOPT2
--        PHASE     MOBILITY
--        --------  --------
FOAMOPTS
          GAS       TAB                                  / FOAM MODEL OPTIONS
```

The above example defines the transport phase to be gas and the gas mobility reduction to be use a table as defined by FOAMMOD keyword as a function of foam concentration, the FOAMMOBS as a function for shear, or as a function of pressure using the FOAMMOBP keyword.

8.3.57 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/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.
2) Each table must contain just one row and one row only.
3) Each table is terminated by "/" and there is no "/" terminator for the keyword.

Table 8.19: 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.

8.3.58 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. 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.

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 users 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.			
		$1/°R^2$ 9.26×10^{-7}	$1/°K^2$ 3.0×10^{-6}	$1/°K^2$ 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 "/" and there is no "/" terminator for the keyword.

Table 8.20: GASDENT Keyword Description

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.

8.3.59 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 keywords 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.

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 users 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 "/" and there is no "/" terminator for the keyword.

Table 8.21: 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.

8.3.60 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.

8.3.61 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.

OPM Flow does not recognize this keyword and the keyword may cause the program to terminate, but it is documented here for completeness.

8.3.62 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 ignored by OPM Flow and has no effect on the simulation.

8.3.63 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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	m^2bars/kg	cm^2atm/gm	
		0.00694	0.0000981	0.000968	Defined
Notes: 1) The keyword is terminated by "/".					

Table 8.22: GRAVCONS Keyword Description

Examples

```
--
--        RE-DEFINE GRAVITY CONSTANT
--
GRAVITY
          0.0000980665                                    /
```

The above example re-defines the gravity constant to be 0.0000980665 ft^2psi/lb from the default value of 0.00694 ft^2psi/lb.

8.3.64 GRAVITY– DEFINE THE SURFACE OIL, WATER GAS GRAVITIES FOR THE FLUIDS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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.

This keyword is not supported by OPM Flow but is documented here for completeness; however, the density data can be entered using the DENSITY keyword.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	OILAPI	OILAPI is a real number defining the density of oil 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 density of water at surface conditions.			
		(water =1.0) 0.7773	(water =1.0) 0.7773	(water =1.0) 0.7773	Defined
3	GRAVGAS	GRAVGAS is a real number defining the density of gas 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) The each data set terminated by terminated by "/" at the end of the line, there is no "/" terminator for the keyword.

Table 8.23: GRAVITY Keyword Description

According to the SPE SI standard[76], ***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.

[76] *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 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 DENSITY 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
```

There is not terminating "/" for this keyword.

8.3.65 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 8.24, for example, HASWL.

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.

This keyword is ignored by OPM Flow and has no effect on the simulation.

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 8.24: HA Keyword List

8.3.66 HDISP – DEFINE TRACER MECHANICAL DISPERSIVITY PARAMETERS

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 ignored by OPM Flow and has no effect on the simulation.

8.3.67 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 8.25, for example, HMSWL.

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.

This keyword is ignored by OPM Flow and has no effect on the simulation.

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 8.25: HM Keyword List

8.3.68 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 ignored by OPM Flow and has no effect on the simulation.

8.3.69 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 ignored by OPM Flow and has no effect on the simulation.

8.3.70 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 ignored by OPM Flow and has no effect on the simulation.

8.3.71 HMROCK – HISTORY MATCH ROCK COMPRESSIBILITY GRADIENT PARAMETERS

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 ignored by OPM Flow and has no effect on the simulation.

8.3.72 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 ignored by OPM Flow and has no effect on the simulation.

8.3.73 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 ignored by OPM Flow and has no effect on the simulation.

8.3.74 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 ignored by OPM Flow and has no effect on the simulation.

8.3.75 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 ignored by OPM Flow and has no effect on the simulation.

8.3.76 HWKRORW – END-POINT SCALING OF GRID CELL KRO(SWCR) (HIGH SALINITY AND WATER WET)

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 ignored by OPM Flow and has no effect on the simulation.

8.3.77 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 S_w = 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 ignored by OPM Flow and has no effect on the simulation.

8.3.78 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 S_w = 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 ignored by OPM Flow and has no effect on the simulation.

8.3.79 HWPCW – END-POINT SCALING OF GRID CELL WATER CAPILLARY PRESSURE (HIGH SALINITY AND WATER WET)

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:

$$P_c = P_{c_{TABLE}} \left(\frac{HWPCW}{P_{c_{TABLE-MAX}}} \right) \quad \textbf{(8.3)}$$

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 ignored by OPM Flow and has no effect on the simulation.

8.3.80 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 ignored by OPM Flow and has no effect on the simulation.

8.3.81 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 ignored by OPM Flow and has no effect on the simulation.

8.3.82 HWSWCR – END-POINT SCALING GRID CELL SWCR (HIGH SALINITY AND WATER WET)

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 ignored by OPM Flow and has no effect on the simulation.

8.3.83 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 ignored by OPM Flow and has no effect on the simulation.

8.3.84 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 ignored by OPM Flow and has no effect on the simulation.

8.3.85 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 ignored by OPM Flow and has no effect on the simulation.

8.3.86 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 ignored by OPM Flow and has no effect on the simulation.

8.3.87 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[77] or Killough[78] 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

Example

```
--
--       ACTIVATE CARLSON AND KILLOUGH ALTERNATIVE DRAINAGE HYSTERESIS OPTION
--
HYMOBGDR
```

[77] *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).*

[78] *Killough, J. E. "Reservoir Simulation with History-dependent Saturation Functions," paper SPE 5106, Society of Petroleum Engineers Journal (1976) 16, No. 1, 37-48.*

8.3.88 HYSTCHCK - ACTIVATE HYSTERESIS INHIBITION 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 ignored by OPM Flow and has no effect on the simulation.

8.3.89 IKRG – END-POINT SCALING OF GRID CELL KRG(SGU) (IMBIBITION)

Description

IKRG defines the imbibition scaling parameter at the maximum gas relative permeability value (ISGU), normally ISGU is equal to 1.0 - S_{wc}, 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 "/".

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. The directional form of the keyword is recognized, but is not supported by OPM Flow.

Table 8.26: IKRG Keyword Description

For the two point scaling option and for the IKRGR gas relative permeability array NOT present in the input deck the k_{rg} value for a grid block is scaled by:

$$k_{rg} = k_{rg\,TABLE}\left(\frac{IKRG}{k_{rg\,TABLE-MAX}}\right) \quad (8.4)$$

Where:

k_{rg} = the resulting k_{rg} 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 (S_{wc}).

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 8.27: 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 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*0.555  1000*0.575  1000.0.600               /
--
--       DEFINE END OF INPUT BOX EDITING OF INPUT ARRAYS
--
ENDBOX
```

8.3.90 IKRGR – END-POINT SCALING OF GRID CELL KRGR(1-SOGCR) (IMBIBITION)

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 (S_{wc}), 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 "/".
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. The directional form of the keyword is recognized, but is not supported by OPM Flow.

Table 8.28: 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 8.29: 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
         1000*0.500  1000*0.570  1000.0.580               /
--
--       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'     0.5500       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
/
```

8.3.91 IKRO – END-POINT SCALING OF GRID CELL KRO(SWL) (IMBIBITION)

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 "/".
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. The directional form of the keyword is recognized, but is not supported by OPM Flow.

Table 8.30: 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 k_{ro} value for a grid block is scaled by:

$$k_{ro} = k_{ro_{TABLE}} \left(\frac{IKRO}{k_{ro_{TABLE-MAX}}} \right) \quad \textbf{(8.5)}$$

Where:

k_{ro} = the resulting k_{ro} 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 (S_{wcr}).

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 8.31: 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 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'      0.8550       1*  1*   1*  1*   1   1  / IKRO FOR LAYER 1
   'IKOG'      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
/
```

8.3.92 IKRORG – END-POINT SCALING OF GRID CELL KRO(SGCR) (IMBIBITION)

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 "/".

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. The directional form of the keyword is recognized, but is not supported by OPM Flow.

Table 8.32: 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 8.33: 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 0.750, 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
         1000*0.755  1000*0.775  1000.0.800               /
--
--       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
/
```

8.3.93 IKRORW – END-POINT SCALING OF GRID CELL KRO(SWCR) (IMBIBITION)

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 "/".

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. The directional form of the keyword is recognized, but is not supported by OPM Flow.

Table 8.34: 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 8.35: 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 0.750, 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
         1000*0.755  1000*0.775  1000.0.800               /
--
--       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
/
```

8.3.94 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 oil relative permeability value (ISWU), that is for S_w = 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 "/".
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. The directional form of the keyword is recognized, but is not supported by OPM Flow.

Table 8.36: IKRW Keyword Description

For the two point scaling option and for the IKRWR water relative permeability array NOT present in the input deck the k_{rw} value for a grid block is scaled by:

$$k_{rw} = k_{rw_{TABLE}} \left(\frac{IKRW}{k_{rw_{TABLE-MAX}}} \right) \quad (8.6)$$

Where:

k_{rw} = 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 8.37: 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 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
         1000*0.855  1000*0.875  1000.0.900               /
--
--       DEFINE END OF INPUT BOX EDITING OF INPUT ARRAYS
--
ENDBOX
```

8.3.95 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 (SOWCR), 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 "/".

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. The directional form of the keyword is recognized, but is not supported by OPM Flow.

Table 8.38: 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 8.39: 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 0.750, 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
         1000*0.755  1000*0.775  1000.0.800               /
--
--       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
/
```

8.3.96 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 ignored by OPM Flow and has no effect on the simulation.

8.3.97 IMPCVD – IMBIBITION MAXIMUM CAPILLARY PRESSURE VERSUS DEPTH FUNCTIONS

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 ignored by OPM Flow and has no effect on the simulation.

8.3.98 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-processoring 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.

8.3.99 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 ignored by OPM Flow and has no effect on the simulation.

8.3.100 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 ignored by OPM Flow and has no effect on the simulation.

8.3.101 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 ignored by OPM Flow and has no effect on the simulation.

8.3.102 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 ignored by OPM Flow and has no effect on the simulation.

8.3.103 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[79] 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 ignored by OPM Flow and has no effect on the simulation.

[79] *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).*

8.3.104 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 "/".

Table 8.40: IPCG Keyword Description

The capillary pressure for a grid block is scaled by:

$$P_c = P_{c_{TABLE}} \left(\frac{IPCG}{P_{c_{TABLE-MAX}}} \right) \quad (8.7)$$

Where:

P_c = 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 inhibition capillary pressure table allocated to the grid block.

$P_{c_{TABLE-MAX}}$ = the maximum capillary pressure in the inhibition 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.

8.3.105 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 "/".

Table 8.41: IPCW Keyword Description

The capillary pressure for a grid block is scaled by:

$$P_c = P_{c_{TABLE}} \left(\frac{IPCW}{P_{c_{TABLE-MAX}}} \right) \quad (8.8)$$

Where:

P_c = 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 inhibition capillary pressure table allocated to the grid block.

$P_{c_{TABLE-MAX}}$ = the maximum capillary pressure in the inhibition 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.

8.3.106 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 "/".
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. The directional form of the keyword is recognized, but is not supported by OPM Flow.

Table 8.42: 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 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 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 ALL 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.

8.3.107 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 "/".

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. The directional form of the keyword is recognized, but is not supported by OPM Flow.

Table 8.43: 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.

8.3.108 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	IGLPC 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 IGLPC 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 "/".
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. The directional form of the keyword is recognized, but is not supported by OPM Flow.

Table 8.44: 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.

8.3.109 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 "/".

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. The directional form of the keyword is recognized, but is not supported by OPM Flow.

Table 8.45: 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.

8.3.110 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 "/".
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. The directional form of the keyword is recognized, but is not supported by OPM Flow.

Table 8.46: 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.

8.3.111 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 "/".
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. The directional form of the keyword is recognized, but is not supported by OPM Flow.

Table 8.47: 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 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

8.3.112 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 "/".

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. The directional form of the keyword is recognized, but is not supported by OPM Flow.

Table 8.48: 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.

8.3.113 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 "/".

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. The directional form of the keyword is recognized, but is not supported by OPM Flow.

Table 8.49: 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.

8.3.114 ISWLPC – END-POINT SCALING GRID CELL CAPILLARY PRESSURE CONNATE WATER SATURATIONS (IMBIBITION)

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 "/".
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. The directional form of the keyword is recognized, but is not supported by OPM Flow.

Table 8.50: 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.

8.3.115 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 "/".

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. The directional form of the keyword is recognized, but is not supported by OPM Flow.

Table 8.51: 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.

8.3.116 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 - S_{wc}, 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 "/".

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. The directional form of the keyword is recognized, but is not supported by OPM Flow.

Table 8.52: KRG Keyword Description

For the two point scaling option and for the KRGR gas relative permeability array NOT present in the input deck the k_{rg} value for a grid block is scaled by:

$$k_{rg} = k_{rg\,TABLE}\left(\frac{KRG}{k_{rg\,TABLE-MAX}}\right) \quad (8.9)$$

Where:

k_{rg} = the resulting k_{rg} 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 (S_{wc}).

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 8.53: 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 suffixed with the letter I, for example IKRG, can be used to define the 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 0.550, 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
1000*0.555  1000*0.575  1000.0.600                       /
--
--       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
/
```

8.3.117 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 (S_{wc}), 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 "/".

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. The directional form of the keyword is recognized, but is not supported by OPM Flow.

Table 8.54: 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 8.55: 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 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 KRRG 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
1000*0.500  1000*0.570  1000.0.580                       /
--
--       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'      0.5500       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
/
```

8.3.118 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 "/".
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. The directional form of the keyword is recognized, but is not supported by OPM Flow.

Table 8.56: 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 k_{ro} value for a grid block is scaled by:

$$k_{ro} = k_{ro\,TABLE}\left(\frac{KRO}{k_{ro\,TABLE-MAX}}\right) \quad \textbf{(8.10)}$$

Where:

k_{ro} = the resulting k_{ro} 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 (S_{wcr}).

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 8.57: 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 0.850, 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
1000*0.855  1000*0.875  1000.0.900                       /
--
--       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
/
```

8.3.119 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 "/".

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. The directional form of the keyword is recognized, but is not supported by OPM Flow.

Table 8.58: 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 8.59: 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 0.750, 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
1000*0.755  1000*0.775  1000.0.800                       /
--
--       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
/
```

8.3.120 KRORW – END-POINT SCALING OF GRID CELL KRO(SWCR) (DRAINAGE)

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 "/".

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. The directional form of the keyword is recognized, but is not supported by OPM Flow.

Table 8.60: 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 8.61: 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 0.750, 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
1000*0.755  1000*0.775  1000.0.800                       /
--
--       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
/
```

8.3.121 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 S_w = 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 "/".

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. The directional form of the keyword is recognized, but is not supported by OPM Flow.

Table 8.62: KRW Keyword Description

For the two point scaling option and for the KRWR water relative permeability array NOT present in the input deck the k_{rw} value for a grid block is scaled by:

$$k_{rw} = k_{rw_{TABLE}} \left(\frac{KRW}{k_{rw_{TABLE-MAX}}} \right) \quad (8.11)$$

Where:

k_{rw} = 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 8.63: 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 0.850, 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
1000*0.855  1000*0.875  1000.0.900                       /
--
--       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
/
```

8.3.122 KRWR – END-POINT SCALING OF GRID CELL KRWR(SOWCR) (DRAINAGE)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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 "/".

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. The directional form of the keyword is recognized, but is not supported by OPM Flow.

Table 8.64: 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 8.65: 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 0.750, 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
1000*0.755  1000*0.775  1000.0.800                       /
--
--       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
/
```

8.3.123 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[80] 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 ignored by OPM Flow and has no effect on the simulation.

[80] *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*

8.3.124 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[81] 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 ignored by OPM Flow and has no effect on the simulation.

[81] *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*

8.3.125 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[82] 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 ignored by OPM Flow and has no effect on the simulation.

[82] *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*

8.3.126 LCUNIT – DEFINE LINEAR COMBINATION RATE AND VOLUME UNITS

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 ignored by OPM Flow and has no effect on the simulation.

8.3.127 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 ignored by OPM Flow and has no effect on the simulation.

8.3.128 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 ignored by OPM Flow and has no effect on the simulation.

8.3.129 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 ignored by OPM Flow and has no effect on the simulation.

8.3.130 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 S_w = 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 ignored by OPM Flow and has no effect on the simulation.

8.3.131 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 S_w = 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 ignored by OPM Flow and has no effect on the simulation.

8.3.132 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:

$$P_c = P_{c_{TABLE}} \left(\frac{LPCW}{P_{c_{TABLE-MAX}}} \right) \quad (8.12)$$

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 ignored by OPM Flow and has no effect on the simulation.

8.3.133 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 ignored by OPM Flow and has no effect on the simulation.

8.3.134 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 ignored by OPM Flow and has no effect on the simulation.

8.3.135 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 ignored by OPM Flow and has no effect on the simulation.

8.3.136 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 ignored by OPM Flow and has no effect on the simulation.

8.3.137 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 ignored by OPM Flow and has no effect on the simulation.

8.3.138 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 ignored by OPM Flow and has no effect on the simulation.

8.3.139 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 ignored by OPM Flow and has no effect on the simulation.

8.3.140 LWKRO – END-POINT SCALING OF GRID CELL KRO(SWL) (LOW SALINITY AND WATER WET)

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 ignored by OPM Flow and has no effect on the simulation.

8.3.141 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 ignored by OPM Flow and has no effect on the simulation.

8.3.142 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 ignored by OPM Flow and has no effect on the simulation.

8.3.143 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 S_w = 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 ignored by OPM Flow and has no effect on the simulation.

8.3.144 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 ignored by OPM Flow and has no effect on the simulation.

8.3.145 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:

$$P_c = P_{c_{TABLE}} \left(\frac{HWPCW}{P_{c_{TABLE-MAX}}} \right) \quad \textbf{(8.13)}$$

Where:

P_c = the resulting high salinity water wet water capillary pressure for a grid cell.

LWPCW = 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 ignored by OPM Flow and has no effect on the simulation.

8.3.146 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 ignored by OPM Flow and has no effect on the simulation.

8.3.147 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 ignored by OPM Flow and has no effect on the simulation.

8.3.148 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 ignored by OPM Flow and has no effect on the simulation.

8.3.149 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 ignored by OPM Flow and has no effect on the simulation.

8.3.150 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 ignored by OPM Flow and has no effect on the simulation.

8.3.151 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 ignored by OPM Flow and has no effect on the simulation.

8.3.152 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.

OPM Flow does not recognize this keyword and the keyword may cause the program to terminate, but it is documented here for completeness.

8.3.153 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.

8.3.154 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.

8.3.155 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 has 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 "/" and there is no "/" terminator for the keyword.

Table 8.66: MISC Keyword Description

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

Example

```
--
--       SOLVENT MISCIBILITY-IMMISCIBLITY TRANSFORM TABLE
--
SGCWMIS
--       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.

8.3.156 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[83] 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 ignored by OPM Flow and has no effect on the simulation.

[83] *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*

8.3.157 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[84] 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 ignored by OPM Flow and has no effect on the simulation.

[84] *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*

8.3.158 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 "/

Table 8.67: MSFN Keyword Description

Examples

```
--
--        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.

8.3.159 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.

8.3.160 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.

8.3.161 NOWARNEP – DEACTIVATE END-POINT SCALING WARNING MESSAGES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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;

8.3.162 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. 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.

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 users 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.			
		$1/°R^2$ 9.26×10^{-7}	$1/°K^2$ 3.0×10^{-6}	$1/°K^2$ 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 "/" and there is no "/" terminator for the keyword.

Table 8.68: OILDENT Keyword Description

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.

8.3.163 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 users 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 "/" and there is no "/" terminator for the keyword.

Table 8.69: 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
```

8.3.164 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.

8.3.165 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.

8.3.166 OVERBURD – DEFINE ROCK OVERBURDEN PRESSURE VERSUS DEPTH TABLES

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 "/" and there is no "/" terminator for the keyword.

Table 8.70: OVERBURD Keyword Description

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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
```

The net result of the two examples in this case is identical.

8.3.167 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness. 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 "/".

Table 8.71: PCG Keyword Description

The capillary pressure for a grid block is scaled by:

$$P_c = P_{c_{TABLE}} \left(\frac{PCG}{P_{c_{TABLE-MAX}}} \right) \quad (8.14)$$

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.

8.3.168 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.

OPM Flow does not recognize this keyword and the keyword may cause the program to terminate, but it is documented here for completeness.

8.3.169 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/".

Table 8.72: PCW Keyword Description

See also the IPCW keyword for the equivalent imbibition functionality.

The capillary pressure for a grid block is scaled by:

$$P_c = P_{c_{TABLE}} \left(\frac{PCW}{P_{c_{TABLE-MAX}}} \right) \quad (8.15)$$

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.

8.3.170 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.

OPM Flow does not recognize this keyword and the keyword may cause the program to terminate, but it is documented here for completeness.

8.3.171 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 ignored by OPM Flow and has no effect on the simulation.

8.3.172 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.

8.3.173 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.

8.3.174 PLMIXPAR – DEFINE THE POLYMER TODD-LONGSTAFF MIXING PARAMETERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The PLMIXPAR keyword defines the Todd-Longstaff[85] 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 "/" and there is no "/" terminator for the keyword.

Table 8.73: 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.

[85] *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.*

8.3.175 PLYADS - DEFINE POLYMER ROCK ADSORPTION TABLES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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 of the saturated concentration of polymer adsorbed by the 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 "/" and there is no "/" terminator for the keyword.

Table 8.74: 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.

8.3.176 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.

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 "/" and there is no "/" terminator for the keyword.

Table 8.75: 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 DEPENDANCY 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.

8.3.177 PLYATEMP – DEFINE POLYMER ADSORPTION TABLE TEMPERATURE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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 ignored by OPM Flow and has no effect on the simulation, but is documented here for completeness.

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 "/".					

Table 8.76: 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 °F 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 °F for the subsequent two polymer rock adsorption tables.

8.3.178 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 ignored by OPM Flow and has no effect on the simulation.

8.3.179 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/" and there is no "/" terminator for the keyword.

Table 8.77: 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
--       POLCON     POLRATIO
--       -------    --------
             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.

8.3.180 PLYESAL – DEFINE POLYMER EFFECTIVE SALINITY COEFFICIENT

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.

OPM Flow does not recognize this keyword and the keyword may cause the program to terminate, but it is documented here for completeness.

8.3.181 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.

OPM Flow does not recognize this keyword and the keyword may cause the program to terminate, but it is documented here for completeness.

8.3.182 PLYMAX - DEFINE POLYMER-SALT VISCOSITY MIXING CONCENTRATIONS

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.

Currently the BRINE option is not implemented in OPM Flow and therefore SALTCON is 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 option is not implemented in OPM Flow.			
		lb/stb	kg/sm^3	gm/scc	None

Notes:

1) The keyword is followed by NTSFUN tables as declared on the TABDIMS keyword in the RUNSPEC section, for each rock region. There should be only one row per table.

2) Each table is terminated by "/" and there is no "/" terminator for the keyword.

Table 8.78: PLYDMAX Keyword Description

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.

8.3.183 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.

This keyword is ignored by OPM Flow and has no effect on the simulation.

8.3.184 PLYROCK - DEFINE POLYMER-ROCK PROPERTIES

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/rtb	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 "/" and there is no "/" terminator for the keyword.

Table 8.79: 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.

8.3.185 PLYSHEAR – ACTIVATE AND DEFINE POLYMER SHEARING PARAMETERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

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.

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 "/" and there is no "/" terminator for the keyword.

Table 8.80: 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.

8.3.186 PLYSHLOG - ACTIVATE AND DEFINE THE POLYMER SHEARING LOGARITHMIC PARAMETERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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 8.81: 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.

8.3.187 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 ignored by OPM Flow and has no effect on the simulation.

8.3.188 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 ignored by OPM Flow and has no effect on the simulation.

8.3.189 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 "/" and there is no "/" terminator for the keyword.

Table 8.82: 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.

8.3.190 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 ignored by OPM Flow and has no effect on the simulation.

8.3.191 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 ignored by OPM Flow and has no effect on the simulation.

8.3.192 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 ignored by OPM Flow and has no effect on the simulation.

8.3.193 PMAX – MAXIMUM AND MINIMUM PRESSURE FOR TOTAL COMPRESSIBILITY CHECK

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 ignored by OPM Flow and has no effect on the simulation.

8.3.194 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 "/" and there is no "/" terminator for the keyword.

Table 8.83: 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.

8.3.195 PPCWMAX – DEFINE SWATINIT CALCULATED CAPILLARY PRESSURE CONSTRAINTS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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.

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/" and there is no "/" terminator for the keyword.

Table 8.84: 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. 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.

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.

8.3.196 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 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.

8.3.197 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[86] 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 (μ_o) at the oil reference pressure.			
		cP	cP	cP	None
5	OVISCOMP	OVISCOMP is a real positive value defining the oil viiscosibility (μ_{oc}) at the oil reference pressure, Uwc(Pref) 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 "/" and there is no "/" terminator for the keyword.

Table 8.85: PVCDO Keyword Description

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.

[86] *"Dead" oil is oil that it contains no dissolved gas or a relatively thick oil or residue that has lost its volatile components.*

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.

8.3.198 PVCO - OIL PVT PROPERTIES FOR LIVE OIL

Description

PVCO defines the oil PVT properties for live[87] 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 (μ_o) 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*

[87] *"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
6	OVISCOMP	OVISCOMP is a real positive value defining the oil viiscosibility (μ_{oc}) at the saturated oil reference pressure with the given RS, where (μ_{oc}) 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 "/" and there is no "/" terminator for the keyword.

Table 8.86: 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 above 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.

There is no terminating "/" for this keyword.

8.3.199 PVDG - GAS PVT PROPERTIES FOR DRY GAS

Description

PVDG defines the gas PVT properties for dry gas[88]. 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 "/" and there is no "/" terminator for the keyword.

Table 8.87: PVDG Keyword Description

See also the RVCONST and RVCONSTT keywords to define the constant Rv for dry gas.

[88] *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

```
--
--        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 above example defines two dry PVT tables assuming NTPVT equals two and NPPVT is greater than or equal to 22 on the TABDIMS keyword in the RUNSPEC section.

There is no terminating "/" for this keyword.

8.3.200 PVDO – OIL PVT PROPERTIES FOR DEAD OIL

Description

PVDO defines the oil PVT properties for dead oil[89]. 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 "/" and there is no "/" terminator for the keyword.

Table 8.88: PVDO Keyword Description

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.

[89] *"Dead" oil is oil that it contains no dissolved gas or a relatively thick oil or residue that has lost its volatile components.*

Examples

```
--
--      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 above example 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.

There is no terminating "/" for this keyword.

8.3.201 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 "/" and there is no "/" terminator for the keyword.

Table 8.89: PVDS Keyword Description

Examples

```
--
--        GAS SOLVENT PVT TABLE
--
PVTS
--        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.

8.3.202 PVTG - GAS PVT PROPERTIES FOR WET GAS

Description

PVTG defines the gas PVT properties for wet gas[90]. 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 wet gas in the model. 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 values that defines the gas phase pressure. that defines 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 Rvs 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
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.			
			cP	cP	cP	None

[90] *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.*

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 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 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 8.90: PVTG Keyword Description

Examples

```
--
--       GAS PVT TABLE FOR WET GAS
--
PVTG
--       PRES       RV          BG          VISC
--       PSIA     STB/MSCF    RB/MSCF      CPOISE
--      ------    ---------   -------      ------
          300     0.000132     0.042340     0.01344
                  0            0.042310     0.01389     /
          600     0.000124     0.020460     0.01420
                  0            0.020430     0.01450     /
          900     0.000126     0.013280     0.01526
                  0            0.013250     0.01532     /
         1200     0.000135     0.009770     0.01660
                  0            0.009730     0.01634     /
         1500     0.000149     0.007730     0.01818
                  0            0.007690     0.01752     /
         1800     0.000163     0.006426     0.01994
                  0            0.006405     0.01883     /
         2100     0.000191     0.005541     0.02181
                  0            0.005553     0.02021     /
         2400     0.000225     0.004919     0.02370
                  0            0.004952     0.02163     /
                                                        / TABLE NO. 1
--       PRES       RV          BG          VISC
--       PSIA     STB/MSCF    RB/MSCF      CPOISE
--      ------    ---------   -------      ------
          300     0.000132     0.042340     0.01344
          600     0.000124     0.020460     0.01420
          900     0.000126     0.013280     0.01526
         1200     0.000135     0.009770     0.01660
         1500     0.000149     0.007730     0.01818
         1800     0.000163     0.006426     0.01994
         2100     0.000191     0.005541     0.02181
         2400     0.000225     0.004919     0.02370
                  0            0.004952     0.02163     /
                                                        / TABLE NO. 2
```

The above example defines two wet 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.

Notice that there is no terminating "/" for this keyword only for a table and a sub table.

8.3.203 PVTO - OIL PVT PROPERTIES FOR LIVE OIL

Description

PVTO defines the oil PVT properties for live[91] 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

[91] *"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 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 8.91: PVTO Keyword Description

Examples

The following example defines live oil 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.

```
--
--       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  /
          0.4050    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. 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 is no terminating "/" for this keyword only for a table and a sub table.

8.3.204 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 REGION 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 (μ_w) 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 viiscosibility (μ_{wc}) at the water reference pressure, Uwc(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 "/" and there is no "/" terminator for the keyword.

Table 8.92: PVTW Keyword Description

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.

```
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.
There is no terminating "/" for this keyword.

8.3.205 PVTWSALT - DEFINE BRINE WATER FLUID PROPERTIES FOR VARIOUS REGIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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 vector 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.

OPM Flow does not recognize this keyword and the keyword may cause the program to terminate, but it is documented here for completeness.

8.3.206 PVZG - GAS PVT PROPERTIES FOR DRY GAS (Z-FACTOR)

Description

PVZG defines the gas PVT properties for dry gas[92] 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/" and there is no "/" terminator for the keyword.

Table 8.93: PVZG Keyword Description

See also the RVCONST and RVCONSTT keywords to define the constant Rv for dry gas.

[92] *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.*

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[93], to account for deviations, from ideal, to the behavior of real gases. The PVT relation for a real gas can be defined by:

$$PV = ZnRT \tag{8.16}$$

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.

$$V_{sc} = \frac{Z_{sc} nRT_{sc}}{P_{sc}} \tag{8.17}$$

$$V_i = \frac{Z_i n RT_i}{P_i} \tag{8.18}$$

Thus the gas formation volume factor can be expressed as:

$$E = \frac{V_{sc}}{V_i} \tag{8.19}$$

And substituting equation (8.17) and (8.18) into (8.19) we obtain

$$E = \left(\frac{P_i}{P_{sc}}\right)\left(\frac{T_{sc}}{T_i}\right)\left(\frac{1}{Z_i}\right) \tag{8.20}$$

Incorporating standard pressure and temperature values gives in SI units we have:

$$E = \left(\frac{P_i}{101.325}\right)\left(\frac{273.15 + 15}{T_i}\right)\left(\frac{1}{Z_i}\right) = 2.84\left(\frac{P_i}{Z_i T_i}\right) \tag{8.21}$$

or in field units

$$E = \left(\frac{P_i}{14.7}\right)\left(\frac{460 + 60}{T_i}\right)\left(\frac{1}{Z_i}\right) = 35.37\left(\frac{P_i}{Z_i T_i}\right) \tag{8.22}$$

Where,

E	= gas formation volume factor (scf /rcft or Sm^3/m^3)
P	= pressure (psia or kPa)
P_{sc}	= pressure standard conditions (psia or kPa)
P_i	= initial reservoir pressure (psia or kPa)
V	= volume (ft^3 or m^3)
T	= absolute temperature (°R or K)
T_{sc}	= temperature at standard conditions (°R or K)
T_i	= initial reservoir temperature (°R or K)
R	= gas constant (10.73 or 8.314)

RTEMP on this keyword is the T_i in the above equations and P_i is PRESS columnar vector.

[93] *Standing, M. B.:"Volumetric and Phase Behaviour of Oil Field Hydrocarbon Systems", Renihold Publishing Corp., New York City (1952).*

Examples

```
---
--       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.

8.3.207 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 ignored by OPM Flow and has no effect on the simulation.

8.3.208 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.

8.3.209 RIVRXSEC – DEFINE RIVER CROSS-SECTION VERSUS DEPTH PARAMETERS

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 ignored by OPM Flow and has no effect on the simulation.

8.3.210 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 ignored by OPM Flow and has no effect on the simulation.

8.3.211 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 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	PRES	PRES is a real number defining the rock reference pressure for the other parameters for this data set.			
		psia 1.032	barsa 1.032	atma 1.032	Default
2	RCOMP	RCOMP is a real number defining the rock compressibility (C_f) 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 tables as declared on the TABDIMS keyword in the RUNSPEC section. 2) Each table is terminated by "/" and there is no "/" terminator for the keyword.					

Table 8.94: ROCK Keyword Description

Examples

The following shows the ROCK 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 (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
--
-- (1) 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 COMPRSSIBILITY REGION 1
    3966.9    5.5E-06                          / ROCK COMPRSSIBILITY REGION 2
    4566.9    6.0E-06                          / ROCK COMPRSSIBILITY 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.

8.3.212 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 "/" and there is no "/" terminator for the keyword.

Table 8.95: ROCK2D Keyword Description

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

See also the OVERBURD, ROCKTAB, ROCK2DTR, and ROCKWNOD keywords in the PROPS section.

Examples

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.

8.3.213 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 "/" and there is no "/" terminator for the keyword.

Table 8.96: ROCK2DTR Keyword Description

This keyword is ignored by OPM Flow and has no effect on the simulation, but is documented here for completeness.

See also the OVERBURD, ROCKTAB, ROCK2D, and ROCKWNOD keywords in the PROPS section.

Examples

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.

8.3.214 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.	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.	NOSTORE
3	ROCKOPT3	ROCKOPT3 is a character string that defines which region array should be used to allocate the various ROCK and ROCKTAB tabless. ROCKOPT3, should be set to ROCKNUM, SATNUM or PVTNUM.	PVTNUM
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.	DEFLATION
Notes: 1) The keyword is terminated by "/".			

Table 8.97: ROCKOPTS Keyword Description

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

Example

```
--
--       ROCKOPT1  ROCKOPT2   ROCKOPT3  ROCKOPT3
--       PRS/STRE  NO/STORE   ARRAY
--       --------  --------   --------  --------
ROCKOPTS
         PRESSURE  NOSTORE    PVTNUM    DEFLATION          / ROCK COMP OPTIONS
```

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

8.3.215 ROCKPAMA – DEFINE COAL PALMER-MANSORRI ROCK MODEL PARAMETERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

ROCKPAMA defines the Palmer-Mansoori[94] and [95] 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 ignored by OPM Flow and has no effect on the simulation.

[94] *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.*

[95] *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.*

8.3.216 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/" and there is no "/" terminator for the keyword.

Table 8.98: 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.

8.3.217 ROCKTABH – ROCK COMPACTION HYSTERESIS TABLES

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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/" and there is no "/" terminator for the keyword.

Table 8.99: 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.

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 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
```

8.3.218 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 ignored by OPM Flow and has no effect on the simulation.

8.3.219 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

8.3.220 ROCKTSIG – ROCK COMPACTION TABLES (DUAL POROSITY)

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 ignored by OPM Flow and has no effect on the simulation.

8.3.221 ROCKWNODE – 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 ROCKWNODE, 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.			
		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 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 8.100: ROCKWNOD Keyword Description

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

See also the OVERBURD, ROCKTAB, ROCK2D and ROCK2DTR keywords in the PROPS section.

Examples

The following example defines two ROCKWNODE 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.

8.3.222 RPTPROPS – DEFINE PROPS SECTION REPORTING

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 ignored by OPM Flow and has no effect on the simulation, but is documented here for completeness.

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 "/".

Table 8.101: 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                                       /
```

8.3.223 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[96] 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.

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 "/".					

Table 8.102: 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.

Examples

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                                  /
```

[96] *"Dead" oil is oil that it contains no dissolved gas or a relatively thick oil or residue that has lost its volatile components.*

8.3.224 RSCONSTT – DEFINE CONSTANT GOR (RS) FOR EACH DEAD OIL PVT FLUID

Description

RSCONSTT defines a constant Gas-Oil Ratio ("GOR"), for each dead oil[97] 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 ignored by OPM Flow and has no effect on the simulation, but is documented here for completeness.

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 "/" and there is no "/" terminator for the keyword.

Table 8.103: 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.

Examples

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
```

[97] *"Dead" oil is oil that it contains no dissolved gas or a relatively thick oil or residue that has lost its volatile components.*

8.3.225 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 ignored by OPM Flow and has no effect on the simulation.

8.3.226 RTEMP - DEFINE THE INITIAL RESERVOIR TEMPERATURE FOR THE MODEL

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword defines the reservoir temperature for when OPM Flow's thermal option has been activated by the THERMAL keyword in the RUNSPEC section. The RTEMPA keyword is alias for RTEMP; however the former is ignored by OPM Flow.

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 users the TEMP keyword in the RUNSPEC section to activate the "black-oil" thermal model.

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: 2) The keyword is terminated by "/".					

Table 8.104: 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 °F.

8.3.227 RTEMPA - DEFINE THE INITIAL RESERVOIR TEMPERATURE FOR THE MODEL

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword defines the reservoir temperature for when temperature or thermal options has been activated by the either the TEMP or THERMAL keywords in the RUNSPEC section in the commercial simulator.

The RTEMP keyword is alias for RTEMPA; however the latter is ignored by OPM Flow.

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 "/".

Table 8.105: RTEMPA Keyword Description

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness. Use the RTEMP keyword instead to set the reservoir temperature.

Example

```
--
--        RESERVOIR
--        TEMPERATURE
--        -----------
RTEMPA
          190.0                                        / RESERVOIR TEMPERATURE
```

The above example defines the reservoir temperature to be 190 °F.

8.3.228 RVCONST – DEFINE CONSTANT CGR (RV) FOR ALL DRY GAS PVT FLUIDS

Description

RVCONST defines a constant Condensate-Gas Ratio ("CGR" or Rv), for all dry gas[98] 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 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 and has no effect on the simulation, but is documented here for completeness.

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 "/".					

Table 8.106: 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.

Examples

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                                  /
```

[98] *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.*

8.3.229 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[99] 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 "/" and there is no "/" terminator for the keyword.

Table 8.107: 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.

Examples

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
```

[99] *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.*

8.3.230 RVGI – DEFINE CONDENSATE-GAS RATIO VERSUS PRESSURE AND GI TABLES

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 ignored by OPM Flow and has no effect on the simulation.

8.3.231 SALNODE – SALT CONCENTRATION BASED PVTNUM ARRAY

Description

SALNODE defines the salt concentration value based on a cells PVTNUM number. The SALNODE 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 SALNODE data sets with four values of salt concentrations need to be entered.

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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/sm^3	gm/scc	None
Notes: 1) Each table is terminated by "/" including the last table; however, there is no "/" terminator for the keyword.					

Table 8.108: SALNODE 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
--
SALNODE
      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.

8.3.232 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 "/".			

Table 8.109: 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
	Oil	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
	Oil	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 8.110: 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.

8.3.233 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 ignored by OPM Flow and has no effect on the simulation.

8.3.234 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 "/" and there is no "/" terminator for the keyword.

Table 8.111: 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.

8.3.235 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 users the value from the cell's relative permeability table.
3) The keyword is terminated by "/".

Table 8.112: 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, 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 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.

8.3.236 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 "/" and there is no "/" terminator for the keyword.

Table 8.113: 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.

8.3.237 SGF32D – GAS SATURATION TABLES VERSUS OIL AND WATER SATURATIONS

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 ignored by OPM Flow and has no effect on the simulation

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.

8.3.238 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 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 "/" and there is no "/" terminator for the keyword.

Table 8.114: SGFN Keyword Description

Example

```
--
--       GAS RELATIVE PERMEABILITY TABLES (SGFN)
--
SGFN
--       SGAS       KRG        PCGO
--       FRAC                  PSIA
--       --------  --------   -------
             0.00    0.0000       1*
             0.20    0.0002       1*
             0.85    0.4450       1*                    / TABLE NO. 01
--       --------  --------   -------
             0.00    0.0000       1*
             0.20    0.0002       1*
--       --------  --------   -------
             0.60    0.1412       1*
             0.70    0.2412       1*
             0.85    0.4450       1*                    / TABLE NO. 02
```

The example defines two SGFN tables for when gas is present in the input deck.

8.3.239 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 users the value from the cell's relative permeability table.
3) The keyword is terminated by "/".

Table 8.115: 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.

8.3.240 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 pressures 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 "/".

Table 8.116: 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, 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 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.

8.3.241 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 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	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 "/" and there is no "/" terminator for the keyword.

Table 8.117: SGOF Keyword Description

Example

```
--
--       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.

8.3.242 SGU – END-POINT SCALING GRID CELL GAS SATURATION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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 users the value from the cell's relative permeability table.
3) The keyword is terminated by "/".

Table 8.118: 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.

8.3.243 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 the 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 "/" and there is no "/" terminator for the keyword.

Table 8.119: 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.

8.3.244 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 ignored by OPM Flow and has no effect on the simulation.

See the PLYSHEAR keyword for the alternative polymer shear thinning/thickening option that is implemented in OPM Flow.

8.3.245 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/".

Table 8.120: 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.

Examples

Thet 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
```

8.3.246 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/".

Table 8.121: 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.

Examples

The example users 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
/
```

8.3.247 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/".

Table 8.122: 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.

Examples

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
```

8.3.248 SKRW – END-POINT SCALING OF GRID CELL KRW(SW =1.0) (SURFACTANT)

Description

SKRW defines the scaling parameter at the maximum surfactant water relative permeability value (SWU), that is for S_w = 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/".

Table 8.123: 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.

Examples

The example users the EQUALS keywordto 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
   '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
/
```

8.3.249 SKRWR – END-POINT SCALING OF GRID CELL KRWR(SOWCR) (SURFACTANT)

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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/".

Table 8.124: 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.

Examples

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
```

8.3.250 SLGOF – GAS-OIL SATURATION TABLES VERSUS GAS (FORMAT TYPE 1)

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 gas 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 "/" and there is no "/" terminator for the keyword.

Table 8.125: SLGOF Keyword Description

Example

```
--
--       GAS-OIL RELATIVE PERMEABILITY TABLES (SLGOF)
SLOF
--       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 SGOF tables for use when oil, gas and water are present in the run.

8.3.251 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 users the value from the cell's relative permeability table.
3) The keyword is terminated by "/".

Table 8.126: 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.

8.3.252 SOF2 – OIL SATURATION TABLES WITH RESPECT TO GAS OR WATER (FORMAT TYPE 2)

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 the 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 "/" and there is no "/" terminator for the keyword.

Table 8.127: SOF2 Keyword Description

Not 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
            0.56      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
            0.56      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.

8.3.253 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 "/" and there is no "/" terminator for the keyword.

Table 8.128: 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
            0.56      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
            0.56      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.

8.3.254 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 ignored by OPM Flow and has no effect on the simulation.

8.3.255 SOGCR – END-POINT SCALING GRID CELL CRITICAL OIL SATURATION WITH RESPECT TO GAS

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± , SOGCRX± and SOGCRX± series of keyword should be used.
2) If the value for a cell has been defaulted then OPM Flow users the value from the cell's relative permeability table.
3) The keyword is terminated by "/".

Table 8.129: 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, 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 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.

8.3.256 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[100] first three phase oil relative permeability model as modified by Aziz and Settari[101]. 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 ignored by OPM Flow and has no effect on the simulation.

[100] *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.*

[101] *Aziz, K. and Settari, A. Petroleum Reservoir Simulation, London, UK, Applied Science Publishers (1979), page 398.*

8.3.257 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[102] first three phase oil relative permeability model as modified by Aziz and Settari[103]. 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 ignored by OPM Flow and has no effect on the simulation.

[102] *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.*

[103] *Aziz, K. and Settari, A. Petroleum Reservoir Simulation, London, UK, Applied Science Publishers (1979), page 398.*

8.3.258 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 "/" and there is no "/" terminator for the keyword.

Table 8.130: 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.

8.3.259 SOWCR – END-POINT SCALING GRID CELL CRITICAL OIL SATURATION WITH RESPECT TO WATER

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± , SOWCRX± and SOWCRX± series of keyword should be used.

2) If the value for a cell has been defaulted then OPM Flow users the value from the cell's relative permeability table.

3) The keyword is terminated by "/".

Table 8.131: 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, 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 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.

8.3.260 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 REGION 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 users 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/°R	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/°R	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/°R	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 "/" and there is no "/" terminator for the keyword.

Table 8.132: 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.

8.3.261 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 REGION 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 users 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/ft^3/°R	kJ/m^3/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 "/" and there is no "/" terminator for the keyword.

Table 8.133: SPECROCK Keyword Description

See also the SPECHEAT keyword to define the specif 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.

8.3.262 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 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 SOLVENT options have 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	KRG^t	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	KRS^t	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 "/" and there is no "/" terminator for the keyword.

Table 8.134: SSFN Keyword Description

Example

```
--
--       SOLVENT RELATIVE PERMEABILITY TABLES
--
SSFN
--       SGAS       KRGT         KRST
--       FRAC
--       --------  --------     --------
          0.0000    0.0000       1.0000
          1.0000    1.0000       0.0000                    / 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       0.0000                    / TABLE NO. 02
```

The above example defines two SSFN tables for use with the MISCIBLE and SOLVENT options.

8.3.263 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 users the value from the cell's relative permeability table.

2) The keyword is terminated by "/".

Table 8.135: 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.

8.3.264 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 users the value from the cell's relative permeability table.

2) The keyword is terminated by "/".

Table 8.136: 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.

8.3.265 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 users the value from the cell's relative permeability table.
2) The keyword is terminated by "/".

Table 8.137: 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.

8.3.266 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 users the value from the cell's relative permeability table.
2) The keyword is terminated by "/".

Table 8.138: 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.

8.3.267 SSWCR – END-POINT SCALING GRID CELL CRITICAL WATER SATURATION

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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 users the value from the cell's relative permeability table.

2) The keyword is terminated by "/".

Table 8.139: 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.

8.3.268 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 users the value from the cell's relative permeability table.

2) The keyword is terminated by "/".

Table 8.140: 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.

8.3.269 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 users the value from the cell's relative permeability table.

2) The keyword is terminated by "/".

Table 8.141: 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.

8.3.270 STOG DEFINE CAPILLARY PRESSURE OIL-GAS SURFACE TENSION VERSUS PRESSURE

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.

OPM Flow does not recognize this keyword and the keyword may cause the program to terminate, but it is documented here for completeness.

8.3.271 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[104] second three phase oil relative permeability model as modified by Aziz and Settari[105]. 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.

[104] *Stone, H. L. "Estimation of Three-Phase Relative Permeability and Residual Oil Data," Journal of Canadian Petroleum Technology (1973) 12, No. 4, 53-61.*

[105] *Aziz, K. and Settari, A. Petroleum Reservoir Simulation, London, UK, Applied Science Publishers (1979), page 398.*

8.3.272 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[106] first three phase oil relative permeability model as modified by Aziz and Settari[107]. 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.

[106] *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.*

[107] *Aziz, K. and Settari, A. Petroleum Reservoir Simulation, London, UK, Applied Science Publishers (1979), page 398.*

8.3.273 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[108] first three phase oil relative permeability model as modified by Aziz and Settari[109]. 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 SOLUTION 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 "/"
3) There is no "/" terminator for the keyword.

Table 8.142: 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                                          / SATURATIOM TABLE NO. 01
         1.000                                          / SATURATIOM TABLE NO. 02
         2.000                                          / SATURATIOM TABLE NO. 03
         1.000                                          / SATURATIOM TABLE NO. 04
         3.000                                          / SATURATIOM 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.

[108] *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.*

[109] *Aziz, K. and Settari, A. Petroleum Reservoir Simulation, London, UK, Applied Science Publishers (1979), page 398.*

8.3.274 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[110] second three phase oil relative permeability model as modified by Aziz and Settari[111]. 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.

[110] *Stone, H. L. "Estimation of Three-Phase Relative Permeability and Residual Oil Data," Journal of Canadian Petroleum Technology (1973) 12, No. 4, 53-61.*

[111] *Aziz, K. and Settari, A. Petroleum Reservoir Simulation, London, UK, Applied Science Publishers (1979), page 398.*

8.3.275 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 ignored by OPM Flow and has no effect on the simulation.

8.3.276 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.

OPM Flow does not recognize this keyword and the keyword may cause the program to terminate, but it is documented here for completeness.

8.3.277 SURFADDW – DEFINED SURFACTANT ADSORBED CONCENTRATION VERSUS WETTABILITY FRACTION

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 ignored by OPM Flow and has no effect on the simulation.

8.3.278 SURFADS - DEFINE SURFACTANT ROCK ADSORPTION TABLES

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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/" and there is no "/" terminator for the keyword.

Table 8.143: 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.

8.3.279 SURFCAPD – CAPILLARY NUMBER VERSUS MISCIBILITY TABLES

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 ignored by OPM Flow and has no effect on the simulation.

8.3.280 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 ignored by OPM Flow and has no effect on the simulation.

8.3.281 SURFROCK - DEFINE SURFACTANT-ROCK PROPERTIES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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
3	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 "/" and there is no "/" terminator for the keyword.

Table 8.144: 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.

8.3.282 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 ignored by OPM Flow and has no effect on the simulation.

8.3.283 SURFSTES - SURFACTANT WATER-OIL SURFACE TENSION VERSUS SURFACTANT AND SALT CONCENTRATIONS

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 ignored by OPM Flow and has no effect on the simulation.

8.3.284 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/" and there is no "/" terminator for the keyword.

Table 8.145: 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.

8.3.285 SWCR – END-POINT SCALING GRID CELL CRITICAL WATER SATURATION

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± , SWCRX± and SWCRX± series of keyword should be used.

2) If the value for a cell has been defaulted then OPM Flow users the value from the cell's relative permeability table.

3) The keyword is terminated by "/".

Table 8.146: 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, 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 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.

8.3.286 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 ignored by OPM Flow and has no effect on the simulation.

8.3.287 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) Each table is terminated by "/" and there is no "/" terminator for the keyword.

Table 8.147: SWFN Keyword Description

Example

```
--
--       WATER RELATIVE PERMEABILITY TABLES (SWFN)
--
SWFN
--       SWAT       KRW        PCOW
--       FRAC       FRAC       PSIA
--       --------   --------   -------
           0.15     0.00000     1*
           0.20     6.25e-6     1*
           0.25     0.00010     1*
           0.30     0.00050     1*
           0.35     0.00160     1*
           0.40     0.00390     1*
           0.45     0.00810     1*
           0.50     0.01500     1*
           0.55     0.02560     1*
           0.60     0.04100     1*
           0.65     0.06250     1*
           0.70     0.09150     1*
           0.75     0.12960     1*
           0.80     0.17850     1*
           0.85     0.24010     1*
           0.90     0.31640     1*
           0.95     0.40960     1*
           1.00     0.52200     1*                    / TABLE NO. 1
--       --------   --------   -------
           0.15     0.00000     1*
           0.20     6.25e-6     1*
           0.25     0.00010     1*
           0.30     0.00050     1*
           0.35     0.00160     1*
           0.40     0.00390     1*
           0.45     0.00810     1*
           0.50     0.01500     1*
           0.55     0.02560     1*
           0.60     0.04100     1*
           0.65     0.06250     1*
           0.70     0.09150     1*
           0.75     0.12960     1*
           0.80     0.17850     1*
           0.85     0.24010     1*
           0.90     0.31640     1*
           0.95     0.40960     1*
           1.00     0.52200     1*                    / TABLE NO. 2
```

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.

8.3.288 SWL – END-POINT SCALING GRID CELL CONNATE WATER SATURATION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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 users the value from the cell's relative permeability table.

3) The keyword is terminated by "/".

Table 8.148: 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.

8.3.289 SWLPC – END-POINT SCALING GRID CELL CAPILLARY PRESSURE CONNATE WATER SATURATIONS

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 "/".

Table 8.149: 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.

8.3.290 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 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 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 "/" and there is no "/" terminator for the keyword.

Table 8.150: SWOF Keyword Description

Example

```
--
--       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
         0.522283   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   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
         0.522283   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   0.5000    0.0000    0.000000
         1.000000   0.9000    0.0000    0.000000          / TABLE NO. 01
```

The example defines two SWFN 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.

8.3.291 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 users the value from the cell's relative permeability table.

3) The keyword is terminated by "/".

Table 8.151: 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.

8.3.292 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 ignored by OPM Flow and has no effect on the simulation.

8.3.293 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 ignored by OPM Flow and has no effect on the simulation.

8.3.294 THERMEXI – DEFINE COMPONENT THERMAL EXPANSION COEFFICIENTS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

THERMEXI defines the thermal expansion coefficients for all components in the model for when the EOS and THERMAL options have been invoked by the EOS and THERMAL keywords in the RUNSPEC section.

This keyword is ignored by OPM Flow and has no effect on the simulation.

8.3.295 TLMIXPAR – DEFINE THE MISCIBLE TODD-LONGSTAFF MIXING PARAMETERS

Description

The TLMIXPAR keyword defines the Todd-Longstaff[112] mixing parameters, for when either the miscible or solvent options have been activated by the MISCIBLE or SOLVENT keyword 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 "/" and there is no "/" terminator for the keyword.

Table 8.152: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.

[112] *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.*

8.3.296 TOLCRIT – DEFINE THE CRITICAL SATURATION TOLERANCE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

Critical fluid saturations are determine 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 keywords 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 "/".					

Table 8.153:TOLCRIT Keyword Description

See also section *8.2.2 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}.

8.3.297 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[113] and [114] 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 ignored by OPM Flow and has no effect on the simulation.

[113] *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.*

[114] *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.*

8.3.298 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[115] and [116] 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 ignored by OPM Flow and has no effect on the simulation.

[115] *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.*

[116] *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.*

8.3.299 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. 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 should be 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	PARPHASE	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 PARPHASE only needs to be defined if the partitioned tracer option has been activate with the PARTTRAC keyword in the RUNSPEC section.			None

Notes:

1) Each record (or row) should be terminated by a "/" and the keyword is terminated by "/".

Table 8.154:TRACER Keyword Description

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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.

8.3.300 TRACERKM – MULTI-PARTITIONED TRACER OPTION K(P) TABLES

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:

$$K_{ow}(P) = \frac{C_{oil}}{C_{water}} \quad \text{and} \quad K_{gw}(P) = \frac{C_{gas}}{C_{water}} \tag{8.23}$$

Where:

$K_{ow}(P)$ = multi-partitioned oil-water K(P)
$K_{gw}(P)$ = multi-partitioned gas-water K(P)
C_{oil} = oil concentration
C_{gas} = gas concentration
C_{water} = water concentration

See also the TRACERKP keyword in the PROPS section that provides similar data for standard partitioned tracers.

This keyword is ignored by OPM Flow and has no effect on the simulation.

8.3.301 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:

$$K(P) = \frac{C_{gas}}{C_{oil}} \tag{8.24}$$

Where:

K(P)	= standard partitioned K(P)
C_{oil}	= oil concentration
C_{gas}	= gas concentration

See also the TRACERKM keyword in the PROPS section that provides similar data for tmulti-partitioned tracers.

This keyword is ignored by OPM Flow and has no effect on the simulation.

8.3.302 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 ignored by OPM Flow and has no effect on the simulation.

8.3.303 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 ignored by OPM Flow and has no effect on the simulation.

8.3.304 TRADS – ENVIRONMENTAL TRACER ADSORPTION TABLES

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 ignored by OPM Flow and has no effect on the simulation.

8.3.305 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 ignored by OPM Flow and has no effect on the simulation.

8.3.306 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 ignored by OPM Flow and has no effect on the simulation.

8.3.307 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 ignored by OPM Flow and has no effect on the simulation.

8.3.308 TRNHD – ACTIVATE DISPERSION NON-HOMOGENEOUS DIFFUSION OPTION

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 ignored by OPM Flow and has no effect on the simulation.

8.3.309 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 ignored by OPM Flow and has no effect on the simulation.

8.3.310 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/".			

Table 8.155: 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.

8.3.311 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[117, 118 and 119]. Dake[120], in chapter eight, reports a typical value of β to be 10.0^7 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 ignored by OPM Flow and has no effect on the simulation.

[117] *Geertsma, J., 1974. Estimating the Coefficient of Inertial Resistance in Fluid Flow Through Porous Media. Soc.Pet.Eng.J., October: 445-450.*

[118] *Gewers, C.W.W. and Nichol, L.R., 1969. Gas Turbulence Factor in a Microvugular Carbonate. J.Can.Pet.Tech., April.*

[119] *Wong, S.W., 1970. Effects of Liquid Saturation on Turbulence Factors for Gas Liquid Systems. J.Can.Pet.Tech., October*

[120] *Dake, L.P. Fundamentals of Reservoir Engineering, Amsterdam, The Netherlands, Elsevier Science BV (1978) Chapter 8.6, pages 252-257.*

8.3.312 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[121, 122 and 123]. Dake[124], in chapter eight, reports a typical value of β to be 10.0^7 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 ignored by OPM Flow and has no effect on the simulation.

[121] *Geertsma, J., 1974. Estimating the Coefficient of Inertial Resistance in Fluid Flow Through Porous Media. Soc.Pet.Eng.J., October: 445-450.*

[122] *Gewers, C.W.W. and Nichol, L.R., 1969. Gas Turbulence Factor in a Microvugular Carbonate. J.Can.Pet.Tech., April.*

[123] *Wong, S.W., 1970. Effects of Liquid Saturation on Turbulence Factors for Gas Liquid Systems. J.Can.Pet.Tech., October*

[124] *Dake, L.P. Fundamentals of Reservoir Engineering, Amsterdam, The Netherlands, Elsevier Science BV (1978) Chapter 8.6, pages 252-257.*

8.3.313 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:

$$VE_{(average)} = (1.0 - \alpha) \times (SATNUM_{curves}) + \alpha \times (VE\ Model_{curves}) \quad \textbf{(8.25)}$$

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 ignored by OPM Flow and has no effect on the simulation.

8.3.314 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:

$$VE_{(average)} = (1.0 - \alpha) \times (SATNUM_{curves}) + \alpha \times (VE\ Model_{curves}) \quad \textbf{(8.26)}$$

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 ignored by OPM Flow and has no effect on the simulation.

8.3.315 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:

$$VE_{(average)} = (1.0 - \alpha) \times (SATNUM_{curves}) + \alpha \times (VE\ Model_{curves}) \quad \textbf{(8.27)}$$

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 ignored by OPM Flow and has no effect on the simulation.

8.3.316 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:

$$VE_{(average)} = (1.0 - \alpha) \times (SATNUM_{curves}) + \alpha \times (VE\ Model_{curves}) \quad \textbf{(8.28)}$$

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 ignored by OPM Flow and has no effect on the simulation.

8.3.317 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 users 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 "/" and there is no "/" terminator for the keyword.

Table 8.156: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 TEMP 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.

8.3.318 WAGHYSTR – DEFINE WATER-ALTERNATING-GAS HYSTERESIS PARAMETERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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[125]; Christensen et al., 1998[126]; and Christensen et al., 2001[127]). 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 inject 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 N_2 and CO_2 Gases. WAG flooding has been successfully applied to various fields worldwide.

This keyword is ignored by OPM Flow and has no effect on the simulation.

[125] *Caudle, B. H., & Dyes, A. B. (1958, January 1). Improving Miscible Displacement by Gas-Water Injection. Society of Petroleum Engineers.*

[126] *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.*

[127] *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.*

8.3.319 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. 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 users 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.			
		$1/°R^2$ 9.26×10^{-7}	$1/°K^2$ 3.0×10^{-6}	$1/°K^2$ 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 "/" and there is no "/" terminator for the keyword.

Table 8.157:WATDENT Keyword Description

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.

8.3.320 WATVISCT – DEFINE WATER VISCOSITY VERSUS TEMPERATURE FUNCTIONS

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 users 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 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 "/" and there is no "/" terminator for the keyword.

Table 8.158: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    0.500
           110.0    0.550
           120.0    0.580
           150.0    0.620
           165.0    0.625                                  / TABLE NO. 01
```

There is no terminating "/" for this keyword.

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, users 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 (PVTNUM), 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.

Properties Section	**Property Allocation**	**REGIONS Section Keyword**
SOLUTION	Equilibrium region allocation based on the EQUIL keyword records.	EQLNUM
REGION	Fluid In-Place reporting via the FIPNUM array that divides the model into different fluid in-place reporting regions.	FIPNUM
PROPS	PVT table allocation of the DENSITY, PVDG, PVDO, PVTG, PVTO, PVCO, PVTW and ROCK tables	PVTNUM
PROPS	Saturation (relative permeability) table allocation of the SGFN, SWFN, SOF2, SOF3, SGOF, and SWOF tables.	SATNUM
Function Specific Regions		
PROPS	ENPTVD and ENKRVD versus depth table allocation for when ENDSCALE option has been activated in the RUNSPEC section.	ENDNUM
PROPS	Imbibition saturation table allocation of the SWFN, SOF2, SOF3 or SWOF imbibition tables.	IBMNUM
PROPS	Miscible regions based on the TLMIXPAR records when the MISCIBLE or SOLVENT keywords have been activated in the RUNSPEC section.	MISNUM
PROPS	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.	ROCKNUM
PROPS	Surfactant saturation (relative permeability) tables allocation allocating the SWFN, SOF2, SOF3 or SWOF as miscible tables.	SURFNUM
PROPS	High salinity water wet saturation table allocation using the high salinity water wet saturation SWFN and SOFN tables.	SURFWNUM

Properties Section	Property Allocation	REGIONS Section Keyword

Notes:

1) Only EQLNUM, FIPNUM, PVTNUM, SATNUM , IMBUM and MISNUM are available in OPM Flow.
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

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[128] field are displayed in Figure 9.1 and Figure 9.2, respectively.

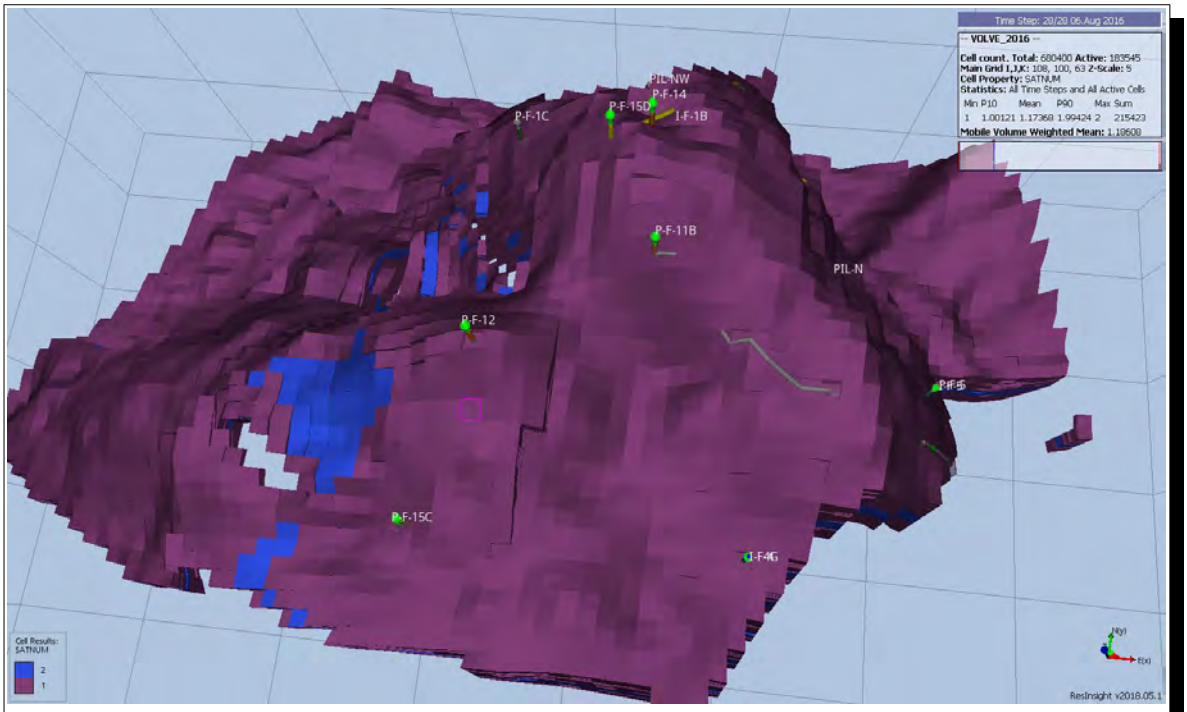


Figure 9.1: Volve Full Field Model SATNUM Array

[128] *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.*

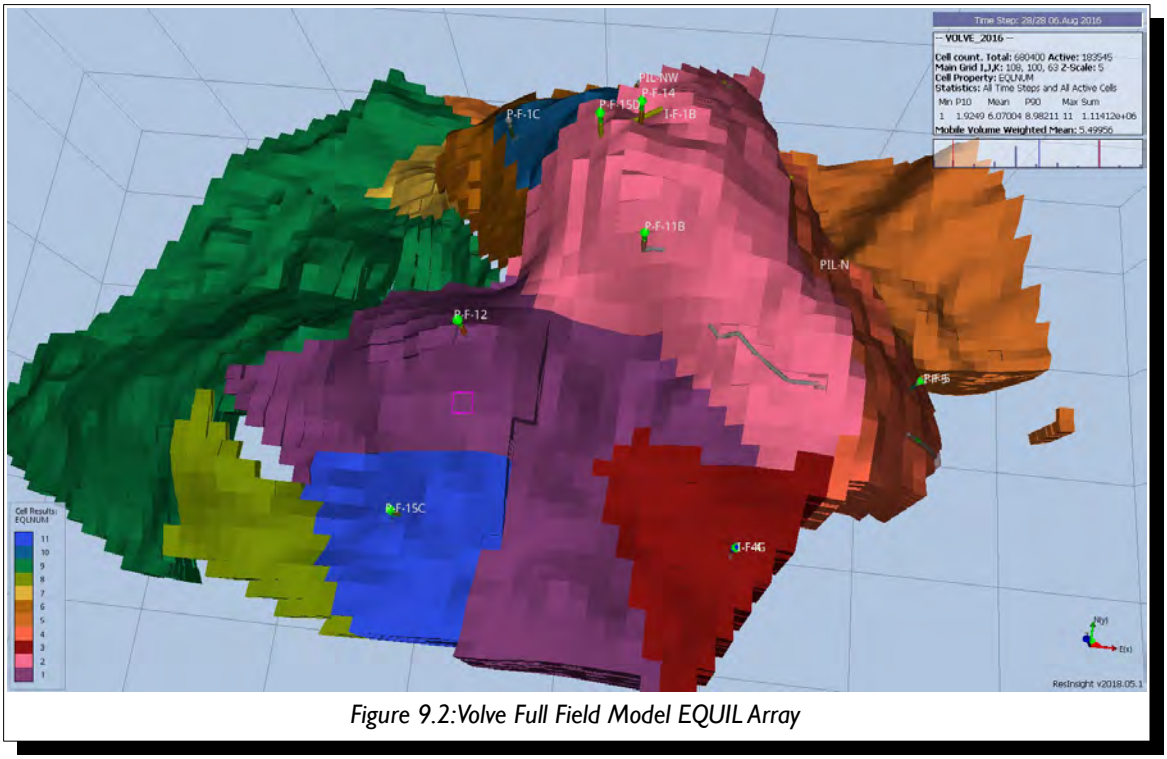


Figure 9.2:Volve Full Field Model EQUIL Array

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 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 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.

9.3.6 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.

9.3.7 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.

9.3.8 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.

9.3.9 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.

9.3.10 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[129] and the ENKRVD[130] 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 "/".

Table 9.2: ENDNUM Keyword Description

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

Examples

The example below sets three ENDNUM regions for a 4 x 5 x 2 model.

```
ENDNUM
2 2 1 1 # layer 1
2 2 1 1
1 1 1 1
1 1 1 1
1 1 1 1

3 3 1 1 # layer 2
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
/
```

[129] *This keyword is ignored by OPM Flow and has no effect on the simulation*

[130] *This keyword is ignored by OPM Flow and has no effect on the simulation*

9.3.11 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 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 fluid in-place region. The maximum number of EQLNUM regions is set by the NTEQUIL 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 EQULNUM 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 "/".

Table 9.3: EQLNUM Keyword Description

Examples

The example below sets three EQLNUM regions for a 4 x 5 x 2 model.

```
EQLNUM

2 2 1 1 # layer 1
2 2 1 1
1 1 1 1
1 1 1 1
1 1 1 1

3 3 1 1 # layer 2
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
/
```

9.3.12 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.

9.3.13 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.

9.3.14 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.

9.3.15 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 FIPNUM and FIP regions must be defined by the NMFIPR variable on the REGDIMS keyword in the RUNSPEC section.

OPM Flow does not recognize this keyword and the keyword may cause the program to terminate, but it is documented here for completeness.

No.	Name	Description	Default
1	FIP	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 FIP. The maximum number of FIP and FIPNUM regions is set by the NTFIP variable on either REGDIMS or TABDIMS keywords 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 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 "/".

Table 9.4: FIP Keyword Description

Examples

The example below defines a region name of UNIT and sets three FIPUNIT regions for a 4 x 5 x 2 model.

```
FIPUNIT

2 2 1 1 # layer 1
2 2 1 1
1 1 1 1
1 1 1 1
1 1 1 1

3 3 1 1 # layer 2
3 3 1 1
1 1 1 1
1 1 1 1
1 1 1 1
/
```

9.3.16 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 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 regions is set by the NTFIP variable on either REGDIMS or TABDIMS keywords 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 acell is not assigned a FIPNUM region then the default value will be used.
3) The keyword is terminated by "/".

Table 9.5: FIPNUM Keyword Description

Examples

The example below sets three FIPNUM regions for a 4 x 5 x 2 model.

```
FIPNUM

2 2 1 1 # layer 1
2 2 1 1
1 1 1 1
1 1 1 1
1 1 1 1

3 3 1 1 # layer 2
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
/
```

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.

9.3.17 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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.

9.3.18 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.

9.3.19 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 ignored by OPM Flow and has no effect on the simulation.

9.3.20 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 ignored by OPM Flow and has no effect on the simulation.

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

9.3.21 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.

9.3.22 HWSNUM – DEFINE THE SATURATION TABLE REGION NUMBERS (HIGH SALINITY AND WATER WET)

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 Low Salt and Surfactant Wettability options have been selected. The region number specifies which set of relative permeability tables are used to calculate the relative permeability and capillary pressure in a grid block.

This keyword is ignored by OPM Flow and has no effect on the simulation, but is documented here for completeness.

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 "/".

Table 9.8: HWSNUM Keyword Description

Examples

The example below sets three HWSNUM regions for a 4 x 5 x 2 model.

```
HWSNUM
2 2 1 1 # layer 1
2 2 1 1
1 1 1 1
1 1 1 1
1 1 1 1

3 3 1 1 # layer 2
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
/
```

9.3.23 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 "/".
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 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.

Note that The directional form of the keyword is recognized, but is not supported by OPM Flow.

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
/
```

9.3.24 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 ignored by OPM Flow and has no effect on the simulation.

9.3.25 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-processoring 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.

9.3.26 KRNUM – DEFINE THE DIRECTIONAL SATURATION TABLE REGION NUMBERS

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.

This keyword is ignored by OPM Flow and has no effect on the simulation, but is documented here for completeness.

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 KRBNUM region then the default value of one will be used.
3) The keyword is terminated by "/".

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-.

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
   'KRNUZM'     3             1*  1*   1*  1*   1*  1* / SET Z-DIR TABLES
/
```

9.3.27 KRNUMMF – DEFINE THE SATURATION TABLE REGION NUMBERS (MATRIX-FRACTURE)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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 ignored by OPM Flow and has no effect on the simulation.

9.3.28 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 ignored by OPM Flow and has no effect on the simulation, but is documented here for completeness.

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 "/".

Table 9.11: LSLTWNUM Keyword Description

Examples

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
/
```

9.3.29 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 ignored by OPM Flow and has no effect on the simulation, but is documented here for completeness.

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 "/".

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.

Examples

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
/
```

9.3.30 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 ignored by OPM Flow and has no effect on the simulation, but is documented here for completeness.

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 "/".

Table 9.13: LWSLTNUM Keyword Description

Examples

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
/
```

9.3.31 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 ignored by OPM Flow and has no effect on the simulation, but is documented here for completeness.

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 "/".

Table 9.14: LWSNUM Keyword Description

The HWSNUM allocated tables correspond to the immiscible high salinity water wet curves.

Examples

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
/
```

9.3.32 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 "/".

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
/
```

9.3.33 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.

9.3.34 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.

9.3.35 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.

9.3.36 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.

9.3.37 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.

9.3.38 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 moduls functions and shear modulus functions as defined by the PECOEFS, PEKTAB and PEGTAB series of keywords in the PROPS section.

This keyword is ignored by OPM Flow and has no effect on the simulation.

9.3.39 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 "/".

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
/
```

9.3.40 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 "/".

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 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.

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.

```
PVTNUM

2 2 1 1 # layer 1
2 2 1 1
1 1 1 1
1 1 1 1
1 1 1 1

3 3 1 1 # layer 2
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.

9.3.41 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.

9.3.42 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.

9.3.43 RESIDNUM – DEFINE VERTICAL EQUILIBRIUM RESIDUAL FLOW REGION NUMBERS

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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/".

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.

Examples

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
/
```

9.3.44 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 "/".

Table 9.19: ROCKNUM Keyword Description

Examples

The example below sets three ROCKNUM regions for a 4 x 5 x 2 model.

```
ROCKNUM

2 2 1 1 # layer 1
2 2 1 1
1 1 1 1
1 1 1 1
1 1 1 1

3 3 1 1 # layer 2
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
/
```

9.3.45 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 "/".

Table 9.20: RPTREGS Keyword Description

This keyword is ignored by OPM Flow and has no effect on the simulation, but is documented here for completeness.

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.

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 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
         DX       DY       DZ       DEPTH    PORO     PERMX                  /
```

9.3.46 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 "/".

Table 9.21: SATNUM Keyword Description

Examples

The example below sets three SATNUM regions for a 4 x 5 x 2 model.

```
SATNUM

2 2 1 1 # layer 1
2 2 1 1
1 1 1 1
1 1 1 1
1 1 1 1

3 3 1 1 # layer 2
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
/
```

9.3.47 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 ignored by OPM Flow and has no effect on the simulation, but is documented here for completeness.

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 "/".

Table 9.22: SURFNUM Keyword Description

Examples

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
/
```

9.3.48 SURFWNUM – DEFINE THE SATURATION TABLE REGION NUMBERS (HIGH SALINITY AND WATER WET)

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 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 ignored by OPM Flow and has no effect on the simulation, but is documented here for completeness.

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 "/".

Table 9.23: SURFWNUM Keyword Description

Examples

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
/
```

9.3.49 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 ignored by OPM Flow and has no effect on the simulation, but is documented here for completeness.

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. 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 "/".

Table 9.25:TNUM Keyword Data Description

See also the TRACER keyword in the PROPS section and the TBLK keyword in the SOLUTION section.

Examples

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.

9.3.50 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 TRACER 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 ignored by OPM Flow and has no effect on the simulation, but is documented here for completeness.

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, 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 "/".

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.

Examples

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.

9.3.51 WH2NUM – DEFINE WAG HYSTERESIS SATURATION TABLE REGION NUMBERS (TWO PHASE)

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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/".

Table 9.28:WH2NUM Keyword Description

Examples

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
/
```

9.3.52 WH3NUM – DEFINE WAG HYSTERESIS SATURATION TABLE REGION NUMBERS (THREE PHASE)

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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/".

Table 9.29:WH3NUM Keyword Description

Examples

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
/
```

10 SOLUTION SECTION

10.1 INTRODUCTION

To be written in a future release of the manual.

10.2 DATA REQUIREMENTS

To be written in a future release of the manual.

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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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, API			
		feet	m	cm	None
2	API	A columnar vector of real values that defines the vaporized oil-gas ratio values, values at the corresponding DEPTH.			
		°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 "/" and there is no "/" terminator for the keyword.

Table 10.1: RVVD 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
--        ------    --------
RSVD
          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

AQANCONL keyword defines how analytical aquifers are connected to an LGR simulation grid, this includes Carter-Tracy and Fetkovich analytical aquifers, although the Carter-Tracy aquifer is the only aquifer currently implemented in OPM Flow.

This keyword is ignored by OPM Flow and has no effect on the simulation.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	AQUNUM	AQUNUM is a positive integer greater than or equal to one and less than the maximum number of analytical aquifers as defined by the NANAQ variable on the AQUDIMS keyword in the RUNSPEC section, that defines the aquifer to be connected to the grid.			None
2	LGRNAME	LGRNAME is a character string of up to eight characters that defines the name of the LGR to be connected to AQUNUM.			None
3	I1	A positive integer that defines the lower bound of the cells in the I-direction to be connected to the LGR grid and 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 of the cells in the I-direction to be connected to the LGR grid and 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 cells in the J-direction to be connected to the LGR grid and 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 cells in the J-direction to be connected to the LGR grid and 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 cells in the K-direction to be to be connected to the LGR grid and 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 cells in the K-direction to be connected to the LGRg rid and must be greater than or equal to K1 and less than or equal to NZ.			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 then this values is applied to all cells declared by this record.			
		ft^2	m^2	cm^2	1*

No.	Name	Description			Default
		Field	Metric	Laboratory	
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 LGR 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 LGR grid. This is the recommended and the default value.			NO

Notes:

1) Where NX, NY and NZ are the dimensions of the LGR grid as defined on the CARFIN or RADFIN keywords in the GRID section.
2) Each record must be terminated by a "/" and the keyword is terminated by "/".

Table 10.2: AQANCONL Keyword Description

Example

The following example defines aquifer number one connected to the I+ face of LGR called LGROP01, LGROP01 must have been previously defined via the CARFIN or RADFIN keywords in the GRID section.

```
--
--                      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    LGROP01    1   1   1   25   1    2   '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.

10.3.5 AQANNC – DEFINE ANALYTIC AQUIFER NON-NEIGHBOR CONNECTIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

AQANNC defines the analytic aquifer non-neighbor connections.

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

10.3.6 AQANTRC - DEFINE ANALYTIC AQUIFER INITIAL TRACER CONCENTRATIONS

Description

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 of greater readability in the output.

OPM Flow does not recognize this keyword and the keyword may cause the program to terminate, but it is documented here for completeness.

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, although the Carter-Tracy aquifer is the only aquifer currently 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.

OPM Flow does not recognize this keyword and the keyword may cause the program to terminate, but it is documented here for completeness.

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.

OPM Flow does not recognize this keyword and the keyword may cause the program to terminate, but it is documented here for completeness.

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 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 ignored by OPM Flow and has no effect on the simulation; however, see the AQUFET 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 (C_t) of the aquifer, that is the rock compressibility (C_f) plus the water compressibility (C_w) 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
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 I1 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

No.	Name	Description			Default
		Field	Metric	Laboratory	
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: 3) X+, Y+, or Z+ for the positive direction, or X-, Y- or Z- for the negative direction transmissibilities. 4) 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) The keyword is followed by up to NANAQ records as defined on the AQUDIMS keyword in the RUNSPEC section
2) The keyword should be terminated by a "/".

Table 10.3: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.

Examples

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   NANAQ    NCAMAX   MXNALI   MXAAQL
AQUDIMS
          1*       1*       5        100      1        1*       1*       1*          /
```

The Fetkovich Analytical aquifer is defined in the SOLUTION sections as:

```
--
--                      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
--
AQUFETP
         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 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
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*
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 (C_t) of the aquifer, that is the rock compressibility (C_f) plus the water compressibility (C_w) 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/barsa	scc/hr/atma	None
7	PVTW	A positive integer that defines the aquifer's PVTW water property table.			1
8	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
9	TEMP	TEMP is a real positive number that defines the initial temperature of the aquifer at DATUM. This variable is ignored by OPM Flow.			
		°F	°C	°C	1*

No.	Name	Description			Default
		Field	Metric	Laboratory	

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 NANAQU on the AQUDIMS keyword in the RUNSPEC section. If the number of records are equal to NANAQU 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 10.4: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 the aquifer is connected to only hydrocarbon reservoir cells. Try commenting out the aquifer and see if this resolves the instabilities.

Examples

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  NANAQ   NCAMAX  MXNALI  MXAAQL
AQUDIMS
         1*      1*      5       100     1       1*      1*      1*          /
```

The Fetkovich Analytical aquifer is defined in the SOLUTION sections as:

```
--
--                      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 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 X- face) of the cell.

10.3.15 AQUFLUX - DEFINE CONSTANT FLUX ANALYTICAL AQUIFER

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The AQUFLUX keword defines the properties of Constant Flux Analytical Aquifers.

This keyword is ignored by OPM Flow and has no effect on the simulation.

10.3.16 BOUNDARY – DEFINE A BOUNDARY BOX FOR PRINTING

The BOUNDARY keyword defines a 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.

10.3.17 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.

10.3.18 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.

10.3.19 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.

10.3.20 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 "/".					

Table 10.5: 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

Example

```
--        DATUM
--        DEPTH
--        ------
DATUM
          5000.0                                    / DATUM DEPTH FOR REPORTING
```

The above example defines the datum for the model to be 5000.0

10.3.21 DATUMR – DEFINE DATUM DEPTHS FOR THE FIPNUM REGIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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 "/".					

Table 10.6: 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

Examples

```
--
--        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.

10.3.22 DATUMRX – DEFINE DATUM DEPTHS FOR THE FIP ALLOCATED REGIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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 a give 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 10.7: 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

Examples

```
--
--        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.

10.3.23 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.

OPM Flow does not recognize this keyword and the keyword may cause the program to terminate, but it is documented here for completeness.

10.3.24 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.

10.3.25 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 ignored by OPM Flow and has no effect on the simulation.

10.3.26 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.

10.3.27 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.

10.3.28 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.

10.3.29 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
5	GASCONT	1) For three phase runs containing oil, gas and water GASCONT is the depth of the oil-water contact (OWC). 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. 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. 2) For two phase runs containing oil and water GASCONT is ignored. 3) For two phase runs containing gas and water GASCONT is ignored.			0.0

No.	Name	Description			Default
		Field	Metric	Laboratory	
		feet	m	cm	
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 GASCAP is ignored. 3) For two phase runs containing 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 than either PBVD or RSVD keywords must be present in the input deck. Note that the allocation of multiple PBVD and RSVD tables to each grid cell is through the EQLNUM keyword and not the PVTNUM keyword. 2) 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. 3) 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
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 than either PDVD or RVVD keywords must be present in the input deck Note that the allocation of multiple PDVD and RVVD tables to each grid cell is through the EQLNUM keyword and not the PVTNUM keyword. 2) 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. 3) 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

No.	Name	Description			Default
		Field	Metric	Laboratory	
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 $2\|N\| + 1$ 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 equal to zero is supported by OPM Flow.			
		dimensionless	dimensionless	dimensionless	0
10		Not used.			
11		Not used			

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 10.8: EQUIL Keyword Description

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.

Examples

```
--
--        DATUM    DATUM    OWC      PCOW    GOC     PCGO    RS    RV    N
--        DEPTH    PRESS    DEPTH    ----    DEPTH   ----    OPT   OPT   OPT
EQUIL
          3650.0   1560.0   3712.0   0.00    1000.0  0.00    1     0     -5          /
          3650.0   1560.0   3741.0   0.00    1000.0  0.00    1     0     -5          /
          3650.0   1560.0   3741.0   0.00    1000.0  0.00    1     0     -5          /
```

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 default value of EQLOPT (-5) has been explicitly stated.

10.3.30 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.

10.3.31 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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/ft^3	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 "/".

Table 10.9: 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.

10.3.32 GASSATC – DEFINE THE INITIAL EQUILIBRATION SATURATED COAL GAS CONCENTRATION FOR ALL GRID BLOCKS

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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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/ft^3	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 "/".

Table 10.10: 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.

10.3.33 GCVD – DEFINE EQUILIBRATION COAL GAS CONCENTRATION VERSUS DEPTH TABLES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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/ft^3	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 "/" and there is no "/" terminator for the keyword.

Table 10.11: 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.

10.3.34 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

10.3.35 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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.

10.3.36 GI - DEFINE THE INITIAL EQUILIBRATION GI VALUES FOR ALL GRID BLOCKS

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 ignored by OPM Flow and has no effect on the simulation.

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.

10.3.37 HMAQUCT – HISTORY MATCH CARTER-TRACY AQUIFER GRADIENT PARAMETERS

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 ignored by OPM Flow and has no effect on the simulation.

10.3.38 HMAQUFET – HISTORY MATCH FETKOVICH AQUIFER GRADIENT PARAMETERS

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 ignored by OPM Flow and has no effect on the simulation.

10.3.39 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 ignored by OPM Flow and has no effect on the simulation.

10.3.40 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 ignored by OPM Flow and has no effect on the simulation.

10.3.41 HMMLTWCN – HISTORY MATCH WELL CONNECTION AND SKIN MULTIPLIERS

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 ignored by OPM Flow and has no effect on the simulation.

10.3.42 HMWELCON – HISTORY MATCH WELL CONNECTION AND SKIN PARAMETERS

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 ignored by OPM Flow and has no effect on the simulation.

10.3.43 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-processoring 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.

10.3.44 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.

10.3.45 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.

10.3.46 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 "/".

Table 10.12: PRESSURE 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.

10.3.47 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.

10.3.48 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.

10.3.49 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.

10.3.50 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 ignored by OPM Flow and has no effect on the simulation.

10.3.51 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 ignored by OPM Flow and has no effect on the simulation.

10.3.52 PBUB – DEFINE THE INITIAL EQUILIBRATION BUBBLE-POINT PRESSURE FOR ALL GRID BLOCKS

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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/".

Table 10.13: 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.

10.3.53 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	PBVASL	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 "/" and there is no "/" terminator for the keyword.

Table 10.14: 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 "/" and there is no keyword terminating "/".

10.3.54 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/".

Table 10.15: 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.

10.3.55 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 "/" and there is no "/" terminator for the keyword.

Table 10.16: 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    3125.0                    / 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 "/" and there is no keyword terminating "/".

10.3.56 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 "/".

Table 10.17: 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.

10.3.57 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.

OPM Flow does not recognize this keyword and the keyword may cause the program to terminate, but it is documented here for completeness.

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 "/" and there is no "/" terminator for the keyword.

Table 10.18: PRVD Keyword Description

See also the PBUB, PDEW, RS, RV, SGAS, SOIL and SWAT keywords to fully define the initial state of the model.

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 "/" and there is no keyword terminating "/".

10.3.58 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 ignored by OPM Flow and has no effect on the simulation.

10.3.59 RBEDCONT – DEFINE RIVER GRID BLOCK CONTACT AREA VERSUS DEPTH

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 ignored by OPM Flow and has no effect on the simulation.

10.3.60 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.

10.3.61 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	Not used.	None
4	RSFORMAT	Not used.	None
Notes: 1) The keyword is terminated by "/".			

Table 10.19: 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-A01DATA*, 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 keywords used to initialize the model (PRESSURE, SGAS, SOIL. SWAT, etc.) in the SOLUTION section.

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.

Examples

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.

10.3.62 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 ignored by OPM Flow and has no effect on the simulation.

10.3.63 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 paarameter is assigned a value, OPTION, using the form BASIC = 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 restart files are written at every report time step until this switch is reset and all the restarts are kept. 3) OPTION = 3 then the restart files are written every n^{th} report time step with the frequency determined by the mnemonic "FREQ=n". This feature is not currently supported by OPM Flow. 4) OPTION = 4 then the restart files are written at the first report step of each year. 5) OPTION = 5 then the restart files are written at the first report step of each month. 6) 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.	
3	DEN	Oil, gas and water fluid phases in-situ densities.	OIL_DEN GAS_DEN WAT_DEN
4	KRG	Gas relative permeability at the grid blocks gas saturation.	GASKR
5	KRO	Oil relative permeability at the grid blocks oil saturation.	OILKR
6	KRW	Water relative permeability at the grid blocks water saturation.	WATKR

No.	Name	Description	Array Name
7	RSSAT	Saturated dissolved gas-oil ratio for each grid block to enable restarts.	RSSAT
8	RVSAT	Saturated vaporized oil-gas ratio for each grid block to enable restarts.	RVSAT
9	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 "/".			

Table 10.20: RPTRST Keyword Description

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
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

No.	Restart Variable Name	Variable Description	Variable Array Name
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, currently not used in this version of OPM Flow.	Not Used
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 niether 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 10.21: Data Sets Automatically Written to the RESTART File

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                                             /
```

10.3.64 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 ignored by OPM Flow and has no effect on the simulation, but is documented here for completeness.

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 "/".

Table 10.22: 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                                     /
```

10.3.65 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 "/".

Table 10.23: 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.

10.3.66 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, RSVALS.			
		feet	m	cm	None
2	RSVALS	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 "/" and there is no "/" terminator for the keyword.

Table 10.24: 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.

10.3.67 RTEMP - DEFINE THE INITIAL RESERVOIR TEMPERATURE FOR THE MODEL

This keyword defines the reservoir temperature for when a temperature option has been activated by the either the TEMP or THERMAL keywords in the RUNSPEC section. The RTEMPA keyword is alias for RTEMP

See *RTEMP - Define the Initial Reservoir Temperature for the Model* in the PROPS section for a full description.

10.3.68 RTEMPA - DEFINE THE INITIAL RESERVOIR TEMPERATURE FOR THE MODEL

This keyword defines the reservoir temperature for when a temperature option has been activated by the either the TEMP or THERMAL keywords in the RUNSPEC section. The RTEMP keyword is alias for RTEMPA

See *RTEMPA - Define the Initial Reservoir Temperature for the Model* in the PROPS section for a full description.

10.3.69 RTEMPVD - DEFINE THE INITIAL RESERVOIR TEMPERATURE VERSUS DEPTH TABLES

Description

This keyword defines the reservoir temperature versus depth tables for when the temperature or thermal options has been activated by either the TEMP or THERMAL keywords in the RUNSPEC section in the commercial simulator. For OPM Flow the THERMAL keyword must be used to invoke the thermal option. The RTEMPVD keyword is an alias for the TEMPVD keyword; however, the latter is ignored by OPM Flow.

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 users the TEMP keyword in the RUNSPEC section to activate the "black-oil" thermal 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 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 "/" and there is no "/" terminator for the keyword.

Table 10.25: RTEMPVD Keyword Description

See also the RTEMP keyword in the PROPS section.

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.

10.3.70 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 "/".					

Table 10.26: 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.

10.3.71 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 vaporize 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, RVVALS.			
		feet	m	cm	None
2	RVVALS	A columnar vector of real values that defines the vaporized oil-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 "/" and there is no "/" terminator for the keyword.

Table 10.27: 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 bubble-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.

10.3.72 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 keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/".

Table 10.28: 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 VALUES FOR ALL CELLS IN THE MODEL
--       BASED ON NX = 100, NY = 100 AND NZ = 3
--
SALT
         1000*0.0000    1000*0.0000    1000*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.

10.3.73 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/".

Table 10.29: 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.

10.3.74 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.

This keyword is ignored by OPM Flow and has no effect on the simulation.

10.3.75 SCVD – DEFINE EQUILIBRATION COAL SOLVENT CONCENTRATION VERSUS DEPTH TABLES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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/ft^3	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 "/" and there is no "/" terminator for the keyword.

Table 10.30: 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

10.3.76 SFOAM – DEFINE THE INITIAL EQUILIBRATION FOAM CONCENTRATION FOR ALL GRID BLOCKS

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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/".

Table 10.31: 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.

10.3.77 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 "/".

Table 10.32: 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.

10.3.78 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.			
		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 "/".

Table 10.33: 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.

10.3.79 SPOLY – DEFINE THE INITIAL EQUILIBRATION POLYMER CONCENTRATION FOR ALL GRID BLOCKS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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 keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/".

Table 10.34: 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.

10.3.80 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 initialized parameters used to initialized 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 run 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.

10.3.81 SOLVCONC – DEFINE THE INITIAL COAL SOLVENT CONCENTRATION FOR ALL GRID BLOCKS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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/ft^3	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 "/".

Table 10.35: 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.

10.3.82 SOLVFRAC – DEFINE THE INITIAL GAS SOLVENT FRACTION FOR ALL GRID BLOCKS

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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/".

Table 10.36: 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.

10.3.83 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 "/".

Table 10.37: 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.

10.3.84 SURF – DEFINE THE INITIAL EQUILIBRATION POLYMER CONCENTRATION FOR ALL GRID BLOCKS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/".

Table 10.38: 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.

10.3.85 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 "/".					

Table 10.39: 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 GAS 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.

10.3.86 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 "/".

Table 10.40: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.

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

Examples

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 concentrations in both the free and solution states.

```
--
--       INITIALIZATION OF TRACER CONCENTRATIONS BY BLOCK
--
--       -- ARRAY    CONSTANT --  ---------- BOX ---------
--                                 I1  I2   J1  J2   K1  K2
EQUALS
         'TBLKFIGS'  0.0000       1*  1*   1*  1*   1*  1* / TRACER CONCENTRATION
         'TBLKSIGS'  0.0000       1*  1*   1*  1*   1*  1* / TRACER CONCENTRATION
/
```

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'     'GAS'      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.

10.3.87 TEMPI – DEFINE THE INITIAL TEMPERATURE VALUES FOR ALL CELLS

Description

TEMPI defines the initial temperature for all the cells in the model via an array for when the THERMAL option has been activated in the RUNSPEC section. This keyword is used to explicitly define the initial reservoir temperature via the Enumeration Initialization method rather than using the Equilibration Initialization method that utilizes the EQUIL keyword in the SOLUTION 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 users the TEMP keyword in the RUNSPEC section to activate the "black-oil" thermal model.

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 "/".					

Table 10.41:TEMPI Keyword Description

See also the RTEMP and RTEMPA keywords in the PROPS section and the RTEMPVD 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 °F for the first, second and third layers in the model for all 300 cells, as defined by the DIMENS keyword in the RUNSPEC section.

10.3.88 TEMPVD - DEFINE THE INITIAL RESERVOIR TEMPERATURE VERSUS DEPTH TABLES

Description

This keyword defines the reservoir temperature versus depth tables for when the temperature or thermal options has been activated by either the TEMP or THERMAL keywords in the RUNSPEC section in the commercial simulator. For OPM Flow the THERMAL keyword must be used to invoke the thermal option.

The TEMPVD keyword is an alias for the RTEMPVD keyword; however, the former is ignored by OPM Flow.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	DEPTH	A real monotonically increasing down the column values that defines the depth for corresponding reservoir temperature RTEMP.			
		feet	m	cm	None
2	RTEMP	A 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 EQLDIMS keyword in the RUNSPEC section.
3) Each table is terminated by "/" and there is no "/" terminator for the keyword.

Table 10.42:TEMPVD Keyword Description

See also the RTEMP keyword in the PROPS section and RTEMPVD keyword in the SOLUTION section.

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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.

10.3.89 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 "/".
2) The keyword is terminated by "/".

Table 10.43: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.

10.3.90 TVDP – DEFINE THE INITIAL EQUILIBRATION TRACER SATURATION VERSUS DEPTH FUNCTIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword defines the tracer name as the subsequent four characters after TVDP characters of the keyword, and then defines the tracer saturation as a function of depth.

This keyword is ignored by OPM Flow and has no effect on the simulation.

10.3.91 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 decrease the rate of vaporization. Typical values for VAPPAR1 range from zero and 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 "/".

Table 10.44: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.

10.3.92 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 ignored by OPM Flow and has no effect on the simulation.

10.3.93 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 ignored by OPM Flow and has no effect on the simulation.

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 plot pressure versus time, etc. The properties to be stored on the summary file are entered in the SUMMARY section of the input file. The items requested are written to the summary file at the end of each successful time step. The SUMMARY section is terminated by the SCHEDULE keyword.

To generate a summary vector, the appropriate keyword must be entered in the SUMMARY section of the input data file. The SUMMARY keywords are listed below.

To be written in a future release of the manual.

11.2 DATA REQUIREMENTS

To be written in a future release of the manual.

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. The SUMMARY files are use by post processing software, for example OPM ResInsight, to plot the results of the simulation. Theses files are normally written in binary format in order to save hard disk space. whereas the RSM file is a tabulated format of the same data and is always written in ASCII format.

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

Examples

```
--
--        EXPORT STANDARD SUMMARY VARIABLE VECTORS TO FILE
--
ALL
--
--        ACTIVATE COLUMNAR SUMMARY DATA REPORTING OPTION
--
RUNSUM
--
--        ACTIVATE SUMMARY DATA RSM FILE OUTPUT OPTION
--
SEPARATE
```

Note currently OPM Flow does not write out RSM files.

11.3.2 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, activating the DATE option also results in the DATE being written out to the SUMMARY file as well. This option is normally used when 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.

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 the 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
.....
```

Note currently OPM Flow does not write out RSM files.

11.3.3 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

Examples

```
--
--       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

Note currently OPM Flow does not write out RSM files.

11.3.4 FWSET - EXPORT WELL STATUS VECTORS FOR THE FIELD TO FILE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This 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.

There is no data required for this keyword.

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

11.3.5 GMWSET - EXPORT WELL STATUS VECTORS BY GROUP TO FILE

Description

This keyword activates the writing out of a set of data vectors that give the production and injections status of all the wells within a group.

There is no data required for this keyword.

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

11.3.6 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.

11.3.7 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 page".

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

This keyword is ignored by OPM Flow and has no effect on the simulation.

11.3.8 NEWTON – ACTIVATE NEWTON ITERATION SUMMARY OUTPUT

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword activates the writing out of the Newtonian 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.

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

Examples

```
--
--       ACTIVATE NEWTON ITERATION SUMMARY OUTPUT
--
NEWTON
```

Note currently OPM Flow does not write out RSM files.

11.3.9 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.

11.3.10 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 ignored by OPM Flow and has no effect on the simulation.

11.3.11 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 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 and there is no terminating "/" for this keyword.

The option can be deactivated by the RPTONLYO keyword in the SUMMARY section.

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

Examples

```
--
--       ACTIVATE REPORT TIME STEPS ONLY SUMMARY FILE OPTION
--
RPTONLY
```

Note currently OPM Flow does not write out RSM files.

11.3.12 RPTONLYO - DEACTIVATE THE REPORT TIME STEPS ONLY OPTION FOR THE SUMMARY FILE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

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 and there is no terminating "/" for this keyword.

The option can be activated 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..

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

Examples

```
--
--       DEACTIVATE REPORT TIME STEPS ONLY SUMMARY FILE OPTION
--
RPTONLYO
```

Note currently OPM Flow does not write out RSM files.

11.3.13 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 RSM file, if the RSM file option has been requested by the RUNSUM keyword in the SUMMARY section.

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 "/".

Table 11.1: RPTSMRY Keyword Description

Examples

```
--
--       ACTIVATE OR DEACTIVATE SUMMARY LIST REPORT
--
RPTSMRY
         1                                                  /
```

The example switches on the summary list report.

11.3.14 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.

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

Examples

```
--
--       ACTIVATE COLUMNAR SUMMARY DATA REPORTING OPTION
--
RUNSUM
```

Note currently OPM Flow does not write out RSM files.

11.3.15 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 been 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.

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

Examples

```
--
--       ACTIVATE SUMMARY DATA RSM FILE OUTPUT OPTION
--
SEPARATE
```

Note currently OPM Flow does not write out RSM files.

11.3.16 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.

11.3.17 SUMTHIN – DEFINE SUMMARY DATA REPORTING TIME STEPS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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 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.

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 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 "/".					

Table 11.2: 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 SUMSTEPS time intervals.
This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

Examples

```
--
--       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.

12 SCHEDULE SECTION

12.1 INTRODUCTION

To be written in a future release of the manual.

12.2 DATA REQUIREMENTS

To be written in a future release of the manual.

12.2.1 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:

$$q = (0.001127)(2\pi)\frac{kh}{\mu}\left[\frac{p_e - p_w}{\left(\ln\left(\frac{r_e}{r_w}\right) + S\right)}\right] \quad (12.1)$$

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:

$$q_o = (0.001127)(2\pi)\frac{kk_{ro}\,h}{B_o\,\mu_o}\left[\frac{p_e - p_w}{\left(\ln\left(\frac{r_e}{r_w}\right) + S\right)}\right] \quad (12.2)$$

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[131]

$$r_b = 0.28 \frac{\left[\sqrt{\frac{K_y}{K_x}}\,\Delta x^2 + \sqrt{\frac{K_x}{K_y}}\,\Delta y^2\right]^{0.5}}{\left(\frac{K_y}{K_x}\right)^{0.25} + \left(\frac{K_x}{K_y}\right)^{0.25}} \tag{12.3}$$

Where:

K_x and K_y = x- and y- direction absolute permeabilities
Δx and Δy = grid block sizes (feet)

The simulator then uses the modified radial flow equation, written for each connection:

$$q_o = \frac{(0.001127)\,(2\pi)}{\left(\ln\left(\frac{r_b}{r_w}\right) + S\right)}\left(\frac{k\,k_{ro}\,h}{B_o\,\mu_o}\,(p_b - p_w)\right) \tag{12.4}$$

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:

$$q_o = \sum_1^N \left(T_{xy}\,M_o\,(p_b - p_w)\right) \tag{12.7}$$

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:

[131] *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.*

$$q_o = \frac{PI}{\left(p_e - p_w \right)} \tag{12.8}$$

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

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 ignored by OPM Flow and has no effect on the simulation; use the ACTIONX keyword instead.

12.3.2 ACTIONG – DEFINE ACTION CONDITIONS AND COMMAND PROCESSING (GROUPS)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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 ignored by OPM Flow and has no effect on the simulation; use the ACTIONX keyword instead.

12.3.3 ACTIONR – DEFINE ACTION CONDITIONS AND COMMAND PROCESSING (REGIONS)

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 ignored by OPM Flow and has no effect on the simulation; 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

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.

OPM Flow does not recognize this keyword and the keyword may cause the program to terminate, but it is documented here for completeness; use the ACTIONX keyword instead.

12.3.5 ACTIONW – DEFINE ACTION CONDITIONS AND COMMAND PROCESSING (WELLS)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

The ACTIONW 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 well level.

This keyword is ignored by OPM Flow and has no effect on the simulation; use the ACTIONX keyword instead.

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 implements quite complex functionality and the implementation is new for the 2019-10 release and it has not been subject to exhaustive testing. Users are encouraged to test the ACTIONX keyword but should read the manual carefully and start with simple expressions.

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.	None
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
1-4	/	Record terminated by a "/"	Not Applicable

No.	Name	Description	Default
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 12.2.	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 12.2 (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`
		Most SCHEDULE keyword can used in an ACTIONX Define Section here, except for the time stepping keywords, i.e, TSTEP and DATES.
		Note only the WELOPEN keyword is currently supported by OPM Flow.
			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 12.1:ACTIONX Keyword Description

The variable types and the associated definitions that are available for use with Boolean conditionals are outlined in Table 12.2.

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).

TIME	TIME variables consists of one parameter that can have three values:
	1) 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 January 1, 2025 one would: use
	`ACTIONX`
	`DAY = 1 AND /`
	`MNTH ='JAN' AND /`
	`YEAR = 2025 /`
	`/`
	`.....`
	`ENDACTIO`

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`

Variable Type	Description
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.
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 12.2: 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.

Examples

The first example users 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
WSHUTIN         10                                                        /
        GWPR  'FIELD' > 30E3  AND                                           /
        WU_LIST 'OP*' > 1     AND                                           /
/

--
--       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.

OPM Flow does not recognize this keyword and the keyword may cause the program to terminate, but it is documented here for completeness.

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 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 ignored by OPM Flow and has no effect on the simulation.

12.3.13 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.

12.3.14 AQUFLUX - DEFINE CONSTANT FLUX ANALYTICAL AQUIFER

The AQUFLUX keword defines the properties of Constant Flux Analytical Aquifers.

See *AQUFLUX - Define Constant Flux Analytical Aquifer* in the SOLUTION section for a full description.

12.3.15 BOUNDARY – DEFINE A BOUNDARY BOX FOR PRINTING

The BOUNDARY keyword defines a 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.

12.3.16 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.

12.3.17 BRANPROP – DEFINE NETWORK BRANCH PROPERTIES FOR EXTENDED NETWORK OPTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

BRANPROP defines network branch properties for the extended network option for when the Extended Network option has been activated by the NETWORK keyword in the RUNSPEC section.

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

12.3.18 CALTRAC – DEFINE A GAS CALORFIC VALUE TRACER

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 ignored by OPM Flow and has no effect on the simulation.

12.3.19 CECON – DEFINE WELL CONNECTIONS ECONOMIC LIMIT CRITERIA

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

CECON sets the economic cut-off criteria for a well's connection to the simulation grid.

This keyword is ignored by OPM Flow and has no effect on the simulation.

12.3.20 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.

OPM Flow does not recognize this keyword and the keyword may cause the program to terminate, but it is documented here for completeness.

12.3.21 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 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, that defines the saturation table number to be used for flow between the reservoir gird block and the well connections. If SATNUM is set to zero or defaulted with 1* then: • The saturation table allocated to the grid block that the connections are located within is used. • 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	RW	A real positive value that defines the well bore diameter of the connections for the well. RW 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
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 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 12.3: 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   28   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).

12.3.22 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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
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: 4) OPEN: the connections are open to flow. 5) SHUT: the connections are closed to flow (shut-in). 6) 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	
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 gird block and the well connections. If SATNUM is set to zero or defaulted with 1* then: • The saturation table allocated to the grid block that the connections are located within is used. • 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	RW	A real positive value that defines the well bore diameter of the connections for the well. RW 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.			
		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*

No.	Name	Description			Default
		Field	Metric	Laboratory	
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 12.4: 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).

12.3.23 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 ignored by OPM Flow and has no effect on the simulation.

12.3.24 COMPIMB – ASSIGN IMBIBITION SATURATION TABLES TO WELL CONNECTIONS

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.

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
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 gird 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

No.	Name	Description			Default
		Field	Metric	Laboratory	
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 12.5: 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.
This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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.

12.3.25 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 ignored by OPM Flow and has no effect on the simulation.

12.3.26 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 12.6: 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     1*   1*   20   56   OPEN    1*     1*     0.708    3*      Z /
          OP01     OP01LGR     1*   1*   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.

12.3.27 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 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 low most 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 12.7: 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       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' /
/
--
--        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 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.

12.3.28 COMPOFF – DEACTIVATE NETWORK AUTOMATIC COMPRESSORS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The COMPOFF keyword deactivates network automatic compressors defined via the GASFCOMP keyword in the SCHEDULE section.

This keyword is ignored by OPM Flow and has no effect on the simulation.

12.3.29 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.			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 12.8: 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.

12.3.30 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 ignored by OPM Flow and has no effect on the simulation.

12.3.31 COMPRP – RE-SCALE FLUID SATURATIONS OF WELL CONNECTIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The COPMPRP keyword re-scales the fluid saturations of a well's connection to the grid block.

This keyword is ignored by OPM Flow and has no effect on the simulation.

12.3.32 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 ignored by OPM Flow and has no effect on the simulation.

12.3.33 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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
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

No.	Name	Description			Default
		Field	Metric	Laboratory	
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

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 12.9: 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.

12.3.34 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: 4) a value between -NX and +NX that is not equal to zero that defines the last connection location in the I-direction, 5) a value between -NY and +NY that is not equal to zero that defines the last connection location in the J-direction, or 6) 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 12.10: 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 defines for these two wells.

12.3.35 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 ignored by OPM Flow and has no effect on the simulation.

12.3.36 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 ignored by OPM Flow and has no effect on the simulation.

12.3.37 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 ignored by OPM Flow and has no effect on the simulation.

12.3.38 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 ignored by OPM Flow and has no effect on the simulation.

12.3.39 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.

This keyword is ignored by OPM Flow and has no effect on the simulation.

12.3.40 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 "/" and the keyword is terminated by a "/".			

Table 12.11: DATES Keyword Description

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.

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.

12.3.41 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 ignored by OPM Flow and has no effect on the simulation.

12.3.42 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 ignored by OPM Flow and has no effect on the simulation.

12.3.43 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.

12.3.44 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 users 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 2019-10 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.

12.3.45 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 users 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 2019-10 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.

12.3.46 DRILPRI – DEFINE PRIORITIZED DRILLING QUEUE PRIORITY PARAMETERS

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 ignored by OPM Flow and has no effect on the simulation.

12.3.47 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/day	$sm^3/sm^3/day$	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 "/".					

Table 12.12: 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/day 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.

12.3.48 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/day	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 "/" and there is no "/" terminator for the keyword.

Table 12.13: 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 ignored by OPM Flow and has no effect on the simulation 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/day 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.

12.3.49 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/day	sm^3/sm^3/day	scc/scc/day	None
Notes: 1) The keyword is terminated by "/".					

Table 12.14: 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.

Examples

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.

12.3.50 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/day	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.
1) Each record is terminated by "/" and there is no "/" terminator for the keyword.

Table 12.15: 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/day 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.

12.3.51 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 ignored by OPM Flow and has no effect on the simulation.

12.3.52 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.

12.3.53 ENDACTIO – END THE DEFINITION OF ACTION COMMANDS

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
```

12.3.54 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.

12.3.55 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.

12.3.56 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.

12.3.57 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.

12.3.58 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.

12.3.59 EXCAVATE - SET THE STATUS OF A GRID BLOCK TO ACTIVE OR EXCAVATE

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 ignored by OPM Flow and has no effect on the simulation.

12.3.60 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 ignored by OPM Flow and has no effect on the simulation.

12.3.61 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.

12.3.62 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 ignored by OPM Flow and has no effect on the simulation.

12.3.63 GASEND – DEFINE END OF ANNUAL SCHEDULING SECTION

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 ignored by OPM Flow and has no effect on the simulation.

12.3.64 GASFCOMP – DEFINE AUTOMATIC GAS COMPRESSORS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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 ignored by OPM Flow and has no effect on the simulation.

12.3.65 GASFDECR – DEFINE FIELD GAS SALES CONTRACT MONTHLY REDUCTION

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:

$$Q_{month} = DCQ \times SWINGFAC_{month} \tag{12.13}$$

Where:

Q_{month} = 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 (12.13) become:

$$Q_{month} = \left(DCQ \times SWINGFAC_{month}\right) - GASFDECR_{month} \tag{12.14}$$

Where:

Q_{month} = 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 ignored by OPM Flow and has no effect on the simulation.

12.3.66 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 ignored by OPM Flow and has no effect on the simulation.

12.3.67 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:

$$Q_{month} = DCQ \times SWINGFAC_{month} \tag{12.15}$$

Where:

Q_{month} = 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 (12.15) become:

$$Q_{month} = Minimum\left(\left(DCQ \times SWINGFAC_{month}\right),\ GASFTARG_{month}\right) \tag{12.16}$$

Where:

Q_{month} = 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 ignored by OPM Flow and has no effect on the simulation.

12.3.68 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 ignored by OPM Flow and has no effect on the simulation.

12.3.69 GASPERIO – ADVANCE SIMULATION BY GAS CONTRACT PERIOD

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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 ignored by OPM Flow and has no effect on the simulation.

12.3.70 GASYEAR – ADVANCE SIMULATION BY GAS CONTRACT YEAR

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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 ignored by OPM Flow and has no effect on the simulation.

12.3.71 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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

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 12.16: 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 10^9 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.

12.3.72 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 1×10^{20} switches off calorific control for the group.			
		Btu/Mscf	kJ/sm^3	J/hour	1.0×10^{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 12.17: GCONCAL Keyword Description

See also the GRUPTREE keyword to define the hierarchy of the groups below the FIELD level, the GCONPROD and GCONINJ 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 WCONINEJ 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 10^3 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.

12.3.73 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 10^{20} switches off the energy target for the group.			
		BTU/day	kJ/day	J/hour	1.0 x 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 12.18: GCONENG Keyword Description

See also the GRUPTREE keyword to define the hierarchy of the groups below the FIELD level, the GCONPROD and GCONINJ 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 WCONINEJ 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 10^9 Btu/day for the field.

12.3.74 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 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 groups 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 groups water production multiplied by item (6). 6) VREP: the target is set to the groups 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 sm^3/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.			
		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 REINJ 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.			
		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 control by the parameters for this group. This variable is ignored if GRPNAME is equal to FIELD. This option is currently not supported by OPM Flow.			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 This feature is not supported by OPM Flow and should be defaulted with 1*.			
		dimensionless	dimensionless	dimensionless	None
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) RATE: the guide phase is set to the surface injection rate. 2) RESV: the guide phase is set to the in situ reservoir volume rate. 3) VOID: the guide rate is calculated at the beginning of each time step based on the group's net voidage rate. This feature is not supported by OPM Flow and should be defaulted with 1*.			None

No.	Name	Description			Default
		Field	Metric	Laboratory	
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.			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.			GRPNAME
13		Not used should be defaulted with 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 12.19: 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 m^3/day total water injection limit.

12.3.75 GCONPRI – GROUP PRODUCTION PRIORITY TARGETS AND CONSTRAINTS

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.

OPM Flow does not recognize this keyword and the keyword may cause the program to terminate, but it is documented here for completeness.

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
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"

No.	Name	Description			Default
		Field	Metric	Laboratory	
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/day	rm^3/day	rcc/hour	None
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 12.20: 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 GCONINJ keyword to define a group's injection targets and constraints, the WCONPROD keyword to define a production well's targets and constraints, and the WCONINEJ 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.

12.3.76 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 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).			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
5	GAS	A real positive value that defines the maximum surface gas production rate target or constraint			
		Mscf/d	sm^3/day	scc/hour	None

No.	Name	Description			Default
		Field	Metric	Laboratory	
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) 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.			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 control by the parameters for this group. This feature is currently not supported by OPM Flow.			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 This feature is not supported by OPM Flow and should be defaulted with 1*.			
		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) ORAT: the guide phase is set to the surface oil production rate. 2) WRAT: the guide phase is set to the surface water production rate. 3) GRAT: the guide phase is set to the surface gas production rate. 4) LRAT: the guide phase is set to the surface liquid (oil plus water) production rate. 5) RESV: the guide phase is set to the in situ reservoir volume rate. This feature is not supported by OPM Flow and should be defaulted with 1*.			1*
11		Not used should be defaulted with 1*.			1*
12		Not used should be defaulted with 1*.			1*
13		Not used should be defaulted with 1*.			1*
14	RESV	A real positive value that defines the maximum reservoir volume production rate target or constraint.			
		rtb/d	rm^3/day	rcc/hour	None
15		Not used should be defaulted with 1*.			1*
16		Not used should be defaulted with 1*.			1*
17		Not used should be defaulted with 1*.			1*
18		Not used should be defaulted with 1*.			1*
19		Not used should be defaulted with 1*.			1*
20		Not used should be defaulted with 1*.			1*
21		Not used 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 12.21: GCONPROD Keyword Description

See also the GRUPTREE keyword to define the hierarchy of the groups below the FIELD level, the GCONINJ keyword to define a group's injection targets and constraints, the WCONPROD keyword to define a production well's targets and constraints, and the WCONINEJ 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.

12.3.77 GCONSALE – DEFINE GROUP SALES GAS PRODUCTION TARGETS AND CONSTRAINTS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

GCONSALE defines the 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.

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, that is:

$$\begin{aligned}\text{Gas Sales Rate} = {} & \text{Total Group Gas Production Rate} \\ & - \text{Group Gas Injection Rate} \\ & + \text{Total Group Gas Import Rate} \\ & - \text{Total Group Gas Consumption}\end{aligned} \tag{12.17}$$

This keyword is ignored by OPM Flow and has no effect on the simulation.

12.3.78 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 also sets the group gas import rate, if required. This is used to import gas 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 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.

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:

$$\begin{aligned}\text{Gas Sales Rate} = {} & \text{Total Group Gas Production Rate} \\ & - \text{Group Gas Injection Rate} \\ & + \text{Total Group Gas Import Rate} \\ & - \text{Total Group Gas Consumption}\end{aligned} \tag{12.18}$$

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:

$$\begin{aligned}\text{Group Gas Injection Rate} = {} & \text{Group Gas Injection Rate} \times \text{Group Re-Injection Fraction} \\ & + \text{Total Group Gas Import Rate} \\ & - \text{Total Group Gas Consumption}\end{aligned} \tag{12.19}$$

This keyword is ignored by OPM Flow and has no effect on the simulation.

12.3.79 GCONTOL – DEFINE GROUP CONSTRAINT TOLERANCE

Description

The GCONTOL keyword defines the tolerance and parameters used to control the accuracy of group targets and constraints. The keyword sets the tolerance and number of Newtonian iterations for each time step so that the wells under group control can match the desired group targets and constraints. See also the WLIMTOL keyword in the SCHEDULE section that controls the tolerance for wells.

This keyword is ignored by OPM Flow and has no effect on the simulation.

12.3.80 GCUTBACK – DEFINE GROUP CUTBACK LIMITS AND PARAMETERS

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 ignored by OPM Flow and has no effect on the simulation.

12.3.81 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 ignored by OPM Flow and has no effect on the simulation.

12.3.82 GDCQ – DEFINE GROUP MULTIPLE DAILY CONTRACT QUANTITIES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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 ignored by OPM Flow and has no effect on the simulation.

12.3.83 GDCQECON – GROUP ECONOMIC CRITERIA FOR DCQ PRODUCTION GROUPS

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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 12.22: 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                                                    /
/
```

12.3.84 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 ignored by OPM Flow and has no effect on the simulation.

12.3.85 GECON – GROUP ECONOMIC CRITERIA FOR PRODUCTION GROUPS

Description

The GECON keyword defines economic criteria for 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 item (9) of the WELSPECS keyword. A value less than or equal to zero switches of this criteria.			
		stb/day	sm^3/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 item (9) of the WELSPECS keyword. A value less than or equal to zero switches of this criteria,			
		Mscf/day	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 of this criteria.			
		dimensionless	dimensionless	dimensionless	0.0

No.	Name	Description			Default
		Field	Metric	Laboratory	
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 of this criteria.			
		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 of this criteria.			
		stb/Mscf	sm^3/sm^3	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. 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
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
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.			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 12.23: GECON 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 economic criteria for the field with a minimum oil rate of 2,000 m^3/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.

12.3.86 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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		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		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		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		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		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		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 12.24: 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.

12.3.87 GEFAC – DEFINE GROUP EFFICIENCY

Description

Defines a group's efficiency or up-time as opposed to setting the efficient 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 efficient 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 efficient 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	NETOPTN	Not used			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 12.25: GEFAC Keyword Description

See also the WELFAC 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.

12.3.88 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 resources available 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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	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 12.26: GCONENG Keyword Description

See also the GRUPTREE keyword to define the hierarchy of the groups below the FIELD level, the GCONPROD and GCONINJ 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 WCONINEJ 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 well 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.

12.3.89 GLIFTOPT – DEFINE GROUP GAS OPTIMIZATION LIMITS

Description

GLIFTOPT defines a group's gas rate limits for when gas lift optimization has been activated via the LIFTOPT keyword in the SCHEDULE section.

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

12.3.90 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 ignored by OPM Flow and has no effect on the simulation.

12.3.91 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

12.3.92 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

12.3.93 GPMAINT – DEFINE GROUP PRESSURE MAINTENANCE TARGETS AND CONTROLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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.

This keyword is ignored by OPM Flow and has no effect on the simulation.

12.3.94 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 ignored by OPM Flow and has no effect on the simulation.

12.3.95 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 ignored by OPM Flow and has no effect on the simulation.

12.3.96 GRADRFT – DEFINE RFT DERIVATIVE HISTORY MATCH GRADIENT OUTPUT

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 ignored by OPM Flow and has no effect on the simulation.

12.3.97 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 ignored by OPM Flow and has no effect on the simulation.

12.3.98 GRDREACH – DEFINE RIVER AND GRID BLOCK CONNECTIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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 ignored by OPM Flow and has no effect on the simulation.

12.3.99 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 ignored by OPM Flow and has no effect on the simulation.

12.3.100 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 group to the HIGHER given the current flowing conditions; the group relationship is defined by the GRUPTREE keyword. The VPP table is entered via the VFPPROD keyword for production pipelines and VFPINJ for injection pipelines. Although these keywords are the same as used for well modeling, they are also used for pipeline modeling as well; however, the manner in which they are generated by an external software is completely different.

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 group 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 networf. 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 VFPPROD assigned to the group via VPFTAB variable. VFPTAB vertical lift performance table and the artificial lift quantity ALQ-WELL are used with the pipeline fluid rates to calculate the pipeline pressures between the LOWER and HIFGHER groups. Note that the units for ALQ-PIPE is dependent on the associated variable on the VFPPROD keyword and may represent a pump or a compressor depending how the VFPPROD table was generated 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, than this group should have 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.			NO

No.	Name	Description			Default
		Field	Metric	Laboratory	
6	OPTION2	A defined character string that defines if how gas lift gas flows through the group's pipeline. OPTION2 should be set to one of the following character strings: 1) NO: no gas lift gas is allowed to flow through the pipeline only produced reservoir gas is allowed to flow through the pipeline. 2) FLO: both gas lift gas and produced reservoir gas are allowed to flow through the pipeline. Gas lift gas is calculated from summing the calculated gas lift values of all the subordinate wells. Here the summed well gas lift gas (ALQ-WELL), the pipeline gas lift gas (ALQ-PIPE), and the reservoir produced gas are added to the gas flow rate along the pipeline. 3) ALQ: both gas lift gas and produced reservoir gas are allowed to flow through the pipeline. Gas lift gas is calculated from summing the calculated gas lift values of all the subordinate wells. Here the summed ALQ-WELL gas lift gas is added to the reservoir produced gas flow rate along the pipeline. This means that 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.			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: 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. 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.			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 12.27: 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.

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

Example

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 Figure 12.1)

12.3.101 GRUPRIG – GROUP DRILLING AND WORKOVER RIG SPECIFICATIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

Defines a groups drilling and workover specifications.

OPM Flow does not recognize this keyword and the keyword may cause the program to terminate.

12.3.102 GRUPSLAV – DEFINE SLAVE GROUPS IN SLAVE RESERVOIRS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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 ignored by OPM Flow and has no effect on the simulation.

12.3.103 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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	sm^3/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 12.28: 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.

12.3.104 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 12.29: 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.

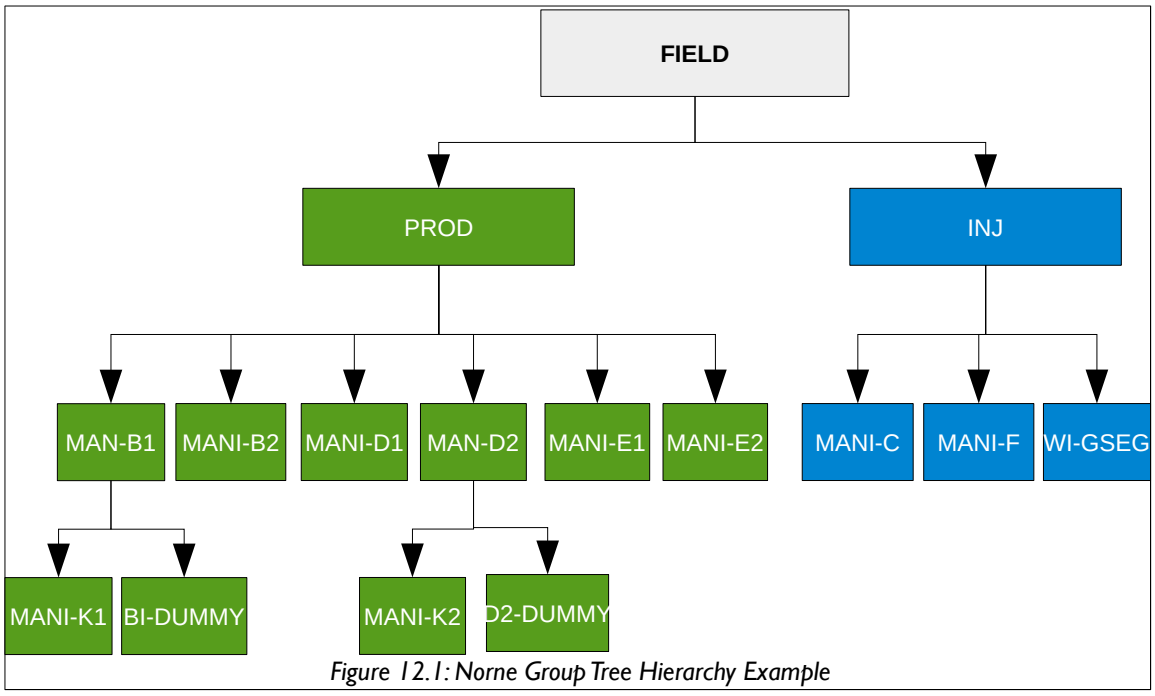


Figure 12.1: Norne Group Tree Hierarchy Example

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.

12.3.105 GSATINJE – DEFINE GROUP SATELLITE INJECTION RATES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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 supplying gas to a power plant (field A, B and C), but only one is being modeled in the current import 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 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. See the GSATPROD keyword in the SCHEDULE section to define satellite production rates.

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

12.3.106 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 gas 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 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. See the GSATINJE keyword in the SCHEDULE section to define satellite injection rates.

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

12.3.107 GSEPCOND – ASSIGN GROUP SEPARATORS

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 (B_{ob}) and the initial saturated gas-oil ratio (R_{si}) entered on the SEVPALS keyword.

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

12.3.108 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 ignored by OPM Flow and has no effect on the simulation.

12.3.109 GSWINGF – DEFINE GROUP MULTIPLE GAS CONTRACT PARAMETERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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:

$$Q_{month} = DCQ \times SWINGFAC_{month} \tag{12.20}$$

Where:

Q_{month} = 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 ignored by OPM Flow and has no effect on the simulation.

12.3.110 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 ignored by OPM Flow and has no effect on the simulation.

12.3.111 GTMULT – MULTIPLY GROUP TARGET OR CONSTRAINT BY A CONSTANT

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 ignored by OPM Flow and has no effect on the simulation.

12.3.112 GUIDECAL – SCALE GUIDE RATES BASED ON GAS CALORIFIC VALUE

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 ignored by OPM Flow and has no effect on the simulation.

12.3.113 GUIDERAT – DEFINE GROUP GUIDE RATE FORMULA

Description

This keyword defines a general formulae used to define a groups guide rate as a function of the group potential.

This keyword is ignored by OPM Flow and has no effect on the simulation.

12.3.114 GUPFREQ – INSTANTANEOUS GRADIENT OPTION UPDATE FREQUENCY

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 ignored by OPM Flow and has no effect on the simulation.

12.3.115 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 ignored by OPM Flow and has no effect on the simulation.

12.3.116 HMWPIMLT – HISTORY MATCH WELL PRODUCTIVITY INDEX PARAMETERS

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 ignored by OPM Flow and has no effect on the simulation.

12.3.117 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 12.30: 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*                                                     /
/
```

12.3.118 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 12.31: 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*                                                     /
/
```

12.3.119 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 12.32: 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                                                 /
/
```

12.3.120 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 12.33: 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                                                 /
/
```

12.3.121 LIFTOPT – ACTIVATE GAS LIFT OPTIMIZATION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The LIPTOPT keyword actives the gas lift optimization option.

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

12.3.122 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 ignored by OPM Flow and has no effect on the simulation.

12.3.123 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 ignored by OPM Flow and has no effect on the simulation.

12.3.124 MESSOPTS – RESET SEVERITY LEVEL FOR FORCED TIME STEPS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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 ignored by OPM Flow and has no effect on the simulation.

12.3.125 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 ignored by OPM Flow and has no effect on the simulation.

12.3.126 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 ignored by OPM Flow and has no effect on the simulation.

12.3.127 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.

12.3.128 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.

12.3.129 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.

12.3.130 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.

12.3.131 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 ignored by OPM Flow and has no effect on the simulation.

12.3.132 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 ignored by OPM Flow and has no effect on the simulation.

12.3.133 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.

12.3.134 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.

12.3.135 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.

12.3.136 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.

12.3.137 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.

12.3.138 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.

12.3.139 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.

12.3.140 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.

12.3.141 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 ignored by OPM Flow and has no effect on the simulation.

12.3.142 NEFAC - NODE EFFICIENCY FACTORS (EXTENDED NETWORK)

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 ignored by OPM Flow and has no effect on the simulation.

12.3.143 NETBALAN – NETWORK BALANCING PARAMETERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword defines the network balancing parameters used to control how network balancing is perform on a network.

OPM Flow does not have this feature and hence this keyword is ignored by OPM Flow and the NETBALAN keyword has no effect on the simulation.

12.3.144 NETCOMPA – DEFINE AUTOMATIC COMPRESSORS (EXTENDED NETWORK)

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 ignored by OPM Flow and has no effect on the simulation.

12.3.145 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 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 "/".					

Table 12.34: 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness. See section *2.2 Running OPM Flow 2019-10 From The Command Line* on how to control time stepping for OPM Flow.

Examples

```
--       NEXT   ALL
--       STEP   TIME
--       ----   ----
NEXTSTEP
         1     'NO'                                        /
```

Here the next step size is set to one day and should only be used once.

12.3.146 NEXTSTEP – MAXIMUM NEXT TIME STEP SIZE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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. 3) NO: Means that NSTEP1 should not be applied to subsequent reporting time steps. 4) 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 "/".

Table 12.35: NEXTSTEP 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness. See section *2.2 Running OPM Flow 2019-10 From The Command Line* on how to control time stepping for OPM Flow.

Examples

The first example shows the direct use of the NEXTSTEP keyword:,

```
--       NEXT   ALL
--       STEP   TIME
--       ----   ----
NEXTSTEP
         1     'NO'                                        /
```

Here the next step size is set to one day and should only be used once.

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.

12.3.147 NEXTSTPL – MAXIMUM NEXT TIME STEP SIZE (LGR)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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. 5) NO: Means that NSTEP1 should not be applied to subsequent reporting time steps. 6) 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 "/".					

Table 12.36: 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness. See section *2.2 Running OPM Flow 2019-10 From The Command Line* on how to control time stepping for OPM Flow.

Examples

```
--
--        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.

12.3.148 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 option has been invoked by the NETWORK keyword in the RUNSPEC secction.

This keyword is ignored by OPM Flow and has no effect on the simulation.

12.3.149 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.

12.3.150 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.

12.3.151 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

12.3.152 NUPCOL – DEFINE THE NUMBER OF NEWTONIAN ITERATIONS USED TO UPDATE WELL TARGETS

The NUPOL keyword defines the maximum number of Newtonian 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 Number of Newtonian Iterations Used to Update Well Targets* in the RUNSPEC section for a full description and also section *2.2 Running OPM Flow 2019-10 From The Command Line* on how to set various other numerical control parameters for OPM Flow.

12.3.153 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 ignored by OPM Flow and has no effect on the simulation.

12.3.154 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.

12.3.155 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")[132] and [133] 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 ignored by OPM Flow and has no effect on the simulation.

[132] *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)*

[133] *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).*

12.3.156 PIMULTAB – DEFINE WELL PRODUCTIVITY INDEX VERSUS WATER CUT TABLES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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 "/" and there is no "/" terminator for the keyword.

Table 12.37: 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 /
```

12.3.157 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.

12.3.158 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.

12.3.159 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.

12.3.160 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 ignored by OPM Flow and has no effect on the simulation.

12.3.161 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.

12.3.162 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.

12.3.163 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.

12.3.164 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.

12.3.165 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.

12.3.166 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.

12.3.167 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:

$$\text{Priority} = \frac{a_1 + a_2 Q_{oil} + a_3 Q_{water} + a_4 Q_{gas}}{b_1 + b_2 Q_{oil} + b_3 Q_{water} + b_4 Q_{gas}} \quad (12.21)$$

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 ignored by OPM Flow and has no effect on the simulation.

12.3.168 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 ignored by OPM Flow and has no effect on the simulation.

12.3.169 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 ignored by OPM Flow and has no effect on the simulation.

12.3.170 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.

12.3.171 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 ignored by OPM Flow and has no effect on the simulation.

12.3.172 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 ignored by OPM Flow and has no effect on the simulation.

12.3.173 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 ignored by OPM Flow and has no effect on the simulation.

12.3.174 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.

12.3.175 RIVDEBUG – DEFINE THE DEBUG DATA TO BE PRINTED TO FILE (RIVERS)

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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

12.3.176 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.

12.3.177 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 ignored by OPM Flow and has no effect on the simulation.

12.3.178 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 ignored by OPM Flow and has no effect on the simulation.

12.3.179 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 ignored by OPM Flow and has no effect on the simulation.

12.3.180 RPTHMG - DEFINE WELL HISTORY MATCH GRADIENT REPORTING (GROUPS)

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 ignored by OPM Flow and has no effect on the simulation.

12.3.181 RPTHMW - DEFINE WELL HISTORY MATCH GRADIENT REPORTING (WELLS)

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 ignored by OPM Flow and has no effect on the simulation.

12.3.182 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.

12.3.183 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.

12.3.184 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.

12.3.185 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 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 define the data in the OPM Flow input deck, for example WELSPECS to defined the basic well definitions. 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.

For the purpose of controlling PRT output, this keyword is ignored by OPM Flow and has no effect on the simulation. However, the keyword can also be used to control the writing of restart files and those parts of this keyword are honored.

No.	Name	Description	Default
1	FIP	Print the fluid in-place report.	N/A
2	FIPRESV	Print the reservoir volumes in-place report.	N/A
3	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.	
4			
….		….	

Notes:

1) The keyword is terminated by "/".

Table 12.38: 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 switch off by this keyword, that is the reports are written out every report time step until requested to stop.

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 SCHEDULE SECTION REPORT OPTION (ORIGINAL FORMAT)
--
RPTSCHED
         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.

```
-- ======================================================================
--
-- 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 switch off for the subsequent reporting time steps.

12.3.186 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

12.3.187 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 ignored by OPM Flow and has no effect on the simulation.

12.3.188 SCDETAB – WELL CONNECTION KARST AQUIFER PROPERTIES FOR SCALE DEPOSIT

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

SCDETAB defines well connection karst[134] 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 ignored by OPM Flow and has no effect on the simulation.

[134] *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.*

12.3.189 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 ignored by OPM Flow and has no effect on the simulation.

12.3.190 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 ignored by OPM Flow and has no effect on the simulation, but is documented here for completeness.

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 "/".					

Table 12.39: 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

/
```

12.3.191 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.

12.3.192 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 (B_o) and Gas Oil Ratio ("GOR" or R_s). 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 (B_{ob}) and the initial saturated gas-oil ratio (R_{si}) 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 ignored by OPM Flow and has no effect on the simulation.

12.3.193 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.

12.3.194 SIMULATE - ACTIVATE THE SIMULATION MODE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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
```

12.3.195 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 RUNSPEC 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.

Note that SKIPREST is not necessary for OPM Flow to restart, the simulator will restart gracefully at the chosen step even without it, and the keyword itself is ignored. It is however advisable to include it if compatibility with other simulators is important.

Examples

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
```

12.3.196 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 ignored by OPM Flow and has no effect on the simulation.

12.3.197 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.

12.3.198 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.

12.3.199 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:

$$Q_{month} = DCQ \times SWINGFAC_{month} \quad \textbf{(12.22)}$$

Where:

Q_{month} = 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 ignored by OPM Flow and has no effect on the simulation.

12.3.200 TIGHTEN – TIGHTEN AND RELAX NUMERICAL CONTROLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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 ignored by OPM Flow and has no effect on the simulation.

12.3.201 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 ignored by OPM Flow and has no effect on the simulation.

12.3.202 TIME – ADVANCE SIMULATION BY CUMULATIVE REPORTING TIME

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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 TIME keywords may be entered to advance the simulator to the next report time.

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 12.40: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.

12.3.203 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.			
		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 12.41: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 fist 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.

12.3.204 TUNING - NUMERICAL TUNING CONTROL

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 2019-10 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
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

No.	Name	Description			Default
		Field	Metric	Laboratory	
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
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

No.	Name	Description			Default
		Field	Metric	Laboratory	
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 Newtonian iterations, and should be set to: 1) 1:The calculation is explicit, that is fully decoupled. 2) 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 Newtonian 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 Newtonian 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 Newtonian 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 Newtonian 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

No.	Name	Description			Default
		Field	Metric	Laboratory	
3-7	DDPLIM	DDPLIM a real positive value that stipulates the maximum pressure change at the last Newtonian 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 Newtonian 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 Newtonian 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 12.42: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 2019-10 From The Command Line*) to instruct the simulator to read the first record of the TUNING keyword. Alternatively one could just use:

```
TUNING
/
/
/
```

12.3.205 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 2019-10 From The Command Line* on how to invoke various numerical schemes via the OPM Flow command line interface.

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 Newtonian 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 Newtonian 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 12.43: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.

12.3.206 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 2019-10 From The Command Line* on how to invoke various numerical schemes via the OPM Flow command line interface.

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 Newtonian 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 Newtonian iteration.			
		dimensionless	dimensionless	dimensionless	25

Notes:

1) There is no keyword terminating "/".

Table 12.44:TUNINGH Keyword Description

Example

```
--
--       DEFAULT TUNINGH PARAMETERS
--
TUNINGH
/
```

The above example explicitly sets the default parameters.

12.3.207 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 2019-10 From The Command Line* on how to invoke various numerical schemes via the OPM Flow command line interface.

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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
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

No.	Name	Description			Default
		Field	Metric	Laboratory	
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
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

No.	Name	Description			Default
		Field	Metric	Laboratory	
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 Newtonian iterations, and should be set to: 1) 1:The calculation is explicit, that is fully decoupled. 2) 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 Newtonian 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 Newtonian 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 Newtonian 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 Newtonian 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

No.	Name	Description			Default
		Field	Metric	Laboratory	
3-7	DDPLIM	DDPLIM a real positive value that stipulates the maximum pressure change at the last Newtonian 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 Newtonian 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 Newtonian 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 12.45: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.

12.3.208 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

See section *2.2 Running OPM Flow 2019-10 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
2-5	XXXTTE	XXXTTE is a real positive value that sets the maximum time truncation error.			
		dimensionless	dimensionless	dimensionless	10.0

No.	Name	Description			Default
		Field	Metric	Laboratory	
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 Newtonian iterations, and should be set to: 1) 1:The calculation is explicit, that is fully decoupled. 2) 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 Newtonian 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 Newtonian 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 Newtonian 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 Newtonian 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 Newtonian 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 Newtonian 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 Newtonian 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 12.46: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

12.3.209 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.

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 12.47: UDQ Keyword Description

Currently, the simulator only supports well variable names (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.

Examples

The fist example shows how to define some constant field variables used for calculating facilities corrected condensate and Liquefied Petroleum Gas[135] ("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 set 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
```

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 users 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.

[135] *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.*

```
                              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  /

                            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  /
/
```

12.3.210 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.

OPM Flow does not recognize this keyword and the keyword may cause the program to terminate, but it is documented here for completeness.

12.3.211 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 ignored by OPM Flow and has no effect on the simulation.

12.3.212 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.

12.3.213 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 followed by a vector of numbers separated by a space and the keyword is terminated by a "/".					

Table 12.48: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
/
```

12.3.214 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 *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 12.49. 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*.			1*
		FIELD	METRIC	LAB	
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
1-7	/	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 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 12.49: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.

12.3.215 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, although the Network option is not currently implemented in OPM Flow. 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 12.50. 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 WPAVDEP 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
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 FLOW 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 FLOW 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 FLOW is set to GAS.			None

No.	Name	Description			Default
		Field	Metric	Laboratory	
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 $\frac{q_w}{q_o}$ and should be used if FLOW 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 FLOW 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 FLOW is set to GAS.			None
1-6	VFPTYPE	A defined character string that should be defaulted or set equal to THP.			THP
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. The default value of 1* is " " or undefined that covers the case when the ALQ variable is not entered.			1*
1-8	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*
1-9	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
1-10	/	Record terminated by a "/"			Not Applicable
2-1	FLO-DATA	A real positive monotonically increasing vector that defines the numerical values of the flowing phase declared by the FLOW 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

No.	Name	Description			Default
		Field	Metric	Laboratory	
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.			
		GRAT: Mscf/day IGLR: Mscf/stb TGLR: Mscf/day 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	None
6-2	/	Record terminated by a "/"			Not Applicable

No.	Name	Description			Default
		Field	Metric	Laboratory	
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 BHP values.			None
	BHP-DATA	BHP-DATA is a real vector of BHP 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 data, is then repeated until all combinations of (NTHP, NWFR, NGFR, NALQ) and the associate BHP data has been entered.			
		psia	barsa	atma	None
7-2	/	Each Index (NTHP, NWFR, NGFR, NALQ. BHP-DATA) data set is terminated by a "/"			Not Applicable

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 12.50: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 *4 GLOBAL SECTION KEYWORDS*, as the data can be quite voluminous.

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.

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.

12.3.216 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 12.51: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
/
```

12.3.217 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 ignored by OPM Flow and has no effect on the simulation.

12.3.218 WAKALIN – 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 ignored by OPM Flow and has no effect on the simulation.

12.3.219 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 ignored by OPM Flow and has no effect on the simulation.

12.3.220 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 (γ_{API}), or degrees API (°API), the relationship between relative density (γ_o) of oil and API gravity (γ_{API}) is given by:

$$\gamma_{API} = \frac{141.5}{\gamma_o} - 131.5 \tag{12.23}$$

This keyword is ignored by OPM Flow and has no effect on the simulation.

12.3.221 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 ignored by OPM Flow and has no effect on the simulation.

12.3.222 WBOREVOL – DEFINE EFFECTIVE WELLBORE STORAGE VOLUME

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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 ignored by OPM Flow and has no effect on the simulation.

12.3.223 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 ignored by OPM Flow and has no effect on the simulation.

12.3.224 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 WHISTCNTL keyword in the SCHEDULE section, from the time the WHISTCNTL is invoked, thus avoiding changing the control model 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.			None

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 VPFTAB 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 marching wells can only 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		Not Used			
12		Not Used			

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:WCONHIST Keyword Description

See also the WHISTCNTL that can be used to reset the TARGET phase, the GCONPROD and GCONINJ 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 gas 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.

12.3.225 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

12.3.226 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 production 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 control 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 control 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 to the bottom-hole pressure as defined by item (7). 4) THP: the target rate is set 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 GCONINJ 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 sm^3/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.			
		rtb/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 of 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		Not Used			
11		Not Used			
12		Not Used			
13		Not Used			
14		Not Used			
15		Not Used			

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:WCONINJE Keyword Description

See also the GCONPROD the GCONINJ 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.

12.3.227 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 a water 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 inject rate to zero means that the well is open to flow with a zero injection rate, this may cause numerical issues.			OPEN
4	RATE	A real positive value that defines the observed surface injection rate.			0.0

No.	Name	Description			Default
		Field	Metric	Laboratory	
		Liquid stb/d Gas Mscf/d	Liquid sm^3/day Gas sm^3/day	Liquid scc/hour Gas scc/hour	
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 marching wells can only 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		Not used and should be defaulted with 1*.			
9		Not used and should be defaulted with 1*.			
10		Not used and should be defaulted with 1*.			
11		Not used and should be defaulted with 1*.			
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
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 12.54: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   RESV   BHP   THP   VFP    NOT   CNTL
-- NAME  TYPE   SHUT   RATE   RATE   PRSES PRES  TABLE  USED  MODE
WCONINJH
GI01     GAS    OPEN  15.5E3  1*     1*    5462   12    4*    1*  /
/
DATES
01 FEB 2000 /
/
--
--       WELL HISTORICAL INJECTION CONTROLS
--
-- WELL  FLUID  OPEN/  SURF   RESV   BHP   THP   VFP    NOT   CNTL
-- NAME  TYPE   SHUT   RATE   RATE   PRSES PRES  TABLE  USED  MODE
WCONINJH
GI01     GAS    OPEN  15.9E3  1*     1*    5468   1*    4*    1*  /
/
DATES
01 MAR 2000 /
/
--
--
--       WELL HISTORICAL INJECTION CONTROLS
--
-- WELL  FLUID  OPEN/  SURF   RESV   BHP   THP   VFP    NOT   CNTL
-- NAME  TYPE   SHUT   RATE   RATE   PRSES PRES  TABLE  USED  MODE
WCONINJH
GI01     GAS    OPEN  17.2E3  1*     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 users 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.

12.3.228 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 ignored by OPM Flow and has no effect on the simulation.

12.3.229 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 (3) 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.			None
4	ORAT	A real positive value that defines the maximum surface oil production rate target or constraint.			
		stb/d	sm^3/day	scc/hour	None
5	WRAT	A real positive value that defines the maximum surface water production rate target or constraint.			
		stb/d	sm^3/day	scc/hour	None
6	GRAT	A real positive value that defines the maximum surface gas production rate target or constraint			
		Mscf/d	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/d	sm^3/day	scc/hour	None
8	RESV	A real positive value that defines the maximum reservoir volume production rate target or constraint.			
		rtb/d	rm^3/day	rcc/hour	None

No.	Name	Description			Default
		Field	Metric	Laboratory	
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 VPFTAB 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.			0.0
13		Not Used			
14		Not Used			
15		Not Used			
16		Not Used			
17		Not Used			
18		Not Used			
19		Not Used			
20		Not Used			

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 12.55:WCONPROD Keyword Description

See also the GCONPROD and GCONINJ 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 three wells are not on group control, for example, 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.

12.3.230 WCUTBACK – DEFINE WELL CUTBACK LIMITS AND PARAMETERS

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 ignored by OPM Flow and has no effect on the simulation.

12.3.231 WCUTBACT – DEFINE WELL TRACER CUTBACK LIMITS AND PARAMETERS

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 ignored by OPM Flow and has no effect on the simulation.

12.3.232 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 ignored by OPM Flow and has no effect on the simulation.

12.3.233 WDFAC – DEFINE GAS FLOW DEPENDENT SKIN FACTOR

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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[136,137 and 138].

See also the WDFACCOR keyword in the SCHEDULE section that users Dake's[139] correlation to calculate the D-factor.

This keyword is ignored by OPM Flow and has no effect on the simulation.

[136] *Geertsma, J., 1974. Estimating the Coefficient of Inertial Resistance in Fluid Flow Through Porous Media. Soc.Pet.Eng.J., October: 445-450.*

[137] *Gewers, C.W.W. and Nichol, L.R., 1969. Gas Turbulence Factor in a Microvugular Carbonate. J.Can.Pet.Tech., April.*

[138] *Wong, S.W., 1970. Effects of Liquid Saturation on Turbulence Factors for Gas Liquid Systems. J.Can.Pet.Tech., October*

[139] *Dake, L.P. Fundamentals of Reservoir Engineering, Amsterdam, The Netherlands, Elsevier Science BV (1978) Chapter 8.6, pages 252-257.*

12.3.234 WDFACCOR – GAS FLOW DEPENDENT SKIN FACTOR (CORRELATION)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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 [140,141,142 and 143]. This keyword users Dake's correlation to calculate the D-factor.

This keyword is ignored by OPM Flow and has no effect on the simulation.

[140] *Dake, L.P. Fundamentals of Reservoir Engineering, Amsterdam, The Netherlands, Elsevier Science BV (1978) Chapter 8.6, pages 252-257.*

[141] *Geertsma, J., 1974. Estimating the Coefficient of Inertial Resistance in Fluid Flow Through Porous Media. Soc.Pet.Eng.J., October: 445-450.*

[142] *Gewers, C.W.W. and Nichol, L.R., 1969. Gas Turbulence Factor in a Microvugular Carbonate. J.Can.Pet.Tech., April.*

[143] *Wong, S.W., 1970. Effects of Liquid Saturation on Turbulence Factors for Gas Liquid Systems. J.Can.Pet.Tech., October*

12.3.235 WDRILPRI – ADD WELLS TO THE DRILLING PRIORITY DRILLING QUEUE

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 ignored by OPM Flow and has no effect on the simulation.

12.3.236 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 ignored by OPM Flow and has no effect on the simulation.

12.3.237 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 ignored by OPM Flow and has no effect on the simulation.

12.3.238 WECON – WELL ECONOMIC CRITERIA FOR PRODUCTION WELLS

Description

The WECON keyword defines 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.			
		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 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 is 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		Not used			
10		Not used			
11		Not used			
12		Not used			
13		Not used			
14		Not used			
15		Not used			
16		Not used			

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 12.56: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.

12.3.239 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 GCONINJ keyword in the SCHEDULE section and the controls specified by this keyword.

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 sm^3/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 12.57:WECONINJ Keyword Description

See also the WELSPECS keyword to define a wells shut-in or stop options and GCONINJ 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/day. The AUTO parameter on the WELSPECS keyword will determine if the wells will be shut-in or stopped.

12.3.240 WECONT – WELL ECONOMIC TRACER CRITERIA FOR PRODUCTION WELLS

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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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
1-5	/	Record one terminated by a "/"			Not Applicable

No.	Name	Description			Default
		Field	Metric	Laboratory	
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		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		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		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		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		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		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 12.58: 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

12.3.241 WEFAC – DEFINE WELL EFFICIENCY

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

Defines a well's efficiency or up-time.

This keyword is ignored by OPM Flow and has no effect on the simulation.

See the GEFAC keyword in the SCHEDULE section that is implemented in OPM Flow and can be used to set a group's efficiency.

12.3.242 WELCNTL – MODIFY WELL CONTROL AND TARGETS

Description

The WELCNTL keyword modifies a wells' 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, WCONDINJE, or WCONDINJH 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 WELCONINJE) 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	sm^3/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 12.59: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.

12.3.243 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), it is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

12.3.244 WELDRAW – DEFINE MAXIMUM DRAW DOWN FOR PRODUCTION WELLS

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 ignored by OPM Flow and has no effect on the simulation.

12.3.245 WELEVNT – DEFINE WELL WPWEM SUMMARY MNEMONIC OUTPUT VALUE

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 ignored by OPM Flow and has no effect on the simulation.

12.3.246 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.

OPM Flow does not recognize this keyword and the keyword may cause the program to terminate, but it is documented here for completeness.

12.3.247 WELOPEN – DEFINE WELL AND WELL CONNECTIONS FLOWING STATUS

Description

The WELOPEN keyword defines the status of wells and well connection 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 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	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 connection location in the K-direction. If connections have been lumped into completions via the COMPLUMP keyword, then K1 refers to the completion number instead of the connection (layer) value.			1*
7	K2	An integer less than or equal to NZ that defines the LOWER connection location in the K-direction. If connections have been lumped into completions via the COMPLUMP keyword, then K2 refers to the completion number instead of the connection (layer) value.			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 12.60: 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 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 with the I, J, K variables the specified location and a completion number in the 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 WCONDINJE 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 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   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' /
OP03      1*  1*  35  90   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    0    35    90       /
OP02     OPEN                                  /
OP02     OPEN     0   0    0     2     5       /
OP03     OPEN                                  /
OP03     OPEN     0   0    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 35 to 90 are opened for flow, connections two to five for well OP02 and all the connections for well OP03.

The next example shows the use of the COMPLUMP keyword to group the well connections into well completions for wells OP01 and OP03, and then use the WELOPEN keyword to open the well and the well connections.

```
--
--       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
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     5                 /
OP03     OPEN                                            /
OP03     OPEN     0   0    0     3     3                 /
/
```

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), connections two to five for well OP02 and completion number three (all the connections) for well OP03.

Note the completion number for well OP03 was named completion number three, but it could have been named number one as well. The reason why it was named number three instead of one 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     5                 /
/
```

In this case well OP01 and OP03 are opened via completion number three, and well OP02 is opened on well connection (or layer) number three – which we do not want. Hence, all the connection for OP02 are shut, and then connections two to five are opened instead for well OP02.

12.3.248 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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: 4) OPEN: the connections are open to flow. 5) SHUT: the connections are closed to flow (shut-in). 6) 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 connection location in the K-direction. If connections have been lumped into completions via the COMPLUMP keyword, then K1 refers to the completion number instead of the connection (layer) value.			1*

No.	Name	Description			Default
		Field	Metric	Laboratory	
8	K2	An integer less than or equal to NZ that defines the LOWER connection location in the K-direction. If connections have been lumped into completions via the COMPLUMP keyword, then K2 refers to the completion number instead of the connection (layer) value.			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 12.61: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 with the I, J, K variables the specified location and a completion number in the range specified by K1 and K2.

See also the COMPDATL keyword to define a well's connections, the COMPLMPL keyword to group LGR well connections into well completions, the WCONPROD and WCONDINJE 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).

12.3.249 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 and values enter on this keyword for a given well will override any previously calculated values and values previously entered using this keyword.

This keyword is ignored by OPM Flow and has no effect on the simulation.

12.3.250 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 ignored by OPM Flow and has no effect on the simulation.

12.3.251 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.			
		ft^3	m^3	cm^3	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 si 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	
1-6	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
1-7	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
1-8	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
1-9	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
1-10	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
1-11	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
2-8	EIPSILON	A real positive value that defines the tubing absolute roughness of the segment for the well.			
		feet	m	cm	None

No.	Name	Description			Default
		Field	Metric	Laboratory	
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.			
		ft^3	m^3	cm^3	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
2-16	/	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 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 12.62: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 12.2 depicts the resulting well configuration for both well segments.

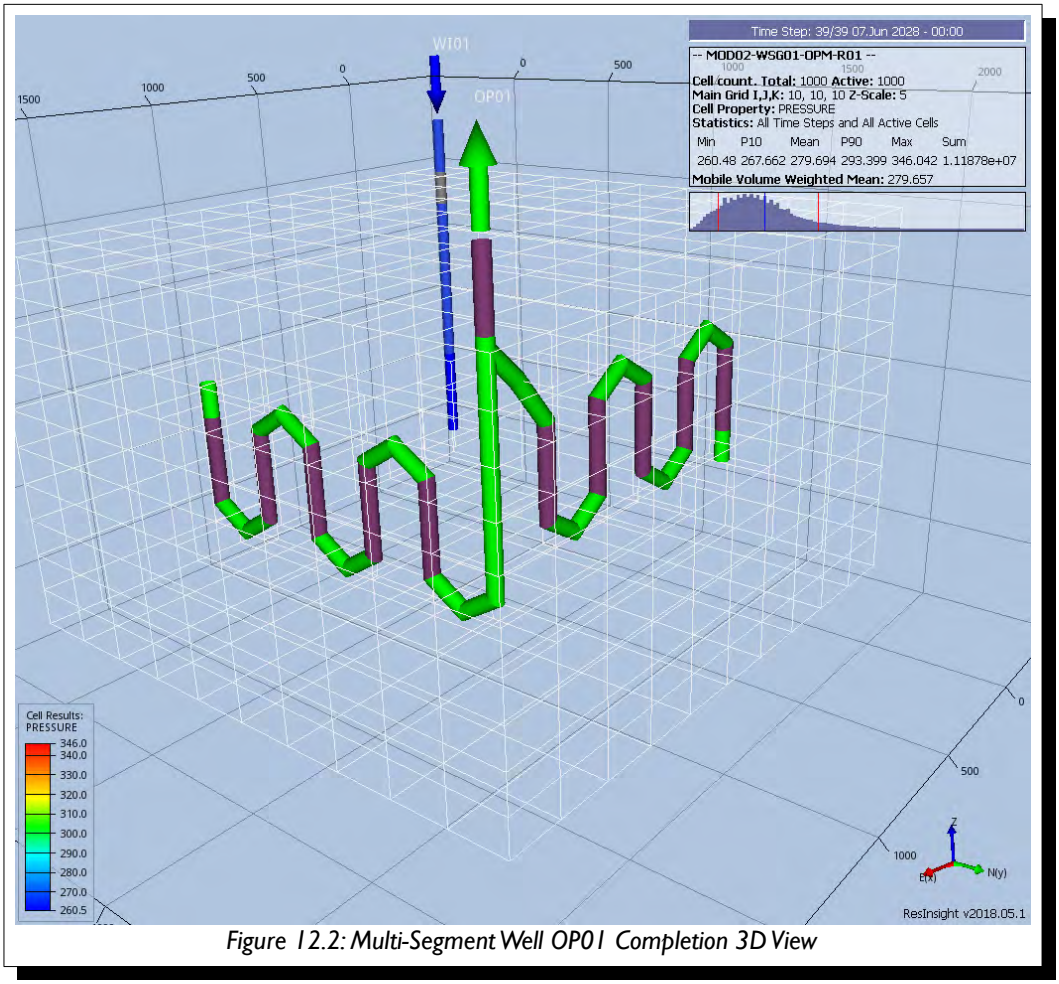


Figure 12.2: Multi-Segment Well OP01 Completion 3D View

12.3.252 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 ignored by OPM Flow and has no effect on the simulation.

12.3.253 WELSPECL – DEFINE WELL SPECIFICATIONS FOR LOCAL GRID REFINEMENTS

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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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.			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. 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
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
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 COMPDAT keyword
		feet	m	cm	

No.	Name	Description			Default
		Field	Metric	Laboratory	
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
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[144] 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.[145] 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

[144] *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.*

[145] *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	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
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

No.	Name	Description			Default
		Field	Metric	Laboratory	
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.			

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 12.63:WELSPECL Keyword Description

See also the COMPDATL keyword to define a well's connections in a LGR, the WCONPROD and WCONDINJE 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 users the dry gas pseudo inflow equation, GP01 is a gas condensate well that users the generalized gas pseudo pressure inflow equation, and finally, OP01 is an oil well that users 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.

12.3.254 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 name of well, 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 connection 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. 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
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.			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..			None
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 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

No.	Name	Description			Default
		Field	Metric	Laboratory	
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
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[146] 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.[147] 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
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

[146] *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.*

[147] *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 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. 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
14		Not used.			
15		Not used.			
16		Not used.			
17		Not used.			

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 12.64:WELSPECS Keyword Description

See also the COMPDAT keyword to define a well's connections, the WCONPROD and WCONDINJE 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 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 users the dry gas pseudo inflow equation, GP01 is a gas condensate well that users the generalized gas pseudo pressure inflow equation, and finally, OP01 is an oil well that users 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.

12.3.255 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 keywords. 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) 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

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	sm^3/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 12.65:WELTARG Keyword Description

If a well is currently a history matching well, then WELTARG 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 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.

12.3.256 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 12.66: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.

12.3.257 WFRICSEG – CONVERT FRICTION WELL TO MULTI-SEGMENT WELL

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 ignored by OPM Flow and has no effect on the simulation.

12.3.258 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 ignored by OPM Flow and has no effect on the simulation.

12.3.259 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 ignored by OPM Flow and has no effect on the simulation.

12.3.260 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 ignored by OPM Flow and has no effect on the simulation.

12.3.261 WGASPROD – DEFINE SALE GAS WELL PRODUCTION TARGETS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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 ignored by OPM Flow and has no effect on the simulation.

12.3.262 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 ignored by OPM Flow and has no effect on the simulation.

12.3.263 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) LRAT: 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 injection phase. This should only be used if the well has been declared an injection via the WCONINJE keyword in the SCHEDULE section.			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 12.67: WGRUPCON Keyword Description

See also the GCONPROD the GCONINJ 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 injector wells are controlled based on their reservoir potential volumes and the producers on their potential gas rates. In comparison, all the oil wells controlled by their oil rates.

12.3.264 WHEDREFD – DEFINE WELL HYDRAULIC HEAD REFERENCE DEPTH

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 12.68: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.

12.3.265 WHISTCTL - DEFINE WELL HISTORICAL TARGET PHASE

Description

The WHISCTL 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 WHISCTL 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 12.69: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 well 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.

12.3.266 WHTEMP – DEFINE WELL TUBING HEAD TEMPERATURE PARAMETERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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 is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 data is being defined.			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 VFPPROD temperature table.			None
		dimensionless	dimensionless	dimensionless	
3	TEMP	A real positive value greater than zero that defines 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 12.70: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 users a constant 150° tubing head temperature.

12.3.267 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.

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

12.3.268 WINJTEMP – DEFINE INJECTION FLUID THERMAL PROPERTIES

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 users 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 12.71: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.

12.3.269 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.

12.3.270 WLIFTOPT – DEFINE WELL GAS LIFT OPTIMIZATION PARAMETERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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.

OPM Flow does not have this feature and hence this keyword is ignored by OPM Flow and the WLIFTOPT keyword has no effect on the simulation.

12.3.271 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 ignored by OPM Flow and has no effect on the simulation.

12.3.272 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 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	ACTION	A defined character string that determines how the WELLNAMES should be handled with respect to the named well list (WLIST). ACTION should be set to one of the following:: 1) ADD: Add the WELLNAMES to an existing WLIST. 2) DEL; Delete WELLNAMES from an existing WLIST. 3) MOV: WELLNAMES from another existing named well list and ADD them to WLIST. 4) NEW: Define a new named well list and add the WELLNAMES 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 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.	
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 12.72: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.

Examples

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.

12.3.273 WLISTARG – MODIFY WELL LIST TARGET AND CONSTRAINT VALUES (STATIC)

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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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
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

No.	Name	Description			Default
		Field	Metric	Laboratory	
3	VALUE Liquid Gas	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.			
	Res Vol	stb/d	sm^3/day	scc/hour	
	Pressure	Mscf/d	sm^3/day	scc/hour	
	VFP	rb/d	rm^3/day	rcc/hour	
	LIFT	psia	barsa	atma	
		dimensionless	dimensionless	dimensionless	
		same as	same as	same as	
		VFPPROD or VFPINJ	VFPPROD or VFPINJ	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 12.73: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.

12.3.274 WLISTNAM – DEFINE WELL LISTS (WLISTARG)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 12.74: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 *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.

Examples

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.

12.3.275 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 ignored by OPM Flow and has no effect on the simulation.

12.3.276 WNETDP – DEFINE WELL THP TO NETWORK PRESSURE DROP

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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 ignored by OPM Flow and has no effect on the simulation.

12.3.277 WORKLIM – DEFINE WELL WORKOVER TIME

Description

WORKLIM sets the numbers of days taken to complete a workover.

This keyword is ignored by OPM Flow and has no effect on the simulation.

12.3.278 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 ignored by OPM Flow and has no effect on the simulation.

12.3.279 WPAVE – WELL BLOCK AVERAGE PRESSURE CALCULATION PARAMETERS FOR ALL WELLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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 file in order to compared with field observed data.

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 to calculated the connection factor weighted average pressures. If WPAVE1 is greater than or equal to zero and less than or equal to one, then the average pressures are calculate based on the connection factors. 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. If WPAVE1 is equal to zero, then average pressure is calculate based on the pore volumes of the 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 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
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 RUNSPEC 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.			OPEN
Notes: 1) The keyword should be terminated by a "/".					

Table 12.75:WPAVE Keyword Description

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.

Example

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.

12.3.280 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 12.76: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.

12.3.281 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 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 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 connection location in the K-direction. If connections have been lumped into completions via the COMPLUMP keyword, then K1 refers to the completion number instead of the connection (layer) value.			1*
7	K2	An integer less than or equal to NZ that defines the LOWER connection location in the K-direction. If connections have been lumped into completions via the COMPLUMP keyword, then K2 refers to the completion number instead of the connection (layer) value.			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 12.77:WPIMULT Keyword Description

If variables I, J K, K1 and K2 are all defaulted with zero 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 variable is applied at the time the WPIMULT keyword is entered and is cumulative if there are intervening time steps between consecutive WPIMULT keywords.

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.

Examples

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, 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.

12.3.282 WPIMULTL – DEFINE WELL CONNECTION MULTIPLIERS (LGR)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 connection location in the K-direction. If connections have been lumped into completions via the COMPLUMP keyword, then K1 refers to the completion number instead of the connection (layer) value.			1*
7	K2	An integer less than or equal to NZ that defines the LOWER connection location in the K-direction. If connections have been lumped into completions via the COMPLUMP keyword, then K2 refers to the completion number instead of the connection (layer) value.			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 12.78:WPIMULT Keyword Description

If variables I, J K, K1 and K2 are all defaulted with zero 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 variable is applied at the time the WPIMULTL keyword is entered and is cumulative if there are intervening time steps between consecutive WPIMULTL keywords.

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.

Examples

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.

12.3.283 WPITAB - ASSIGN WELL PRODUCTIVITY INDEX VERSUS WATER CUT TABLES

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.

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	NTABLE	An integer positive 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 12.79: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.

This keyword is ignored by OPM Flow and has no effect on the simulation.

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.

12.3.284 WPLUG – DEFINE WELL PLUG BACK LENGTH

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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 ignored by OPM Flow and has no effect on the simulation.

12.3.285 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 variable is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.			
		lb/stb	kg/sm^3	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 variable is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.			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 12.80: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.

12.3.286 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

12.3.287 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

12.3.288 WRFT – ACTIVATE WELL RFT REPORTING TO THE RFT FILE

Description

This keyword activates reporting of a well's depth pressure profile 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 12.81: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.

12.3.289 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
4		Not Used.			

Notes:

1) The keyword is followed by any number of records terminated by a "/" and the keyword should be terminated by a "/".

Table 12.82: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.

12.3.290 WSALT - DEFINE WATER INJECTION WELL SALT CONCENTRATIONS

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. Currently the Brine option is not implemented in OPM Flow.

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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	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; or, 2) 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. Note if SALTCON is defaulted (1*) then the well's salt concentration will be use the well's group salt concentration.			
		lb/stb	kg/sm^3	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 12.83: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.

Example

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, 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.

12.3.291 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 ignored by OPM Flow and has no effect on the simulation.

12.3.292 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 ignored by OPM Flow and has no effect on the simulation.

12.3.293 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 ignored by OPM Flow and has no effect on the simulation.

12.3.294 WSEGAICD – DEFINE MULTI-SEGMENT WELL AUTONOMOUS ICD CONNECTIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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. 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 ignored by OPM Flow and has no effect on the simulation.

12.3.295 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[148] and [149] 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 ignored by OPM Flow and has no effect on the simulation.

[148] *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).*

[149] *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).*

12.3.296 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[150] and [151] 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 ignored by OPM Flow and has no effect on the simulation.

[150] *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).*

[151] *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).*

12.3.297 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[152] and [153] 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 ignored by OPM Flow and has no effect on the simulation.

[152] *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).*

[153] *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).*

12.3.298 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 ignored by OPM Flow and has no effect on the simulation.

12.3.299 WSEGFLIM - DEFINE MULTI-SEGMENT WELL ARTIFICIAL CHOKE CONNECTIONS

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 ignored by OPM Flow and has no effect on the simulation.

12.3.300 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 ignored by OPM Flow and has no effect on the simulation.

12.3.301 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 ignored by OPM Flow and has no effect on the simulation.

12.3.302 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 users 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 2019-10 From The Command Line* on how to set the numerical control parameters for OPM Flow.

12.3.303 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 ignored by OPM Flow and has no effect on the simulation.

12.3.304 WSEGLINK - DEFINE MULTI-SEGMENT WELL LOOPED FLOW PATHS

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 ignored by OPM Flow and has no effect on the simulation.

12.3.305 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:

$$Frictional\ Loss\ Multipler\ =\ min\left(x_1 + x_2(WOR)^{x_3} + x_4\left(\frac{GOR}{GOR_{min}}\right)^{x^5},\ 1.0\right) \quad \textbf{(12.24)}$$

Where the constants x_1 to x_5 are defined by the values on this keyword.

This keyword is ignored by OPM Flow and has no effect on the simulation.

12.3.306 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 ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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	EIPSILON	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
		ft^3	m^3	cm^3	
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	

No.	Name	Description			Default
		Field	Metric	Laboratory	
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 12.84: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.

12.3.307 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 ignored by OPM Flow and has no effect on the simulation.

12.3.308 WSEGSEP – DEFINE A MULTI-SEGMENT WELL DOWN-HOLE SEPARATOR

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 ignored by OPM Flow and has no effect on the simulation.

12.3.309 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 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	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$(rft^3/day)^2$	barsa$/(rm^3/day)^2$	atma$/(rcc/day)^2$	None

No.	Name	Description			Default
		Field	Metric	Laboratory	
5	ICDLEN	A real value 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 to 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	0.45
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 to 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	None
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 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 12.85: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.

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 users one ICD for segments seven to ten, branch number three again users only one ICD for segments 12 to 15 and similarly branch number three users 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.

12.3.310 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 ignored by OPM Flow and has no effect on the simulation.

12.3.311 WSEGTABL – ASSIGN MULTI-SEGMENT WELL VLP TABLES TO SEGMENTS

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 ignored by OPM Flow and has no effect on the simulation.

12.3.312 WSEGVALV – DEFINE MULTI-SEGMENT WELL SUB-CRITICAL VALVE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The WSEGVALV keyword stipulates a multi-segment well sub-critical valve segment that adds an additional pressure drop due to a flow restriction. The data on the keyword contains the pertinent data required to model this type of flow device.

This keyword is ignored by OPM Flow and has no effect on the simulation.

12.3.313 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 12.86: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.

12.3.314 WSURFACT - DEFINE WATER INJECTION WELL SURFACTANT CONCENTRATION

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.

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	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 12.87: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.

```
--
--       DEFINE WATER INJECTION WELL SURFACTANT CONCENTRATION
--
-- WELL  SURFACT
-- NAME  SURCON
--       --------
WSURFACT
WI01     0.2000                                          /
WI02     0.2000                                          /
WI03     0.2000                                          /
/
```

Here the surfactant concentration has been set to 0.200 for all three wells.

12.3.315 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 ignored by OPM Flow and has no effect on the simulation.

12.3.316 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 users 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 12.88: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 °F and well WI02's injection water temperature is 37 °F.

12.3.317 WTEMPQ – OUTPUT WELL NAMES AND WELL LISTS TO THE PRINT FILE

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 ignored by OPM Flow and has no effect on the simulation.

12.3.318 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) G: meaning the well was closed due to a group 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. 4) D: not used by OPM Flow. 5) C: not used by OPM Flow. The default value is an empty string " " that switches of testing. ***Note that only the E option is currently supported in OPM Flow.***			" "
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 tiime step size, then the well is brought on gradually, if it is less then the well is opened faster. The default value of 0.0 means the well is opened immediately.			
		days	days	hours	0.0

No.	Name	Description			Default
		Field	Metric	Laboratory	
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 12.89: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 E option is currently supported in OPM Flow.***

12.3.319 WTHPMAX – DEFINE A WELL'S MAXIMUM FLOWING THP FOR SHUT-IN

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 ignored by OPM Flow and has no effect on the simulation.

12.3.320 WTMULT – MULTIPLE A WELL TARGET OR CONSTRAINT BY A CONSTANT

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

This keyword, WTMULT multiplies a define 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.

This keyword is ignored by OPM Flow and has no effect on the simulation.

12.3.321 WTRACER – DEFINE AN INJECTION WELL'S TRACER CONCENTRATION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The WELTRACER 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 TRACER keyword in the RUNSPEC section.

OPM Flow has not implemented tracer modeling and therefore this keyword is ignored by OPM Flow and has no effect on the simulation.

12.3.322 WVFPDP – MODIFY WELL BHP OBTAINED FROM VFP TABLES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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.

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

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 (BHP_{VFP}). 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 BHP_{VFP} 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 12.90: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%.

12.3.323 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 and the WCONINJE keyword for assigning the VFPINJ tables for injection 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 specially requestd. All the aforementioned keywords are in the SCHEDULE section.

This keyword is ignored by OPM Flow and has no effect on the simulation.

12.3.324 WWPAVE – WELL BLOCK AVERAGE PRESSURE CALCULATION PARAMETERS FOR INDIVIDUAL WELLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Description

The WWPAVE keyword defines the method and parameters for calculating a well's block average pressures for individual wells. 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.

This keyword is ignored by OPM Flow and has no effect on the simulation.

12.3.325 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 TUNNING keyword in the SCHEDULE section, combined with posterior knowledge gained from previous time steps.

This keyword is ignored by OPM Flow and has no effect on the simulation.

See section *2.2 Running OPM Flow 2019-10 From The Command Line* on how to control time stepping for OPM Flow.

12.3.326 ZIPPY2 – ACTIVATE AUTOMATIC TIME STEP CONTROL

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 ignored by OPM Flow and has no effect on the simulation.

See section *2.2 Running OPM Flow 2019-10 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	
ALPOLDS – 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	
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	

A

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter A	Status
AQANCONL – Define Analytical Connections to a LGR Grid	
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
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	
BPDIMS – 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	
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	
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	
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	
COMPSEGL – Define Well Connections for Multi-Segment Wells in a LGR	
COMPSEGS – Define Well Connections for Multi-Segment Wells	
COMPVE – Re-Define Well Connection Depths	
COMPVEL – Re-Define Well LGR Connection Depths	

C

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter C	Status
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	
DEADOIL – Activate the Dead Oil Phase (No Free or Dissolved Gas)	
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	
DIFFCOAL – Define Coal Bed Methane Gas Diffusion Data	
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	
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	

D

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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 Dimension 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	
DISPDIMS – Define the Maximum Number of Dispersion Tables	
DISPERSE– Define Dispersion Tables	
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	
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	
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	

D

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

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	
ENPTVDX	
ENPTVDX-	
ENPTVDY	
ENPTVDY-	

E

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter E	Status
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	
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	
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	
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	

F

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter F	Status
FOAMOPTS - Define Foam Model Options	
FOAMROCK - Define Foam Rock Properties	
FORMFEED – Defined the Print File Form-Feed Character	
FWSET - 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	
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	
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	
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	
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	

G

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter G	Status
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	
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	

G

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter G	Status
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	
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
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	
HMKRORW	
HMKRW	
HMKRWR	
HMPCG	
HMPCG	

H

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter H	Status
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	
HMSIGMA	
HMSOGCR	
HMSOWCR	
HMSWCR	
HMSWL	
HMSWLPC	
HMTRANX	
HMTRANY	
HMTRANZ	

H

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter H	Status
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	
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	

H

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter H	Status
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)	
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 Inhibition 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	
IKRORGY-	
IKRORGZ	
IKRORGZ-	
IKRORW – End-Point Scaling of Grid Cell Kro(Swcr) (Imbibition)	

I

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter I	Status
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	
IMBNUMY	
IMBNUMZ	
IMBNUMX-	
IMBNUMY-	
IMBNUMZ-	
IMBNUMMF – Define the Imbibition Saturation Table Region Numbers (Matrix-Fracture)	
IMKRVD – Imbibition Relative Permeability End-Points versus Depth Functions	
IMKRVDX	

I

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter I	Status
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	
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-	

I

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter I	Status
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)	
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.)	

I

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter I	Status
ISOWCRX	
ISOWCRX-	
ISOWCRY	
ISOWCRY-	
ISOWCRZ	
ISOWCRZ-	
ISWCR – End-Point Scaling of Grid Cell Critical Water Saturation (Imbibition)	
ISWCRX	
ISWCRX-	
ISWCRY	
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	
KRNUMMF – Define the Saturation Table Region Numbers (Matrix-Fracture)	
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-	

K

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter K	Status
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-	
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	
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)	

L

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter L	Status
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	
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	
MINNNCT – Set a Minimum Non-Neighbor Connection Transmissibility	
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	
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	

M

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter M	Status
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	
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	
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	
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	
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	

N

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter N	Status
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 Number of Newtonian 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	
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	
PEKTAB0	
PEKTAB1	
PEKTAB2	
PEKTAB3	
PEKTAB4	

P

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter P	Status
PEKTAB5	
PEKTAB6	
PEKTAB7	
PENUM – Define the Petro-Elastic Region Numbers	
PERMAVE – Define Average Transmissibility Coefficients	
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	
PINCHOUT - Define Pinch-Out Layers Option (Fixed)	
PINCHREG - Define Pinch-Out Region Options	
PINCHXY – Define Pinch-Out Areal Options	
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	

P

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter P	Status
PLYKRRF – Define Polymer Rock Permeability Reduction by Cell	
PLYMAX - Define Polymer-Salt Viscosity Mixing Concentrations	
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	
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	
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	
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	

P

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter P	Status
PVDS - Solvent PVT Properties for the Solvent Model	
PVTG - Gas PVT Properties for Wet Gas	
PVTNUM – Define the PVT Regions	
PVTO - Oil PVT Properties for Live Oil	
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)	

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	
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	

R

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter R	Status
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)	
ROCKWNODE – 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	
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	

R

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter R	Status
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	

S

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter S	Status
SALT – Define the Initial Equilibration Salt Concentration for All Grid Blocks	
SALNODE – Salt Concentration Based PVTNUM Array	
SALTREST – Define the Restart Salt Concentration for All Grid Blocks	
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	
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	
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-	

S

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter S	Status
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)	
SGU – End-Point Scaling Grid Cell Gas Saturation	
SGUX	
SGUX-	
SGUY	
SGUY-	
SGUZ	
SGUZ-	
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 Date	
SKIP100 – Activate Skipping of "Black-Oil" Keywords and Input Date	
SKIP300 – Activate Skipping of "Compositional" Keywords and Input Date	
SKIPREST – Activate Skipping of Restart Schedule Data	

S

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter S	Status
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)	
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	

S

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter S	Status
SOLVNUM – Define PEBI Grid Correspondence to Solver Order	
SOMGAS – STONE1 Model Minimum Oil Saturation versus Gas Saturation	
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-	
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	
SPOLY – Define the Initial Equilibration Polymer Concentration for All Grid Blocks	
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	
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	

S

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter S	Status
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	
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	
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)	

S

Alphabetic Listing Of Keywords Starting With The Letter S	Status
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-	
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	
THCSOLID – Define Solid Phase Thermal Conductivity for All Cells	
THCWATER – Define Water Phase Thermal Conductivity for All Cells	
THERMAL– Activate the Thermal Modeling Option	
THERMEXI – Define Component Thermal Expansion Coefficients	
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	
TPAMEPSS - Volumetric Strain versus Coal Solvent Concentration Tables	
TRACER – Define Passive Tracer Variables	

T

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter T	Status
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	
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 Vaporize Oil in Wet Gas Phase in the Model	
VAPPARS – Oil Vaporization Parameters	
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	
WAITBAL – Wait On Network Balance Before Allowing Further Actions	
WAKALIN – 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	
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	
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	

W

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter W	Status
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	
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	
WFOAM - Define Well Foam Injection Concentrations	
WFRICSEG – Convert Friction Well to Multi-Segment Well	
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	
WINJMULT – Define Well Pressure Dependent Injectivity Multipliers	
WINJTEMP – Define Injection Fluid Thermal Properties	

W

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter W	Status
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)	
WNETCTRL – Define Well Control for Network Control Option	
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	
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	

W

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE

Alphabetic Listing Of Keywords Starting With The Letter W	Status
WSEGDIMS – Define Multi-Segment Well Dimensions	
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	
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	
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 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 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	

APPENDIX B: OPM FLOW RELEASE HISTORY

B.1 RELEASE 2019-10

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.1.

No.	Keyword	Comment
1	ACTION	Define Action Conditions and Command Processing (Field). Recognized by the input deck parser but not supported by OPM Flow.
2	ACTIONG	Define Action Conditions and Command Processing (Groups). Recognized by the input deck parser but not supported by OPM Flow.
3	ACTIONR	Define Action Conditions and Command Processing (Regions). Recognized by the input deck parser but not supported by OPM Flow.
4	ACTIONS	Define Action Conditions and Command Processing (Well Segments). OPM Flow does not recognize this keyword and the keyword may cause the program to terminate.
5	ACTIONW	Define Action Conditions and Command Processing (Wells). Recognized by the input deck parser but not supported by OPM Flow.
6	ACTIONX	Define Action Conditions and Command Processing. Recognized by the input deck parser and supported by OPM Flow.
7	ACTPARAM	Define Action Facility Target and Tolerance Parameters. Recognized by the input deck parser but not supported by OPM Flow.
8	ENDACTIO	End the Definition of ACTION Commands. Recognized by the input deck parser and supported by OPM Flow.
9	UDADIMS	Define the Dimensions of the User Defined Arguments. Recognized by the input deck parser and supported by OPM Flow.
10	UDQ	Declare User Define Quantities ("UDQ"). Recognized by the input deck parser and supported by OPM Flow.
11	UDQDIMS	Define the Dimensions of the User Defined UDQ Feature. Recognized by the input deck parser and supported by OPM Flow.
12	UDQPARAM	Define Parameters for the User Defined Quantity Feature. Recognized by the input deck parser and supported by OPM Flow.
13	UDT	Declare User Define Tables ("UDT"). Recognized by the input deck parser but not supported by OPM Flow.
14	UDTDIMS	Define the Dimensions of the User Defined Tables. Recognized by the input deck parser but not supported by OPM Flow.
15	FILLEPS	Activate Saturation End Point Export to INIT File. Recognized by the input deck parser and supported by OPM Flow.
16	FOAM	Activate the Foam Phase and Model. Recognized by the input deck parser and supported by OPM Flow.
17	FOAMADS	Define Foam Rock Adsorption Tables. Recognized by the input deck parser and supported by OPM Flow.
18	FOAMDCYO	Define Foam Decay versus Oil Saturation Tables. Recognized by the input deck parser but not supported by OPM Flow.
19	FOAMDCYW	Define Foam Decay versus Water Saturation Tables. Recognized by the input deck parser but not supported by OPM Flow.
20	FOAMFCN	Define Foam Gas Mobility Reduction versus Capillary Number. Recognized by the input deck parser but not supported by OPM Flow.

No.	Keyword	Comment
21	FOAMFRM	Define Foam Gas Mobility Reduction versus Reference Mobility. Recognized by the input deck parser but not supported by OPM Flow.
22	FOAMFSC	Define Foam Gas Mobility versus Surfactant Concentration Functions. Recognized by the input deck parser and supported by OPM Flow.
23	FOAMFSO	Define Foam Gas Mobility Reduction versus Oil Saturation. Recognized by the input deck parser but not supported by OPM Flow.
24	FOAMFST	Define Foam Gas-Water Surface Tension versus Surfactant Concentration. Recognized by the input deck parser but not supported by OPM Flow.
25	FOAMFST	Define Foam Gas-Water Surface Tension versus Surfactant Concentration. Recognized by the input deck parser but not supported by OPM Flow.
26	FOAMMOB	Define Foam Gas Mobility versus Foam Concentration Tables. Recognized by the input deck parser and supported by OPM Flow.
27	FOAMMOBP	Define Foam Mobility Reduction versus Oil Pressure. Recognized by the input deck parser but not supported by OPM Flow.
28	FOAMMOBS	Define Foam Mobility Reduction versus Shear. Recognized by the input deck parser but not supported by OPM Flow.
29	FOAMOPTS	Define Foam Model Options. Recognized by the input deck parser and supported by OPM Flow.
30	FOAMROCK	Define Foam Rock Properties. Recognized by the input deck parser and supported by OPM Flow.
31	OVERBURD	Define Rock Overburden Pressure versus Depth Tables. Recognized by the input deck parser and supported by OPM Flow.
32	ROCK2D	Pore Volume Compaction versus Pressure and Sw Tables. Recognized by the input deck parser and supported by OPM Flow.
33	ROCK2DTR	Transmissibility Compaction versus Pressure and Sw Tables Recognized by the input deck parser and supported by OPM Flow.
34	ROCKCOMP	Activate Rock Compaction. Recognized by the input deck parser and supported by OPM Flow.
35	ROCKFRAC	Define the Rock Volume to Bulk Volume Fraction for All the Cells. Recognized by the input deck parser but not supported by OPM Flow
36	ROCKNUM	Define Rock Compaction Table Region Numbers. Recognized by the input deck parser and supported by OPM Flow.
37	ROCKOPTS	Define Rock Compaction and Compressibility Options. Recognized by the input deck parser but not supported by OPM Flow
38	ROCKPAMA	Define Coal Palmer-Mansorri Rock Model Parameters. Recognized by the input deck parser but not supported by OPM Flow
39	ROCKTAB	Rock Compaction Tables. Recognized by the input deck parser but not supported by OPM Flow
40	ROCKTABH	Rock Compaction Hysteresis Tables. Recognized by the input deck parser but not supported by OPM Flow
41	ROCKTABW	Rock Compaction Tables (Water Induced). Recognized by the input deck parser but not supported by OPM Flow

No.	Keyword	Comment
42	ROCKWNOD	Water Saturation Values for Compaction Pressure-Sw Tables. Recognized by the input deck parser but not supported by OPM Flow
43	WFOAM	Define Well Foam Injection Concentrations. Recognized by the input deck parser and supported by OPM Flow.
44	WLIST	Define Well Lists (Static). Recognized by the input deck parser and supported by OPM Flow.
45	WLISTARG	Modify Well List Target and Constraint Values (Static). Recognized by the input deck parser but not supported by OPM Flow
46	WLISTNAM	Define Well Lists (WLISTARG). Recognized by the input deck parser but not supported by OPM Flow.

Table B.1: New Keywords for the 2019-10 Release

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).

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.

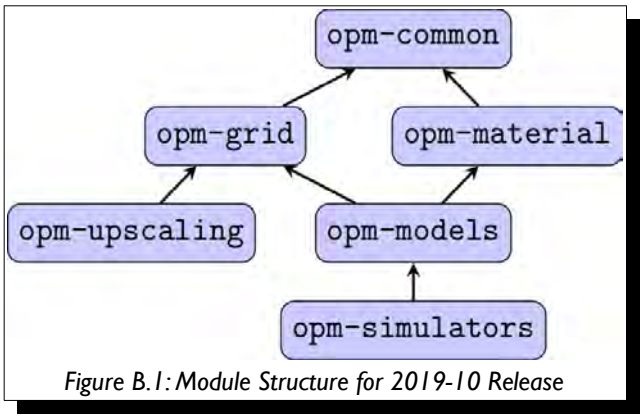


Figure B.1: Module Structure for 2019-10 Release

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.2 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 2019-10 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.2.

No.	Keyword	Comment
1	AQUFETP	The AQUFETP keyword defines Fetkovich Analytical aquifers and the aquifer properties. Fully functional.
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. Fully functional.
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. Fully functional.
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. Fully functional.
	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. Not supported by OPM Flow.
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.

No.	Keyword	Comment
8	MULTIN	This keyword switches on the Multiple Input Files option for all input files. Fully functional.
8	MULTOUT	This keyword switches on the Multiple Output Files option for all output files. Fully functional.
9	OVERBURD	Recognized by the input deck parser and simulator support is available in the experimental "ebos" simulator. Not supported by OPM Flow.
10	ROCK2D	Recognized by the input deck parser and simulator support is available in the experimental "ebos" simulator. Not supported by OPM Flow.
11	ROCK2DTR	Recognized by the input deck parser and simulator support is available in the experimental "ebos" simulator. Not supported by OPM Flow.
12	ROCKWNOD	Recognized by the input deck parser and simulator support is available in the experimental "ebos" simulator. Not supported by OPM Flow.
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. Not supported by OPM Flow.
18	THPRESFT	Recognized by the input deck parser and simulator support is available in the experimental "ebos" simulator. Not supported by OPM Flow.

No.	Keyword	Comment
19	WLIST	WLIST declares a group of wells to belong to a named well list. Fully functional.

Table B.2: New Keywords for the 2019-04 Release

Bård Skaflestad, Tor Harald Sandve, and David Baxendale.

B.3 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 *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 *5.2.115 THERMAL– Activate the Thermal Modeling Option* keyword in the RUNSPEC section outlining the available keywords.

Apart form the new thermal keywords summarized in section *5.2.115 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. Fully functional.
3	TOLCRIT	This keyword defines the Critical Saturation Tolerance. Fully functional.
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. Full functional.

Table B.3: New Keywords for the 2018-10 Release

Joakim Hove

B.4 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 *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.

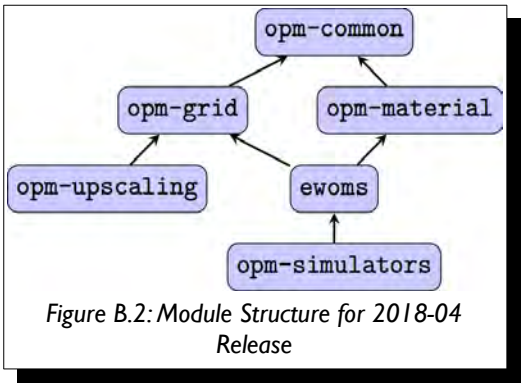


Figure B.2: Module Structure for 2018-04 Release

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. Fully functional.
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. Fully functional.
3	AQUDIMS	The AQUDIMS keyword defines the dimensions of the various aquifer property data. Fully functional.
4	AQUTAB	AQUTAB keyword defines additional Carter-Tracy aquifer functions to be used in the model. Fully functional.
5	COMSEGS	The COMSEGS keyword defines how a multi-segment well is connected to the reservoir by defining or modifying existing well connections.. Fully functional.
6	DRSDT	DRSDT defines the maximum rate at which the solution gas-oil ratio (Rs) can be increased in a grid cell. Fully functional.
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. Fully functional.
8	WELSEGS	The WELSEGS keyword defines a well to be a multi-segment well and defines the well's segment structure. Fully functional.

No.	Keyword	Comment
9	WSEGSDIMS	The WSEGSDIMS keyword defines the multi-segment well dimensions for the multi-segment well model. Fully functional.

Table B.4: New Keywords for the 2018-04 Release

Tor Harald Sandve

B.5 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.6 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.7 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: RUNNING PREVIOUS RELEASES OF OPM FLOW

C.1 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 C.1.

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: 4) 0: bicgstab, 5) 1: cg, 6) 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

OPM Flow 2019-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
5	--cpr-solver-verbose	A positive integer value that defines the output from CPR solver: 1) 0: no output 2) 1: output summary of inner linear solver 3) 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
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

OPM Flow 2019-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
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 *5.2.79 NOSIM – Activate the No Simulation Mode for Data File Checking*).	true
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 2019-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
	OPM Flow Specific Command Line Parameters		
28	--flow-linear-solver-verbosity	A positive integer value that defines the output from linear solver: 1) 0: no extra output 2) 1: output per solution iteration 3) >1: 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
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 pre-conditioner.	0
35	--ilu-redblack	A Boolean value set to true or false that instructs OPM Flow to use red-black partitioning for the ILU pre-conditioner. (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 pre-conditioner	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 pre-conditioner 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

OPM Flow 2019-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
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
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 pre-conditioner 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

OPM Flow 2019-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
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.	""
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 *.DEBUG 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 pre-conditioner matrix only. **Note this parameter is missing from the help when executing:** `flow -h`	false
59	--pri-var-oscilation-threshold	A real positive vale 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: 1) 0: No output to the files. 2) 1: Output *.DBG file 3) 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: 1) 0: No output to the files. 2) 1: Output *.DBG file 3) 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

OPM Flow 2019-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
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
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

OPM Flow 2019-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
71	--threads-per-process	A positive integer value that stipulates the maximum number of threads to be instantiated per process ('-1' means 'automatic').	1
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[154]. 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 Newtonian iterations which the time step control scheme should aim for (if applicable).	8

[154] *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**
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
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.	1.0×10^{6}
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 pre-conditioner	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 pre-conditioner	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

OPM Flow 2019-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
	VTK Graphics Command Line Parameters[155]		
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

[155] *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 C.1: OPM Flow 2019-04 Command Line Options

As per all UNIX and LINUX based system 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 CASNAME.DATA
```

or:

```
flow --EnableDryRun=false CASNAME.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 CASNAME.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.

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
#
ecl-output-dir=OPM
#
# ------------------------------------------------------------------------
# NEWTON SOLVER PARAMETER
# ------------------------------------------------------------------------
#
# Define Numerical Tolerances
#
flow-tolerance-mb=1e-5
flow-tolerance-cnv=1e-2
flow-tolerance-wells=1e-2
#
# Set Min Newtonian Solver iterations to 1 and Max to 15
#
flow-newton-min-iterations=1
flow-newton-max-iterations=15
#
# ------------------------------------------------------------------------
```

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

C.2 RUNNING OPM FLOW 2018-10

OPM Flow release 2018-10 and beyond have switched to the eWoms/ebos[156] 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 depreciated 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 C.2.

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

[156] *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**
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
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 *5.2.79 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: 1) 0: no extra output 2) 1: output per solution iteration 3) >1: 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 pre-conditioner.	0
26	--ilu-redblack	A Boolean value set to true or false that instructs OPM Flow to use red-black partitioning for the ILU pre-conditioner. (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 pre-conditioner	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 pre-conditioner 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 pre-conditioner 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 *.DEBUG 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 pre-conditioner matrix only.	false
50	--pri-var-oscilation-threshold	A real positive vale 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: 1) 0: No output to the files. 2) 1: Output *.DBG file 3) 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: 1) 0: No output to the files. 2) 1: Output *.DBG file 3) 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[157]. 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 Newtonian 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

[157] *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.	1.0×10^{5}
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 pre-conditioner	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[158]		
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

[158] *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**
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
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

OPM Flow 2018-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
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
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

OPM Flow 2018-10 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
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 C.2: OPM Flow 2019-04 Command Line Options

As per all UNIX and LINUX based system 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 CASNAME.DATA
```

or:

```
flow --EnableDryRun=false CASNAME.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 CASNAME.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.

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
#
ecl-output-dir=OPM
#
# ------------------------------------------------------------------------
# NEWTON SOLVER PARAMETER
# ------------------------------------------------------------------------
#
# Define Numerical Tolerances
#
flow-tolerance-mb=1e-5
flow-tolerance-cnv=1e-2
flow-tolerance-wells=1e-2
#
# Set Min Newtonian Solver iterations to 1 and Max to 15
#
flow-newton-min-iterations=1
flow-newton-max-iterations=15
#
# ------------------------------------------------------------------------
```

Notice that the leading "--" have not be incorporated in the parameter file, as per the notes in Table C.2.

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
```

C.3 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[159] 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 C.3.

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 *.DEBUG files: 4) none: No output to the files. 5) log or false: Output logging information only. 6) 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

[159] *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: 1) 0: no extra output 2) 1: output per solution iteration 3) >1: 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 pre-conditioner instead of ilu.	false
13	ilu_relaxation	A real positive double precision value that sets the relaxation parameter for the ILU pre-conditioner.	0.9
14	ilu_fillin_level	A positive integer value that sets the fill in level for the ILU pre-conditioner.	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
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

OPM Flow 2018-04 Command Line Options			
No.	**Variable Name**	**Description**	**Default**
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[160]. 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 C.3: OPM Flow 2018-04 Command Line Options

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

As per all UNIX and LINUX based system 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 CASNAME.DATA
```

or:

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

but in the parameter file use:

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

enable-dry-run=false
```

or:

```
EclDeckFileName=CASENAME.DATA

EnableDryRun=false
```

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
//
deck_filename=NORNE_ATW2013.DATA
//
// Output and Output Directory
//
output=true
output_dir=OPM
// ------------------------------------------------------------------------
// NEWTON SOLVER PARAMETER
// ------------------------------------------------------------------------
//
//Define Numerical Tolerances
//
tolerance_mb=1e-5
tolerance_cnv=1e-2
tolerance_wells=1e-2
//
// Set Min Newtonian Solver iterations to 1 and Max to 15
//
min_iter=1
max_iter=15
//
// Set Residual Threshold for Time Step Chop (Restart Solver)
//
max_residual_allowed=1e5
//
// USE BiCG Solver
//
newton_use_gmres=false
//
// Set Linear Solver Parameters
```

```
//
linear_solver_reduction=0.01
linear_solver_maxiter=50
linear_solver_restart=40
// ------------------------------------------------------------------------
// TIME STEPPING PARAMETERS
// ------------------------------------------------------------------------
//
// Set Time Stepping Scheme Option to Adaptive and Control Scheme
//
timestep.adaptive=true
timestep.control=pid+iteration
//
// PID Control Tolerance (default = 1e-3)
//
timestep.control.tol=4e-5
//
// Set Target iteration that is the Sum of all Linear Iterations Over All
// Newton Iterations per Time Step
//
timestep.control.targetiteration=8
//
// Set the Minimum Allowed Value a Time Step Can be Decreased
// After the Solver Failes to Converge
//
solver.restartfactor=0.3
//
// Set the Maximum Allowed Value a Time Step Can be Increased by,
// Subject to the Maximum Allowable Time Step Size
//
timestep.control.maxgrowth=1.6
//
// Max Number of Restarts (Time Step Chops) Before the Simulation is Terminated
//
solver.restart=10
//
// Verbosity of Solver and Adaptive Time Stepping
//
solver.verbose=true
timestep.verbose=true
// ------------------------------------------------------------------------
```

APPENDIX D: OPM FLOW OUTPUT FILE FORMATS

D.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.

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 2019-10 From The Command Line* for the various command line options for setting the output format type.

D.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 D.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.	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. 2) The extended GRID file format. 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.	Fully Supported

File Type	Data Type	Description	OPM Flow Status
INIT	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 file being written out.	Fully Supported
LOG	Output File	The LOG file for the commercial simulator contains an ASCII copy of the output from the terminal. 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` Will copy the terminal output to the CASENAME.LOG file.	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.	Fully Supported
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. The data is used to visualize the simulation results of the model in two and three dimensional space using post-processing software. The data is also used to "restart" from a previous simulation case.	Fully Supported

File Type	Data Type	Description	OPM Flow Status
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. 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 to compare the actual wire line logging data with the simulation derived results in post processing software.	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 *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.	Not 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
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.	Fully Supported

Notes:

1) All files can be written out in either ASCII or binary formats, except for DBG , LOG and PRT 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 D.1: OPM Flow Output File Types Summary

As mentioned in Table D.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) is set via a set of keywords in the RUNSPEC section, as outlined in Table D.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
	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 *.PRT, *.LOG and *.DEBUG 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
	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

Process	RUNSPEC Keyword	Description	Files

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 D.2: RUNSPEC Input and Output File Format Keywords

D.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 D.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 D.3.

File Type	Unformatted Extension	Description	Formatted Extension
DATA	*.DATA	Input data file.	*.DATA
DBG	*.DBG	Output debug file.	*.DBG
EGRID	*.EGRID	EGRID files contain the same information as the GRID files but employs the extensible GRID format, which is more compact than the GRID formatted files 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	*.LOG	Output log file.	*.LOG
PRT	*.PRT	Output print file.	*.PRT

File Type	Unformatted Extension	Description	Formatted Extension
RESTART Index	*.RSSPEC	The RESTART index file for both unified and non-unified formats.	*.FRSSPEC

RESTART Data Non-Unified	*.Xnnnn	The RESTART data files containing the solution arrays requested to be written to the RESTART files. For example, typing the following command from the terminal:
		`flow CASENAME`
		Will generate the following RESTART data non-unified binary files:
		`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, typing the following command from the terminal:
		`flow CASENAME`
		Will generate the following SUMMARY data unified binary files:
		`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, typing the following command from the terminal:
		`flow CASENAME`
		Will generate the following RFT data binary file:
		`CASENAME.RFT`
			*.FRFT

RSM	*.RSM	Output RSM file.	*.RSM
SAVE	*.SAVE	The SAVE file type is currently not supported by OPM Flow.	*.FSAVE
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, typing the following command from the terminal:
		`flow CASENAME`
		Will generate the following SUMMARY non-unified binary files:
		`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, typing the following command from the terminal:
		`flow CASENAME`
		Will generate the following SUMMARY data unified binary files:
		`CASENAME.UNSMRY`
		`CASENAME.SMSPEC`
			*.FUNSMRY

File Type	Unformatted Extension	Description	Formatted Extension

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.

Table D.3: OPM Flow File Naming Conventions

D.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 (2^{31} 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 for 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 D.4: Big-Endian and Little-Endian Representation

Developers using the Intel FORTRAN or GNU FORTRAN compilers need to use CONVERT='BIG_ENDIAN' option in the FORTRAN OPEN statement when opening unformatted files. For "LOGICAL" data "false" has all bits set to zero and "true" has at least one non-zero bit.

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. For example, OPM Flow's grid file writing support is implemented via the ecl_grid_fwrite_EGRID2() library function from LibECL. In turn, the fwrite_EGRID2() function unconditionally writes unformatted files:

```
void ecl_grid_fwrite_EGRID2(ecl_grid_type * grid, const char * filename,
          ert_ecl_unit_enum output_unit) {
  bool fmt_file        = false;
  fortio_type * fortio = fortio_open_writer(filename, fmt_file, ECL_ENDIAN_FLIP);
```

Where fmt_file is a flag that indicates whether or not to create a formatted output file (fmt_file = true) or an unformatted output file (fmt_file = false).

The following sections outline the format of the various individual files supported by OPM Flow, except for the DBG, LOG and PRT ASCII format files.

D.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 records that define the header record data type, the number of data entries, and then the associated header record data. There are a total of four major record header types for this file type (1) File Header, (2) Global Grid Header, (3) Local Grid Refinement Header (repeated for each local grid refinement in the model), and (4) Non-Neighbor Connection Header. Each major header is subdivided into a series of sub-headers that define a particular data set record. The general format for header record is the record name (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).

For a formatted EGRID file (*.FGRID) file structure for a global grid is of the form:

```
 '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  ' '         '
 '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'
 '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
...........................................
```

Whereas the unformatted file looks something like this if viewed in a text editor:

```
    FILEHEAD   dINTE                Ô



                                   GRIDHEAD   dINTE



                                                                                                  COORD       Ö
REAL        X                                    Dz   Dz            Dz       Dz   Dú                 Dú
Dz   E;€             E;€       Dz   Ez                   Ez        Dz   Eœ@            Eœ@       Dz   E»€
          E»€        Dz   EÚÀ             EÚÀ       Dz   Eú             Eú        Dz   F
F         Dz   F @             F @       Dz        Dz                  Dz   Dz   Dz   Dz        Dz   Dz
Dz   Dú   Dz         Dú   Dz   Dz   E;€  Dz        E;€  Dz   Dz   Ez   Dz        Ez   Dz   Dz   Eœ@
Dz        Eœ@  Dz   Dz   E»€  Dz        E»€  Dz   Dz   EÚÀ  Dz        EÚÀ  Dz   Dz   Eú   Dz
Eú   Dz   Dz   F    Dz        F    Dz   Dz   F @  Dz        F @  Dz   Dz        Dú             Dú
Dz   Dz   Dú        Dz   Dú   Dz   Dú   Dú        Dú   Dú   Dz   E;€  Dú        E;€  Dú   Dz   Ez
Dú        Ez   Dú   Dz   Eœ@  Dú        Eœ@  Dú   Dz   E»€  Dú        E»€  Dú   Dz   EÚÀ  Dú
EÚÀ  Dú   Dz   Eú   Dú        Eú   Dú   Dz   F    Dú        F    Dú   Dz   F @  Dú        F @  Dú
Dz        E;€            E;€  Dz   Dz   E;€       Dz   E;€  Dz   Dú   E;€       Dú   E;€  Dz   E;€
E;€       E;€  E;€  Dz   Ez   E;€       Ez   E;€  Dz   Eœ@  E;€       Eœ@  E;€  Dz   E»€  E;€
E»€  E;€  Dz   EÚÀ  E;€       EÚÀ  E;€  Dz   Eú   E;€       Eú   E;€  Dz   F    E;€       F    E;€
Dz   F @  E;€       F @  E;€  Dz        Ez             Ez   Dz   Dz   Ez        Dz   Ez   Dz   Dú
Ez        Dú   Ez   Dz   E;€  Ez        E;€  Dz   Ez   Ez        Ez   Ez   Dz   Eœ@  Ez
Eœ@  Ez   Dz   E»€  Ez        E»€  Ez   Dz   EÚÀ  Ez        EÚÀ  Ez   Dz   Eú   Ez        Eú   Ez
Dz   F    Ez        F    Ez   Dz   F @  Ez        F @  Ez   Dz        Eœ@            Eœ@  Dz   Dz
Eœ@       Dz   Eœ@  Dz   Dú   Eœ@       Dú   Eœ@  Dz   E;€  Eœ@       E;€  Eœ@  Dz   Ez   Eœ@
Ez   Eœ@  Dz   Eœ@  Eœ@       Eœ@  Eœ@  Dz   E»€  Eœ@       E»€  Eœ@  Dz   EÚÀ  Eœ@       EÚÀ  Eœ@
Dz   Eú   Eœ@       Eú   Eœ@  Dz   F    Eœ@       F    Eœ@  Dz   F @  Eœ@       F @  Eœ@  Dz
```

Figure D.1: EGRID Unformatted File Format Viewed in a Text Editor

The following sections describe the format of the various record header data and the associated data set.

D.2.1 EGRID HEADER RECORD

The FILEHEAD record defines the key attributes of the file via a series of integer constants, this is then followed by the MAPUNITS, MAPAXES, GRIDUNIT and GDORIENT records and their associated data sets. Note that some records 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 D.5 outlines the structure of the File Header Record and the associated sub-records for this data set.

No.	Record Name	EGRID Header Record			Status Or Value
		Keyword	No. of Entries	Data Type	
	FILEHEAD	File header record for the EGRID file defines the start of this record type.			Required
1-1	Format	FILEHEAD	100	INTE	
2-1	Data	Version number for this file type, for example 3.			
2-2	Data	The year this version of the file format was released, for example 2004.			
2-3	Data	Not used.			
2-4	Data	Version number of earliest this file format is supported, normally set to 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).			
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.			
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	Map units header record that defines the start the MAPUNITS record Normally set to FEET or METRES.			Optional
1-1	Format	MAPUNITS	1	CHAR	
2-1	Data	The units of the map data, normally set to FEET or METRES.			
	MAPAXES	Start of the MAPAXES header record data that 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.			

No.	Record Name	EGRID Header Record			Status Or Value
		Keyword	No. of Entries	Data Type	
	GRIDUNIT	The GRIDUNIT header record 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 (' ').			
	GDORIENT	Grid orientation header record, that 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 sub-records in this header record type.
2) Item No. cells colored orange indicate that this item is not supported by OPM Flow and thos colred red are either "Not Used" or "Undefined".
3) Note all character variables, including the Header Record Name should be enclosed in single quotations, for example the FILEHEAD header Record Name should be written out as 'FILEHEAD'.
4) GDORIENT keyword in the GRID section is not supported by OPM Flow.

Table D.5: EGRID Header Record

Example: EGRID Header Record

The following example shows a typical formatted file header record 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           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 record is not written out.

D.2.2 EGRID GLOBAL IRREGULAR CORNER POINT GRID RECORD

Following the File Header data set, the Global Grid Header data set 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 D.6.

No.	Record Name	Global Irregular Corner Point Grid Record			Status Or Value
		Keyword	No. of Entries	Data Type	
	GRIDHEAD	Global grid header record for the EGRID file defines the start of this record 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.			
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 0 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 refence 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.			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 girds 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
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

No.	Record Name	Global Irregular Corner Point Grid Record			Status Or Value
		Keyword	No. of Entries	Data Type	
	BOXORIG	Not supported.			Optional
1-1	Format	BOXORIG	3	INTE	
2-1 to 2-3	Data	Ignored by OPM Flow.			
	COORD	Property header record 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 header record 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			
	COORDSYS	Coordinate definitions for each NUMRES reservoir. The number of data values for this header record 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 header record 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 header record 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			
	ACTNUM	Property header record 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 is status is set to Optional, it is normally always written out by pre-processing software. The number of data values for this header record 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			

No.	Record Name	Global Irregular Corner Point Grid Record			Status Or Value
		Keyword	No. of Entries	Data Type	
	CORSNUM	Property header record that defines the start of the CORSNUM data set., for when the grid has be coarsened Grid coarsening is currently not supported in OPM Flow. The number of data values for this header record be NX x NY x NZ integer values.			Optional
1-1	Format	CORSNUM	NX x NY x NZ	INTE	
2-1	Data	CORSNUM data set.			
1-1	ENDGRID	The ENDGRID header record marks the end of the Global Grid Header section. Always set to 0 to indicate that there is no Data section for this record type.			Required
1-2	Format	ENDGRID	0	INTE	

Notes:

1) Rows shaded in gray indicate sub-records in this header record type.
2) Item No. cells colored orange indicate that this item is not supported by OPM Flow and those colored red are either "Not Used" or "Undefined".
3) Note all character variables, including the Header Record Name should be enclosed in single quotations, for example the FILEHEAD header Record Name should be written out as 'FILEHEAD'.

Table D.6: Global Irregular Corner Point Grid Record

Example: Global Irregular Corner Point Grid Record

The following example shows a typical formatted global grid header record 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           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

D.2.3 EGRID LGR IRREGULAR CORNER POINT GRID RECORD

The file structure for Local Gride Refinement ("LGR") record for Irregular Corner Point Grids is similar to the global grid record 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 record is repeated for each LGR in the model. The record description is outlined in Table D.7.

Note that currently OPM Flow does not support LGR grids and therefore this record format cannot be used by OPM Flow.

No.	Header Record Name	EGRID LGR Irregular Corner Point Grid Record (Repeated For Each LGR In The Model)			Status Or Value
		Keyword	No. of Entries	Data Type	
	LGR	LGR grid header record for the EGRID file defines the start of this record type 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 record data set that defines the parent LGR for when nested LGR have been used.			Required
1-1	Format	LGRPARNT	1	CHAR	
2-1	Data	Name of parent LGR			
	LGRSGRID	Start of the LGRSGRID record for LGRs parented from a sub-grid.			Optional
1-1	Format	LGRSGRID	1	CHAR	
2-1	Data	Name of parent LGR			
	GRIDHEAD	Global grid header record for the EGRID file defines the start of this record type. The format is the same as the Global grid see Table D.6.			Required
	BOXORIG	Not supported. The format is the same as the Global grid see Table D.6.			Optional
	COORD	Property header record that defines the start of the LGR COORD data set. The COORD data defines a set of coordinate lines or pillars for athe LGR grid via an array. The format is the same as the Global grid see Table D.6.			Required
	COORDSYS	Coordinate definitions for the LGR. The format is the same as the Global grid see Table D.6.			Optional
	ZCORN	Property header record 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 format is the same as the Global grid see Table D.6.			Required
	ACTNUM	Property header record 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 is status is set to Optional, it is normally always written out by pre-processing software. The format is the same as the Global grid see Table D.6.			Optional

No.	Header Record Name	EGRID LGR Irregular Corner Point Grid Record (Repeated For Each LGR In The Model)			Status Or Value
		Keyword	No. of Entries	Data Type	
	HOSTNUM	Property header record that defines the start of the HOSTNUM data set. which is only applicable to LGR grids. Number of data values for this header record 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 header record marks the end of the Global Grid Header section. Always set to 0 to indicate that there is no Data section for this record type.			Required
1-1	Format	ENDGRID	0	INTE	
	ENDLGR	ENDLGRecord marks the end of the LGR Header section. Always set to 0 to indicate that there is no Data section for this record type.			Optional
1-1	Format	ENDLGR	0	INTE	

Notes:

1) Rows shaded in gray indicate sub-records in this header record type.
2) Item No. cells colored orange indicate that this item is not supported by OPM Flow and those colored red are either "Not Used" or "Undefined".
3) Note all character variables, including the Header Record Name should be enclosed in single quotations, for example the LGR header Record Name should be written out as 'LGR '.
4) COORDSYS keyword in the GRID section is not supported by OPM Flow.

Table D.7: EGRID LGR Irregular Corner Point Grid Records

Note that this record type is repeated for each LGR in the model as demonstrated in the following example.

Example: EGRID LGR Irregular Corner Point Grid Record

The following example shows a typical formatted global grid header record 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 Grid Header Record for LGR Irregular Corner Point Grids is 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
           2           2           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
 '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
  0.00000000E+00   0.10000000E+04   0.15000000E+04   0.10000000E+04
  0.15000000E+04   0.15000000E+04   0.00000000E+00   0.15000000E+04
  0.15000000E+04   0.10000000E+04   0.20000000E+04   0.15000000E+04
  0.00000000E+00   0.20000000E+04   0.15000000E+04   0.10000000E+04
  0.10000000E+04   0.20000000E+04   0.00000000E+00   0.10000000E+04
  0.20000000E+04   0.10000000E+04   0.15000000E+04   0.20000000E+04
  0.00000000E+00   0.15000000E+04   0.20000000E+04   0.10000000E+04
  0.20000000E+04   0.20000000E+04   0.00000000E+00   0.20000000E+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
  0.83350000E+04   0.83350000E+04   0.83350000E+04   0.83350000E+04
  0.83350000E+04   0.83350000E+04   0.83350000E+04   0.83350000E+04
  0.83350000E+04   0.83350000E+04   0.83350000E+04   0.83350000E+04
  0.83350000E+04   0.83350000E+04   0.83350000E+04   0.83350000E+04
  0.83350000E+04   0.83350000E+04   0.83350000E+04   0.83350000E+04
  0.83350000E+04   0.83350000E+04   0.83350000E+04   0.83350000E+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
  0.83450000E+04   0.83450000E+04   0.83450000E+04   0.83450000E+04
  0.83450000E+04   0.83450000E+04   0.83450000E+04   0.83450000E+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           0           0
          0           0           0           0           0           0
          0           0           0           0           0           0
          1           1           0           9           9           1
          9           9           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
'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
  0.00000000E+00   0.85000000E+04   0.80000000E+04   0.10000000E+04
  0.90000000E+04   0.80000000E+04   0.00000000E+00   0.90000000E+04
  0.80000000E+04   0.10000000E+04   0.80000000E+04   0.85000000E+04
  0.00000000E+00   0.80000000E+04   0.85000000E+04   0.10000000E+04
  0.85000000E+04   0.85000000E+04   0.00000000E+00   0.85000000E+04
  0.85000000E+04   0.10000000E+04   0.90000000E+04   0.85000000E+04
  0.00000000E+00   0.90000000E+04   0.85000000E+04   0.10000000E+04
  0.80000000E+04   0.90000000E+04   0.00000000E+00   0.80000000E+04
  0.90000000E+04   0.10000000E+04   0.85000000E+04   0.90000000E+04
  0.00000000E+00   0.85000000E+04   0.90000000E+04   0.10000000E+04
  0.90000000E+04   0.90000000E+04   0.00000000E+00   0.90000000E+04
  0.90000000E+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
  0.83350000E+04   0.83350000E+04   0.83350000E+04   0.83350000E+04
  0.83350000E+04   0.83350000E+04   0.83350000E+04   0.83350000E+04
  0.83350000E+04   0.83350000E+04   0.83350000E+04   0.83350000E+04
  0.83350000E+04   0.83350000E+04   0.83350000E+04   0.83350000E+04
  0.83350000E+04   0.83350000E+04   0.83350000E+04   0.83350000E+04
  0.83350000E+04   0.83350000E+04   0.83350000E+04   0.83350000E+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
  0.83450000E+04   0.83450000E+04   0.83450000E+04   0.83450000E+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

D.2.4 EGRID NON-NEIGHBOR CONNECTIONS FOR IRREGULAR CORNER POINT GRID RECORD

No.	Header Record Name	EGRID Non-Neighbor Connections For Irregular Corner Point Grid Record			Status Or Value
		Keyword	No. of Entries	Data Type	
	NNCREAD	This record marks the start of the non-neighbor records and defines the number of non-neighbor connections.			
1-1	Format	NNCREAD	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 the LGR number.			
2-3 to 2-10	Data	Not used.			
	NNC1	NNC1 starts the record for the cell numbers for UPSTREAM non-neighbor connections, where NUMNNC is the number of non-neighbor connections.			
1-1	Format	NNC1	NUMNNC	INTE	
2-1	Data	NUMNNC integer values defining the cell numbers for UPSTREAM non-neighbor connections.			
	NNC2	NNC2 starts the record for the cell numbers for DOWNSTREAM non-neighbor connections, where NUMNNC is the number of non-neighbor connections			
1-1	Format	NNC2	NUMNNC	INTE	
2-1	Data	NUMNNC integer values defining the cell numbers for DOWNSTREAM non-neighbor connections.			
	NNCL	This record defines the local cells connected to the global grid, where NCONGL is the number of entries.			
1-1	Format	NNCL	NCONCL	INTE	
2-1	Data	NCONCL integer values defining the local cells connected to the global grid.			
	NNCG	This record defines the global cells connected to the current local grid being defined on this record, where NCONGL is the number of entries.			
1-1	Format	NNCG	NCONGL	INTE	
2-1	Data	NCONGL integer values defining the global cells connected to the current local grid.			
	NNCHEADA	This record defines the start of records that specify the connection between two LGRs that have been amalgamated.			
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).			

No.	Header Record Name	EGRID Non-Neighbor Connections For Irregular Corner Point Grid Record			Status Or Value
		Keyword	No. of Entries	Data Type	
2-3 to 2-10		Not used			
	NNA1	This record defines the ILOC1 cells connected in the amalgamation, where NUMNCA is the number of entries.			
1-1	Format	NNA1	NUMNCA	INTE	
2-1	Data	NUMNCA integer values defining the ILOC1 cells connected in the amalgamation.			
	NNA2	This record 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) Note all character variables, including the Header Record Name should be enclosed in single quotations, for example the FILEHEAD header Record Name should be written out as 'NNCHEAD '.

2) Item No. cells colored orange indicate that this item is not supported by OPM Flow and those colored red are either "Not Used" or "Undefined".

3) Local Grid Refinements are currently not supported by OPM Flow and neither therefore is LGR amalgamation of LGRs.

Table D.8: EGRID Non-Neighbor Connections for Irregular Corner Point Grid Record

Example: EGRID Grid Record for Non-Neighbor Connections for Irregular Corner Point Grids

```
'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

D.3 EGRID - MODEL STRUCTURAL DATA FOR UNSTRUCTURED GRIDS FILE

This file format is currently not supported by OPM Flow.

D.4 EGRID - MODEL STRUCTURAL DATA FOR MIXED GRIDS FILE

This file format is currently not supported by OPM Flow.

D.5 GRID - MODEL STRUCTURAL DATA FILE

This file format is currently not supported by OPM Flow.

D.6 INIT – MODEL INITIALIZATION AND STATIC DATA FILE

To be written in a future release of the manual.

D.6.1 INIT INDEX FILE

To be written in a future release of the manual.

D.6.2 INIT DATA FILE SPECIFICATION

This file type contains the global and LGR grid property data, 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. The individual data records are structured in as shown in Table D.9.

Reference Table	Global Records	Global Data Records	LGR Records	LGR Data Records	Notes
D.10	GRID DATA				Global grid data record for the INIT Data file that defines the start of global grid data
D.12	PROPERTY	Property 1			Global grid property records that defines the global grid property data defined in the GRID and Edit sections.
		Property 2			
		etc.			
D.18			LGR-1	Property 1	LGR property record that defines the start of the PORV global grid data set. for the first LGR This must be the first data set for this record type. Subsequent PROPERTY records can be in any order, and the record is repeated until all the PROPERTY data has been defined for an LGR. The LGR records are repeated for each LGR in the model
				Property 2	
				etc.	
			LGR-2	Property 1	
				Property 2	
				etc.	
			etc.		
			LGRSGONE		End of LGR section record.
	TABDIMS				Tabulated data, PVT tables, relative permeability tables etc. records
	TAB				
	CON				
D.13	REGION	Region 1			Region property data for the global grid as defined in the REGION section (PVTNUM, SATNUM etc.). The record is repeated to account for all region property data.
		Region 2			
		etc.			
D.19			LGR-1	Region 1	LGR Region property data for the LGR grid as defined in the REGION section (PVTNUM, SATNUM etc.). The record is repeated to account for all region property data. The LGR records are repeated for each LGR in the model
				Region 2	
				etc.	
			LGR-2	Region 1	
				Region 2	
				etc.	
			etc.		
			LGRSGONE		End of LGR section record.
D.11	NNC	Nnc 1			Non-neighbor connection property data (NNC) for the global grid. The record is repeated to account for various NNC arrays.
		Nnc 2			
		etc.			

Reference Table	Global Records	Global Data Records	LGR Records	LGR Data Records	Notes
D.17			LGR-1	Nnc 1	LGR Region property data for the LGR grid as defined in the REGION section (PVTNUM, SATNUM etc.). The record is repeated to account for all region property data. The LGR records are repeated for each LGR in the model
				Nnc 2	
				etc.	
			LGR-2	Nnc 1	
				Nnc 2	
				etc.	
			etc.		
			LGRSGONE		End of LGR section record.
D.14	SATFUNS	SatFuns 1			Global Grid Saturation and End-Point Data Records. The record users the same keywords outlined in the GRID and PROPS section, for example, the SWL, SWATINIT, KRG, PCW, etc. arrays
		SatFuns 2			
		etc.			
D.20			LGR-1	Region 1	LGR Grid Saturation and End-Point Data Records. The record is repeated to account for all region property data. The LGR records are repeated for each LGR in the model
				Region 2	
				etc.	
			LGR-2	Region 1	
				Region 2	
				etc.	
			etc.		
			LGRSGONE		End of LGR section record.

Table D.9: INIT Data File Structure

D.6.3 INIT DATA - GLOBAL GRID DATA RECORD

The INIT Data - Global Grid Data Record is the first data set that should be read or written to the INIT file. This data set includes INITHEAD, LOGIHEAD and DOUBHEAD records that define versus parameters used in subsequent records in the INIT file. This is then followed by a series of global grid PROPERTY data records that defined the various global grid property arrays for each grid cell in the model.

The structure for this type of record is defined in Table D.10.

No.	Record Name	INIT Data - Global Grid Data Record			Status Or Value
		Keyword	No. of Entries	Data Type	
	SEQNUM	Report time step number, only used for unified restart files. Should be set to 0.			Optional
1-1	Format	SEQNUM	1	INTE	
2-1	Data	Report time step number.			0
	INITHEAD	Global grid header record for the INIT Data file that defines the start of INIT Global Grid Record and contains the integer variables for this header record.			Required
1-1	Format	INITHEAD	249	INTE	
2-1	Data	Undefined			0
2-2	Data	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	Data	Undefined.			0
2-9	Data	The number of grid blocks in the x-direction (NX).			
2-10	Data	The number of grid blocks in the y-direction (NY).			
2-11	Data	The number of grid blocks in the z-direction (NZ).			
2-12	Data	Number of global active cells in the model			
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.			
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.			
2-16 to 2-64	Data	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
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

No.	Record Name	INIT Data - Global Grid Data Record			Status Or Value
		Keyword	No. of Entries	Data Type	
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	Data	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.			
2-96 to 206	Data	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.			
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.			
2-209 to 410	Data	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.			SS
2-412 to 249	Data	Undefined.			0
	LOGIHEAD	Global grid record for the INIT Data file that defines the logical variables (T for true and F for false) for this record.			Required
1-1	Format	LOGIHEAD	79	LOGI	
2-1	Data	Dissolved gas is present in the model indicator, set to T if present in the model else set to F.			
2-2	Data	Vaporized oil is present in the model indicator, set to T if present in the model else set to F.			
2-3	Data	Model utilizes directional relative permeabilities in the model, set to T or F.			
2-4	Data	Model utilizes reversible directional relative permeabilities in the model indicator, set to T or F.			
2.5	Data	Radial grid geometry has been used for the model indicator, set to T or F.			
2.6	Data	Undefined.			0

No.	Record Name	INIT Data - Global Grid Data Record			Status Or Value
		Keyword	No. of Entries	Data Type	
2-7	Data	The hysteresis option has been activated in the model indicator, set to T or F.			
2-8 to 2-14	Data	Undefined.			0
2-15		The dual porosity option has been activated in the model indicator, set to T or F.			
2-16	Data	Undefined.			0
2-17	Data	The end-point scaling option has been activated in the model indicator, set to T or F.			
2-18	Data	The directional end-point scaling option has been activated in the model indicator, set to T or F.			
2.19	Data	The reversible end-point scaling option has been activated in the model indicator, set to T or F.			
2-20	Data	The alternative end-point scaling option has been activated in the model indicator, set to T or F. See the *SCALECRS – Define End-Point Scaling Option* keyword in the PROPS section for further information.			
2-21 to 2-35	Data	Undefined.			0
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	Data	Undefined.			0
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 2-79	Data	Undefined.			0
	DOUBHEAD	Global grid record for the INIT Data file that defines the double precision REAL variables for this record.			Required
1-1	Format	DOUBHEAD	185	DOUB	
2-1 to 2-185	Data	Undefined.			0.0

No.	Record Name	INIT Data - Global Grid Data Record			Status Or Value
		Keyword	No. of Entries	Data Type	

Notes:

1) Rows shaded in gray indicate sub-records in this header record type.

2) Item No. cells colored orange indicate that this item is not supported by OPM Flow and those colored red are either "Not Used" or "Undefined".

3) Note all character variables, including the Header Record Name should be enclosed in single quotations, for example the PORV and PROPERTY fields should be written out as 'PORV ' and 'DEPTH ', etc.

Table D.10: INIT Data - Global Grid Data Record

Example: INIT Data - Global Grid Data Record

The following example shows a typical formatted global grid property data record.

```
'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'
  0.000000000000000D+00   0.100000000000000D+01   0.365000000000000D+03
  0.100000000149012D+00   0.150000000596046D+00   0.300000000000000D+01
  0.300000001192093D+00   0.100000000149012D+00   0.100000000149012D+00
 ……….……..
 -0.100000000200409D+21   0.100000000000000D+01   0.100000000000000D+01
  0.100000000000000D+01   0.000000000000000D+00   0.000000000000000D+00
  0.100000000000000D+01   0.100000000000000D-03
```

End of Example

D.6.4 INIT DATA – GLOBAL GRID NON-NEIGHBOR CONNECTION DATA RECORD

The INIT Data – Global Grid Non-Neighbor Connection Data Record defines the Global Grid Non-Neighbor Connection arrays as generated by OPM Flow. This includes the TRANCC, DIFFNNC, HEATNNC, TRANGL arrays.

Item No.	Record Name	INIT Data – Global Grid Non-Neighbor Connection Data Record			Status Or Value
		Keyword	**No. of Entries**	**Data Type**	
	NNC	The NNC record defines the start of a gobal grid non-neighbor connection array. NNC is the keyword name of the non-neighbor connection array being written out, and should be set to: 1) DIFFNNC: the non-neighbor connection diffusivities within the LGR. 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. 2) HEATNNC: the non-neighbor connection thermal transmissibilities within the LGR. Only required if OPM Flow's THERMAL option has been activated by the THERMAL keyword in the RUNSPEC section. The following two addition NNC records 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 record data set should set to the number of non-neighbor connection to be read or written out (NUMNNC) for the TRANCC, DIFFNNC, and HEATNCC arrays The NNC record (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.			

Notes:

1) Rows shaded in gray indicate sub-records in this header record type.
2) Note all character variables, including the Header Record Name should be enclosed in single quotations, for example the LGR header Record Name should be written out as 'LGR '.

Table D.11: INIT Data – Global Grid Non-Neighbor Connection Data Record

This record type is repeated for each non-neighbor connection data set in the model; see the following example.

Example: INIT Data - Global Non-Neighbor Connection Data Record

The following example shows a typical formatted INIT Data - Global Non-Neighbor Connection Data Record.

```
 'TRANNNC '       11363 'REAL'
   0.52157912E-03   0.60928073E-05   0.73730538E-03   0.51154610E-03
   0.36839943E-03   0.44034168E-03   0.14111123E-02   0.27143997E-02
   0.14248013E-02   0.18686920E-02   0.27998611E-02   0.76166220E-03
   0.72630920E-03   0.34133402E-04   0.24275912E-04   0.19163921E-03
.....................................
```

End of Example

D.6.5 INIT DATA - GLOBAL GRID PROPERTY DATA RECORD

The INIT Data - Global Grid Property Data Record records defined various global grid property arrays for each grid cell in the model. The PROPERTY record users 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 PROPERTY records also include the calculated pore volumes (PORV) and transmissibility arrays (TRANX, TRANY and TRANZ) after processing the GRID and EDIT sections.

Note for the PROPERTY record the first property should always be the PORV array.

The structure for this type of record is defined in Table D.12.

No.	Record Name	INIT Data - Global Grid Property Data Record			Status Or Value
		Keyword	No. of Entries	Data Type	
	PROPERTY	The PROPERTY record defines the start of a global 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 record (items 1-1 and 2-1) is repeated for each global property. The PORV property must be the first data set for this record type and is written out in full for each cell in the model (that is NXYZ = NX x NY x NZ values). Note the array values for inactive cells etc. are set to a constant value of zero. Subsequent global 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.			

Notes:

1) Rows shaded in gray indicate sub-records in this header record type.

2) Note all character variables, including the Header Record Name should be enclosed in single quotations, for example the PORV and PROPERTY fields should be written out as 'PORV ' and 'DEPTH ', etc.

Table D.12: INIT Data - Global Grid Property Data Record

Example: INIT Data - Global Grid Property Data Record

The following example shows a typical formatted global grid property data record.

```
 '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
   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
   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
   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
   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
  ..............................
   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
..............................
etc.
```

End of Example

D.6.6 INIT DATA - GLOBAL GRID REGION DATA RECORD

The INIT Data - Global Grid Region Data Record defines the global grid REGION arrays. The REGION record name users the same keywords outlined in the REGION section of the manual in naming the subsequent data arrays. For example, the EQLNUM, FIPNUM, PVTNUM, ROCKNUM, SATNUM arrays etc.

Note that this record type does not include the INITHEAD, LOGIHEAD and DOUBHEAD records and there is no specific terminating record for this record type

No.	Record Name	INIT Data - Global Grid Region Data Record			Status Or Value
		Keyword	No. of Entries	Data Type	
	REGION	The REGION record defines the start of the global grid region array. 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. Region global REGION arrays can be in any order and only the active cells are written out (NACTIV). The REGION record (items 1-1 and 2-1) is repeated for each global region array..			Required
1-1	Format	REGION	NACTIV	INTE	
2-1	Data	REGION data set.			

Notes:

1) Rows shaded in gray indicate sub-records in this header record type.

2) Note all character variables, including the Header Record Name should be enclosed in single quotations, for example the EQLNUM should be written out as 'EQLNUM '.

Table D.13: INIT Data - Global Grid Region Data Record

Example: INIT Data - Global Grid Region Data Record

The following example shows a typical formatted INIT Data - Global Grid Region Data Record data set.

```
'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
……….……..
           2           2           2           2           2           2
           2           2           2           2
'EQLNUM  '          100 'INTE'
           1           1           1           1           1           1
           1           1           1           1           1           1
……….……..
           1           1           1           1
'FIPNUM  '          100 'INTE'
           1           1           1           1           1           1
           1           1           1           1           1           1
……….……..
           1           1           1           1
```

End of Example

D.6.7 INIT DATA - GLOBAL GRID SATURATION AND END-POINT DATA RECORD

The INIT Data - Global Grid Saturation and End-Point Data Record defines the global grid saturation and end-point arrays. The SATFUNS record users the same keywords outlined in the GRID and PROPS section of the manual in naming the subsequent data arrays. For example, the SWL, SWATINIT, KRG, PCW, etc. arrays

Note that this record type does not include the INITHEAD, LOGIHEAD and DOUBHEAD records and there is no specific terminating record for this record type.

No.	Record Name	INIT Data - Global Grid Saturation And End-Point Data Record Data			Status Or Value
		Keyword	**No. of Entries**	**Data Type**	
	SATFUNCS	The SATFUNS record defines the start of the global grid saturation and end-point data REAL arrays. SATFUNS 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. Type of data in the Data fields should be set to REAL for arrays with real values (SWL, SWATINIT, etc) or INTE for integer arrays (ENDNUM etc.) Region global SATFUNS arrays can be in any order and only the active cells are written out (NACTIV). This record (items 1-1 and 2-1) is repeated for each global REAL saturation and end-point array being written out., as the arrays 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	SATFUNS	NACTIV	INTE or REAL	
2-1	Data	SATFUNS data set.			

Notes:

1) Rows shaded in gray indicate sub-records in this header record type.
2) Note all character variables, including the Header Record Name should be enclosed in single quotations, for example the EQLNUM should be written out as 'EQLNUM '.

Table D.14: INIT Data - Global Grid Saturation and End-Point Data Record

Example: INIT Data - Global Grid Saturation and End-Point Data Record

The following example shows a typical formatted INIT Data - Global Grid Saturation and End-Point Data Record.

```
  'SWCR    '         100 'REAL'
   0.24444000E+00   0.23999999E+00   0.23999999E+00   0.23999999E+00
   0.23999999E+00   0.79999998E-01   0.79999998E-01   0.79999998E-01
   0.79999998E-01   0.79999998E-01   0.23999999E+00   0.23999999E+00
……..……..
   0.79999998E-01   0.79999998E-01   0.79999998E-01   0.79999998E-01
   0.79999998E-01   0.79999998E-01   0.79999998E-01   0.79999998E-01
   0.79999998E-01   0.79999998E-01   0.79999998E-01   0.79999998E-01
 'SGCR    '         100 'REAL'
   0.10000000E+00   0.10000000E+00   0.10000000E+00   0.10000000E+00
   0.10000000E+00   0.30000001E+00   0.30000001E+00   0.30000001E+00
   0.30000001E+00   0.30000001E+00   0.10000000E+00   0.10000000E+00
……..……..
   0.30000001E+00   0.30000001E+00   0.30000001E+00   0.30000001E+00
   0.30000001E+00   0.30000001E+00   0.30000001E+00   0.30000001E+00
   0.30000001E+00   0.30000001E+00   0.30000001E+00   0.30000001E+00
 'SOWCR   '         100 'REAL'
   0.15000001E+00   0.15000001E+00   0.15000001E+00   0.15000001E+00
   0.15000001E+00   0.18000001E+00   0.18000001E+00   0.18000001E+00
   0.18000001E+00   0.18000001E+00   0.15000001E+00   0.15000001E+00
……..……..
   0.18000001E+00   0.18000001E+00   0.18000001E+00   0.18000001E+00
   0.18000001E+00   0.18000001E+00   0.18000001E+00   0.18000001E+00
   0.18000001E+00   0.18000001E+00   0.18000001E+00   0.18000001E+00
 'SOGCR   '         100 'REAL'
   0.15000001E+00   0.15000001E+00   0.15000001E+00   0.15000001E+00
   0.15000001E+00   0.18000001E+00   0.18000001E+00   0.18000001E+00
   0.18000001E+00   0.18000001E+00   0.15000001E+00   0.15000001E+00
……..……..
   0.18000001E+00   0.18000001E+00   0.18000001E+00   0.18000001E+00
   0.18000001E+00   0.18000001E+00   0.18000001E+00   0.18000001E+00
   0.18000001E+00   0.18000001E+00   0.18000001E+00   0.18000001E+00
 'SWL     '         100 'REAL'
   0.24444000E+00   0.23999999E+00   0.23999999E+00   0.23999999E+00
   0.23999999E+00   0.79999998E-01   0.79999998E-01   0.79999998E-01
   0.79999998E-01   0.79999998E-01   0.23999999E+00   0.23999999E+00
……..……..
   0.79999998E-01   0.79999998E-01   0.79999998E-01   0.79999998E-01
   0.79999998E-01   0.79999998E-01   0.79999998E-01   0.79999998E-01
   0.79999998E-01   0.79999998E-01   0.79999998E-01   0.79999998E-01
 'SWU     '         100 'REAL'
   0.10000000E+01   0.10000000E+01   0.10000000E+01   0.10000000E+01
   0.10000000E+01   0.10000000E+01   0.10000000E+01   0.10000000E+01
   0.10000000E+01   0.10000000E+01   0.10000000E+01   0.10000000E+01
……..……..
   0.10000000E+01   0.10000000E+01   0.10000000E+01   0.10000000E+01
   0.10000000E+01   0.10000000E+01   0.10000000E+01   0.10000000E+01
   0.10000000E+01   0.10000000E+01   0.10000000E+01   0.10000000E+01
 'SGL     '         100 '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
 'SGU     '         100 'REAL'
   0.69999999E+00   0.69999999E+00   0.69999999E+00   0.69999999E+00
```

```
   0.69999999E+00   0.85000002E+00   0.85000002E+00   0.85000002E+00
   0.85000002E+00   0.85000002E+00   0.69999999E+00   0.69999999E+00
……..……..
   0.85000002E+00   0.85000002E+00   0.85000002E+00   0.85000002E+00
   0.85000002E+00   0.85000002E+00   0.85000002E+00   0.85000002E+00
   0.85000002E+00   0.85000002E+00   0.85000002E+00   0.85000002E+00
 'KRW     '         100 'REAL'
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
……..……..
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
 'KRG     '         100 'REAL'
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
……..……..
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
 'KRO     '         100 'REAL'
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
……..……..
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
 'SWATINIT'        100 'REAL'
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
……..……..
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
 'KRWR    '         100 'REAL'
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
……..……..
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
 'KRGR    '         100 'REAL'
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
……..……..
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
 'KRORG   '         100 'REAL'
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
……..……..
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
 'KRORW   '         100 'REAL'
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
```

```
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
……..……..
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
 'SWLPC    '          100 'REAL'
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
……..……..
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
 'SGLPC    '          100 'REAL'
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
……..……..
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
 'PCW      '          100 'REAL'
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
……..……..
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
 'PCG      '          100 'REAL'
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
……..……..
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
 'ENDNUM   '          100 'INTE'
           1           1           1           1           1           1
           1           1           1           1           1           1
           1           1           1           1           1           1
……..……..
           1           1           1           1           1           1
           1           1           1           1           1           1
           1           1           1           1
```

End of Example

D.6.8 INIT DATA - LGR GRID DATA RECORD

The file structure for INIT Data - LGR Grid Data Record record is similar to the global grid INIT Data - Global Grid Data Record described previously. This data is immediately written after the first INIT Data - LGR Grid Header Record for each LGR. The record description is outlined in Table D.15.

Note that currently OPM Flow does not support LGR grids and therefore this record format cannot be used by OPM Flow.

No.	Record Name	INIT Data - LGR Grid Data Record (Repeated For Each LGR In The Model)			Status Or Value
		Keyword	No. of Entries	Data Type	
	INITHEAD	LGR grid record for the INIT Data file that defines the start of INIT LGR Grid record and contains the integer variables for this record. This record is the same as the Global INITHEAD record but the data pertains to the current LGR. See Table D.10 for a detailed description of this record type.			Required Only Once
	LOGIHEAD	Global grid record for the INIT Data file that defines the logical variables (T for true and F for false) for this record. This record is the same as the Global LOGIHEAD record but the data pertains to the current LGR. See Table D.10 for a detailed description of this record type.			Required Only Once
	DOUBHEAD	Global grid record for the INIT Data file that defines the double precision REAL variables for this record. This record is the same as the Global DOUBHEAD record but the data pertains to the current LGR. See Table D.10 for a detailed description of this record type.			Required Only Once

Notes:

1) Rows shaded in gray indicate sub-records in this header record type.
2) Item No. cells colored orange indicate that this item is not supported by OPM Flow and those colored red are either "Not Used" or "Undefined".
3) Note all character variables, including the Header Record Name should be enclosed in single quotations, for example the LGR header Record Name should be written out as 'LGR '.

Table D.15: INIT Data - LGR Grid Data Record

Example: INIT Data - LGR Grid Data Record

The following example shows a typical formattedINIT Data - LGR Grid Data Record data set for a single LGR grid named LGR-1.

```
'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'
  0.000000000000000D+00   0.100000000000000D+01   0.365000000000000D+03
  0.100000000149012D+00   0.150000000596046D+00   0.300000000000000D+01
 ..........................................
  0.100000000000000D+01   0.000000000000000D+00   0.000000000000000D+00
  0.100000000000000D+01   0.100000000000000D-03
```

End of Example

D.6.9 INIT DATA - LGR GRID HEADER RECORD

The INIT Data - LGR Grid Header Record defines an LGR's properties, LGR Name for example, and always proceeds an LGR data set for a given LGR; that is the record is repeated for each LGR data set in the model. Note that an LGR data set is terminated by the LGRSGONE record which switches input back to the global grid properties. The record description is outlined in Table D.16.

Note that currently OPM Flow does not support LGR grids and therefore this record format cannot be used by OPM Flow.

No.	Record Name	INIT Data - LGR Grid Header Record (Repeated For Each LGR In The Model)			Status Or Value
		Keyword	No. of Entries	Data Type	
	LGR	LGR grid header record 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 record.			Always Required
1-1	Format	LGRHEADI	45	INTE	
2-1 to 2-45	Data	Undefined.			
	LGRHEADQ	LGR grid record for the INIT Data file that defines the logical variables (T for true and F for false) for this record.			Always Required
1-1	Format	LGRHEADQ	5	LOGI	
2-1 to 2-5	Data	Undefined			
	LGRHEADD	LGR grid record for the INIT Data file that defines the double precision REAL variables for this record.			Always Required
1-1	Format	LGRHEADD	5	DOUB	5
2-1 to 2-5	Data	Undefined.			
	LGRSGONE	The LGRSGONE record 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	

No.	Record Name	INIT Data - LGR Grid Header Record (Repeated For Each LGR In The Model)			Status Or Value
		Keyword	No. of Entries	Data Type	

Notes:

1) Rows shaded in gray indicate sub-records in this header record type.
2) Item No. cells colored orange indicate that this item is not supported by OPM Flow and those colored red are either "Not Used" or "Undefined".
3) Note all character variables, including the Header Record Name should be enclosed in single quotations, for example the LGR header Record Name should be written out as 'LGR '.

Table D.16: INIT Data - LGR Grid Header Record

This record type is repeated for each LGR in the model with the last LGR data set terminated by the LGSGONE record to mark the end of the LGR input; see the following example.

Note

The LGR, LGRHEADI, LGRHEADQ and LGRHEADD always proceed LGR data sets and the LGRSGONE record terminates the end of a data section of LGR data.

Example: INIT Data - LGR Grid Header Record

The following example shows a typical formatted INIT Data Record for LGR Grid Header data set dor 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
'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

D.6.10 INIT DATA - LGR NON-NEIGHBOR CONNECTION DATA RECORD

The INIT Data - LGR Non-Neighbor Connection Data Record defines the LGR Non-Neighbor Connection arrays as generated by OPM Flow. This includes the TRANCC, DIFFNNC, HEATNNC, TRANGL arrays.

Note that currently OPM Flow does not support LGR grids and therefore this record format cannot be used by OPM Flow.

No.	Record Name	INIT Data - LGR Non-Neighbor Connection Data Record (Repeated For Each LGR In The Model)	Status Or Value
I-1	LGR	LGR grid header record (see Table D.16)	Required
	LGRHEADI		
	LGRHEADQ		
	LGRHEADD		
I-2	NNC	LGR NNC data for the current LGR – same format as the Global NCC record (see Table D.11).	Required
I-3	LGRSGONE	The LGRSGONE record marks the end of the LGR section.	Required

Notes:

1) Rows shaded in gray indicate sub-records in this header record type.
2) Item No. cells colored orange indicate that this item is not supported by OPM Flow and those colored red are either "Not Used" or "Undefined".
3) Note all character variables, including the Header Record Name should be enclosed in single quotations, for example the LGR header Record Name should be written out as 'LGR '.

Table D.17: INIT Data - LGR Non-Neighbor Connection Data Record

This record type is repeated for each LGR in the model with the last LGR data set terminated by the LGSGONE record to mark the end of the LGR input; see the following example.

Example: INIT Data - LGR Non-Neighbor Connection Data Record

The following example shows a typical formatted INIT Data - LGR Non-Neighbor Connection Data Record 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
'TRANNNC '           0 'REAL'
'TRANGL  '          16 'REAL'
  0.37566667E+01   0.37566667E+01   0.37566667E+01   0.37566667E+01
  0.37566667E+01   0.37566667E+01   0.37566667E+01   0.37566667E+01
```

```
  0.37566667E+01   0.37566667E+01   0.37566667E+01   0.37566667E+01
  0.37566667E+01   0.37566667E+01   0.37566667E+01   0.37566667E+01
'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
'TRANNNC '           0 'REAL'
'TRANGL  '          16 'REAL'
  0.37566667E+01   0.37566667E+01   0.37566667E+01   0.37566667E+01
  0.37566667E+01   0.37566667E+01   0.37566667E+01   0.37566667E+01
  0.37566667E+01   0.37566667E+01   0.37566667E+01   0.37566667E+01
  0.37566667E+01   0.37566667E+01   0.37566667E+01   0.37566667E+01
'LGRSGONE'           0 'MESS'
```

End of Example

D.6.11 INIT DATA - LGR GRID PROPERTY DATA RECORD

The file structure for INIT Data - LGR Grid Property Data Record is similar to the global grid INIT Data Grids record described in the previous section. However, additional data that the defines the LGR properties (LGR Name for example) are included in this record definition and the LGR record is repeated for each LGR in the model. The record description is outlined in Table D.18.

Note that currently OPM Flow does not support LGR grids and therefore this record format cannot be used by OPM Flow.

No.	Record Name	INIT Data - LGR Grid Property Data Record (Repeated For Each LGR In The Model)	Status Or Value
1-1	LGR	LGR Grid Header Record (see Table D.16)	Required
	LGRHEADI		
	LGRHEADQ		
	LGRHEADD		
1-2	INITHEAD	LGR Grid Data Record (see Table D.15)	
	LOGIHEAD		
	DOUBHEAD		
1-3	PROPERTY	LGR PROPERTY data for the current LGR – same format as the Global PROPERTY record (see TableD.27).	Required
1-4	LGRSGONE	LGR grid header section termination record (see Table D.16)	Required

Notes:

1) Rows shaded in gray indicate sub-records in this header record type.
2) Item No. cells colored orange indicate that this item is not supported by OPM Flow and those colored red are either "Not Used" or "Undefined".
3) Note all character variables, including the Header Record Name should be enclosed in single quotations, for example the LGR header Record Name should be written out as 'LGR '.

Table D.18: INIT Data - LGR Grid Property Data Record

This record type is repeated for each LGR in the model with the last LGR data set terminated by the LGSGONE record to mark the end of the LGR input; see the following example.

Example: INIT Data - LGR Grid Property Data Record

The following example shows a typical formatted INIT Data - LGR Grid Property Data Record 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
'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
'PERMY   '           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
'PERMZ   '           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
'MULTX   '           8 'REAL'
  0.10000000E+01   0.10000000E+01   0.10000000E+01   0.10000000E+01
  0.10000000E+01   0.10000000E+01   0.10000000E+01   0.10000000E+01
'MULTY   '           8 'REAL'
  0.10000000E+01   0.10000000E+01   0.10000000E+01   0.10000000E+01
  0.10000000E+01   0.10000000E+01   0.10000000E+01   0.10000000E+01
'MULTZ   '           8 'REAL'
  0.10000000E+01   0.10000000E+01   0.10000000E+01   0.10000000E+01
  0.63999999E+00   0.63999999E+00   0.63999999E+00   0.63999999E+00
'PORO    '           8 'REAL'
  0.30000001E+00   0.30000001E+00   0.30000001E+00   0.30000001E+00
  0.30000001E+00   0.30000001E+00   0.30000001E+00   0.30000001E+00
'NTG     '           8 'REAL'
  0.10000000E+01   0.10000000E+01   0.10000000E+01   0.10000000E+01
  0.10000000E+01   0.10000000E+01   0.10000000E+01   0.10000000E+01
'TOPS    '           8 'REAL'
  0.83250000E+04   0.83250000E+04   0.83250000E+04   0.83250000E+04
  0.83350000E+04   0.83350000E+04   0.83350000E+04   0.83350000E+04
'DEPTH   '           8 'REAL'
  0.83300000E+04   0.83300000E+04   0.83300000E+04   0.83300000E+04
  0.83400000E+04   0.83400000E+04   0.83400000E+04   0.83400000E+04
'TRANX   '           8 'REAL'
  0.56349998E+01   0.00000000E+00   0.56349998E+01   0.00000000E+00
```

```
  0.56349998E+01   0.00000000E+00   0.56349998E+01   0.00000000E+00
'TRANY   '           8 'REAL'
  0.56349998E+01   0.56349998E+01   0.00000000E+00   0.00000000E+00
  0.56349998E+01   0.56349998E+01   0.00000000E+00   0.00000000E+00
'TRANZ   '           8 'REAL'
  0.14087500E+05   0.14087500E+05   0.14087500E+05   0.14087500E+05
  0.00000000E+00   0.00000000E+00   0.00000000E+00   0.00000000E+00
'MINPVV  '           8 'REAL'
  0.10000000E-05   0.10000000E-05   0.10000000E-05   0.10000000E-05
  0.10000000E-05   0.10000000E-05   0.10000000E-05   0.10000000E-05
'MULTPV  '           8 'REAL'
  0.10000000E+01   0.10000000E+01   0.10000000E+01   0.10000000E+01
  0.10000000E+01   0.10000000E+01   0.10000000E+01   0.10000000E+01
'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
'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           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
'PERMY   '           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
'PERMZ   '           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
'MULTX   '           8 'REAL'
  0.10000000E+01   0.10000000E+01   0.10000000E+01   0.10000000E+01
  0.10000000E+01   0.10000000E+01   0.10000000E+01   0.10000000E+01
'MULTY   '           8 'REAL'
  0.10000000E+01   0.10000000E+01   0.10000000E+01   0.10000000E+01
  0.10000000E+01   0.10000000E+01   0.10000000E+01   0.10000000E+01
'MULTZ   '           8 'REAL'
  0.10000000E+01   0.10000000E+01   0.10000000E+01   0.10000000E+01
  0.63999999E+00   0.63999999E+00   0.63999999E+00   0.63999999E+00
'PORO    '           8 'REAL'
  0.30000001E+00   0.30000001E+00   0.30000001E+00   0.30000001E+00
  0.30000001E+00   0.30000001E+00   0.30000001E+00   0.30000001E+00
'NTG     '           8 'REAL'
  0.10000000E+01   0.10000000E+01   0.10000000E+01   0.10000000E+01
  0.10000000E+01   0.10000000E+01   0.10000000E+01   0.10000000E+01
'TOPS    '           8 'REAL'
  0.83250000E+04   0.83250000E+04   0.83250000E+04   0.83250000E+04
  0.83350000E+04   0.83350000E+04   0.83350000E+04   0.83350000E+04
'DEPTH   '           8 'REAL'
  0.83300000E+04   0.83300000E+04   0.83300000E+04   0.83300000E+04
  0.83400000E+04   0.83400000E+04   0.83400000E+04   0.83400000E+04
'TRANX   '           8 'REAL'
  0.56349998E+01   0.00000000E+00   0.56349998E+01   0.00000000E+00
  0.56349998E+01   0.00000000E+00   0.56349998E+01   0.00000000E+00
'TRANY   '           8 'REAL'
  0.56349998E+01   0.56349998E+01   0.00000000E+00   0.00000000E+00
  0.56349998E+01   0.56349998E+01   0.00000000E+00   0.00000000E+00
'TRANZ   '           8 'REAL'
  0.14087500E+05   0.14087500E+05   0.14087500E+05   0.14087500E+05
  0.00000000E+00   0.00000000E+00   0.00000000E+00   0.00000000E+00
'MINPVV  '           8 'REAL'
  0.10000000E-05   0.10000000E-05   0.10000000E-05   0.10000000E-05
  0.10000000E-05   0.10000000E-05   0.10000000E-05   0.10000000E-05
'MULTPV  '           8 'REAL'
  0.10000000E+01   0.10000000E+01   0.10000000E+01   0.10000000E+01
  0.10000000E+01   0.10000000E+01   0.10000000E+01   0.10000000E+01
'LGRSGONE'           0 'MESS'
```

End of Example

D.6.12 INIT DATA - LGR REGION DATA RECORD

The INIT Data - LGR Region Data Record defines the LGR REGION arrays, The REGION record users the same keywords outlined in the REGION section of the manual in naming the subsequent data arrays. For example, the EQLNUM, FIPNUM, PVTNUM, ROCKNUM, SATNUM arrays etc.

Note that currently OPM Flow does not support LGR grids and therefore this record format cannot be used by OPM Flow.

No.	Record Name	INIT Data - LGR Region Data Record (Repeated For Each LGR In The Model)	Status Or Value
1-1	LGR	LGR grid header record (see Table D.16)	Required
	LGRHEADI		
	LGRHEADQ		
	LGRHEADD		
1-2	REGION	LGR REGION data for the current LGR – same format as the Global REGION record (see Table D.19).	Required
1-3	LGRSGONE	LGR grid header section termination record (see Table D.16)	Required

Notes:

1) Rows shaded in gray indicate sub-records in this header record type.
2) Item No. cells colored orange indicate that this item is not supported by OPM Flow and those colored red are either "Not Used" or "Undefined".
3) Note all character variables, including the Header Record Name should be enclosed in single quotations, for example the LGR header Record Name should be written out as 'LGR '.

Table D.19: INIT Data - LGR Region Data Record

This record type is repeated for each LGR in the model with the last LGR data set terminated by the LGSGONE record to mark the end of the LGR input; see the following example.

Example: INIT Data - LGR Region Data Record

The following example shows a typical formatted INIT Data - LGR Region Data Record 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

D.6.13 INIT DATA - LGR SATURATION AND END-POINT DATA RECORD

The INIT Data - LGR Saturation and End-Point Data Record defines the global grid saturation and end-point arrays. The SATNUM record users the same keywords outlined in the GRID and PROPS section of the manual in naming the subsequent data arrays. For example, the SWL, SWATINIT, KRG, PCW, etc. arrays

Note that currently OPM Flow does not support LGR grids and therefore this record format cannot be used by OPM Flow.

No.	Record Name	INIT Data - LGR Saturation And End-Point Data Record	Status Or Value
I-1	LGR	LGR grid header record (see Table D.16)	Required
	LGRHEADI		
	LGRHEADQ		
	LGRHEADD		
I-2	SATFUNS	LGR SATFUNS data for the current LGR – same format as the Global SATFUNS record (see Table D.20).	Required
I-3	LGRSGONE	LGR grid header section termination record (see Table D.16)	Required

Notes:

1) Rows shaded in gray indicate sub-records in this header record type.
2) Item No. cells colored orange indicate that this item is not supported by OPM Flow and those colored red are either "Not Used" or "Undefined".
3) Note all character variables, including the Header Record Name should be enclosed in single quotations, for example the EQLNUM should be written out as 'EQLNUM '.

Table D.20: INIT Data - LGR Saturation and End-Point Data Record

This record type is repeated for each LGR in the model with the last LGR data set terminated by the LGSGONE record to mark the end of the LGR input; see the following example.

Example: INIT Data - LGR Saturation and End-Point Data Record

The following example shows a typical formatted INIT Data - LGR Saturation and End-Point Data Record.

```
'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
'SWCR    '          8 'REAL'
  0.13300000E+00   0.22200000E+00   0.22200000E+00   0.22200000E+00
  0.11100000E+00   0.11100000E+00   0.50000000E+00   0.11100000E+00
'SGCR    '          8 'REAL'
  0.22200000E+00   0.22200000E+00   0.22200000E+00   0.22200000E+00
  0.22200000E+00   0.22200000E+00   0.22200000E+00   0.22200000E+00
'SOWCR   '          8 'REAL'
  0.15000001E+00   0.15000001E+00   0.15000001E+00   0.15000001E+00
  0.15000001E+00   0.15000001E+00   0.15000001E+00   0.15000001E+00
'SOGCR   '          8 'REAL'
  0.15000001E+00   0.15000001E+00   0.15000001E+00   0.15000001E+00
  0.15000001E+00   0.15000001E+00   0.15000001E+00   0.15000001E+00
'SWL     '          8 'REAL'
  0.13300000E+00   0.22200000E+00   0.22200000E+00   0.22200000E+00
  0.11100000E+00   0.11100000E+00   0.50000000E+00   0.10101000E+00
'SWU     '          8 'REAL'
  0.10000000E+01   0.10000000E+01   0.10000000E+01   0.10000000E+01
  0.10000000E+01   0.10000000E+01   0.10000000E+01   0.10000000E+01
'SGL     '          8 '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
'SGU     '          8 'REAL'
  0.69999999E+00   0.69999999E+00   0.69999999E+00   0.69999999E+00
  0.69999999E+00   0.69999999E+00   0.69999999E+00   0.69999999E+00
'KRW     '          8 'REAL'
 -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
 -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
'KRG     '          8 'REAL'
 -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
 -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
'KRO     '          8 'REAL'
 -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
 -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
'SWATINIT'          8 'REAL'
 -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
 -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
'KRWR    '          8 'REAL'
 -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
 -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
'KRGR    '          8 'REAL'
 -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
 -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
'KRORG   '          8 'REAL'
 -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
```

```
 -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
'KRORW   '           8 'REAL'
 -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
 -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
'SWLPC   '           8 'REAL'
 -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
 -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
'SGLPC   '           8 'REAL'
 -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
 -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
'PCW     '           8 'REAL'
 -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
 -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
'PCG     '           8 'REAL'
 -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
 -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
'ENDNUM  '           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       -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
'SWCR    '           8 'REAL'
  0.13300000E+00   0.22300000E+00   0.22300000E+00   0.22300000E+00
  0.11300000E+00   0.11300000E+00   0.93000001E+00   0.11300000E+00
'SGCR    '           8 'REAL'
  0.22300000E+00   0.22300000E+00   0.22300000E+00   0.22300000E+00
  0.22300000E+00   0.22300000E+00   0.22300000E+00   0.22300000E+00
'SOWCR   '           8 'REAL'
  0.18000001E+00   0.18000001E+00   0.18000001E+00   0.18000001E+00
  0.18000001E+00   0.18000001E+00   0.18000001E+00   0.18000001E+00
'SOGCR   '           8 'REAL'
  0.18000001E+00   0.18000001E+00   0.18000001E+00   0.18000001E+00
  0.18000001E+00   0.18000001E+00   0.18000001E+00   0.18000001E+00
'SWL     '           8 'REAL'
  0.13300000E+00   0.22300000E+00   0.22300000E+00   0.22300000E+00
  0.11300000E+00   0.11300000E+00   0.93000001E+00   0.10103000E+00
'SWU     '           8 'REAL'
  0.10000000E+01   0.10000000E+01   0.10000000E+01   0.10000000E+01
  0.10000000E+01   0.10000000E+01   0.10000000E+01   0.10000000E+01
'SGL     '           8 '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
'SGU     '           8 'REAL'
  0.85000002E+00   0.85000002E+00   0.85000002E+00   0.85000002E+00
  0.85000002E+00   0.85000002E+00   0.85000002E+00   0.85000002E+00
'KRW     '           8 'REAL'
 -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
 -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
'KRG     '           8 'REAL'
```

```
 -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
 -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
'KRO     '           8 'REAL'
 -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
 -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
'SWATINIT'           8 'REAL'
 -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
 -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
'KRWR    '           8 'REAL'
 -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
 -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
'KRGR    '           8 'REAL'
 -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
 -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
'KRORG   '           8 'REAL'
 -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
 -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
'KRORW   '           8 'REAL'
 -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
 -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
'SWLPC   '           8 'REAL'
 -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
 -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
'SGLPC   '           8 'REAL'
 -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
 -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
'PCW     '           8 'REAL'
 -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
 -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
'PCG     '           8 'REAL'
 -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
 -0.10000000E+21  -0.10000000E+21  -0.10000000E+21  -0.10000000E+21
'ENDNUM  '           8 'INTE'
          1           1           1           1           1           1
          1           1
'LGRSGONE'           0 'MESS'
```

D.6.14 INIT DATA - TABULAR DATA RECORD

To be written in a future release of the manual.

D.7 RESTART FILES – SOLUTION DATA

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[161]. 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.

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 *2.2Running OPM Flow 2019-10 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 following sections outline the format for these two file types.

[161] *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.*

D.7.1 RESTART INDEX FILES

To be written in a future release of the manual.

D.7.2 RESTART DATA FILE SPECIFICATION

This file type contains the global and LGR grid solution data property data[162], for example porosity (PRESSURE, SGAS, SOIL and SWAT) and group, well and connection data[163]. The overall structure of this file is similar to the INIT file, and the individual data records are structured in the following manner:

Reference Section	Global Records	Global Data Records	LGR	LGR Records	LGR Data Records	Notes
D.7.3	SEQNUM					Global grid header record for that defines the start of a RESTART Global Grid Record time step data set.
D.7.3	INTEHEAD LOGIHEAD DOUBHEAD					Global grid header record and defines the integer, logical and double precision variables for this header record.
D.7.4	IGRP SGRP etc.					Group, well, and connection data status records for this reporting time step.
D.7.5	IAAQ SAAQ et.					Aquifer definition arrays.
D.7.6	HIDDEN					Defines HIDDEN solution data, that is a list of solution arrays NOT to be read by post processing software.
D.7.7	STARTSOL					This record marks the start of the solution variable section for the global grid.
D.7.7	SOLUTION	Solution 1 Solution 2 etc.				SOLUTION array section (PRESSURE, SGAS, SOILS, WAT, etc.) for the Global grid.
D.7.7	LGRNAMES					A list of LGRs in this report step
D.7.7	ENDSOL					Marks the end of the solution variable section for the global grid.
D.6.8			LGR-1	LGR LGRHEADI LGRHEADQ LGRHEADD		LGR grid header record for that defines the start of a RESTART Global Grid Record time step data set.

[162] *Several record types are used for both global and LGR grids, for example the INTEHEAD, LOGIHEAD, DOUBHEAD and SOLUTION arrays.*

[163] *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 Records	Global Data Records	LGR	LGR Records	LGR Data Records	Notes
D.7.3				INTEHEAD LOGIHEAD DOUBHEAD		LGR header record and defines the integer, logical and double precision variables for this header record.
D.7.4				IGRP SGRP etc.		Group, well, and connection data status records for this reporting time step.
D.7.6				HIDDEN		Defines HIDDEN solution data, that is a list of solution arrays NOT to be read by post processing software.
D.7.7				STARTSOL		Start of SOLUTION array section.
D.7.7				SOLUTION		Solution arrays.
					Solution 1 Solution 2 etc.	SOLUTION array section (PRESSURE, SGAS, SOILS, WAT, etc.) for the current LGR.
D.7.7				ENDSOL		End of the solution variable section for the current LGR grid.
D.6.8				ENDLGR		End of current LGR section.
D.6.8			LGR-2	LGR LGRHEADI LGRHEADQ LGRHEADD		LGR grid header record for that defines the start of a RESTART Global Grid Record time step data set.
D.7.4				IGRP SGRP etc.		Group, well, and connection data status records for this reporting time step.
D.7.6				HIDDEN		Defines HIDDEN solution data, that is a list of solution arrays NOT to be read by post processing software.
D.7.7				STARTSOL		Start of SOLUTION array section.
D.7.7				SOLUTION		Solution arrays.
					Solution 1 Solution 2 etc.	SOLUTION array section (PRESSURE, SGAS, SOILS, WAT, etc.) for the current LGR.
D.7.7				ENDSOL		End of the solution variable section for the current LGR grid.
D.6.8				ENDLGR		End of current LGR section.

Reference Section	Global Records	Global Data Records	LGR	LGR Records	LGR Data Records	Notes
D.7.3	SEQNUM					Global grid header record for that defines the start of a RESTART Global Grid Record time step data set.

Table D.21: RESTART Data File Structure

The individual records section are described in the following sections.

D.7.3 RESTART DATA - HEADER RECORD

The Restart Data - Header Record[164] is the first data set that should be read or written to the RESTART file. This data set includes INITHEAD, LOGIHEAD and DOUBHEAD records that define versus parameters used in subsequent records in the RESTART file for the global and LGR grids. This is then followed by a series of global:

1) Well, group and connection records for the grid.
2) Grid SOLUTION data records that defined the various grid solution arrays for each grid cell in the model.
3) Grid NNC solution arrays.
4) List of LGRs
5) LGR Data for each LGR.

The structure for this type of record is defined in Table D.23.

No.	Record Name	RESTART Data - Header Record			Status Or Value
		Keyword	No. of Entries	Data Type	
	SEQNUM	Global grid header record for the RESTART Data file that defines the start of a RESTART Global Grid Record time step report data set for unified restart files only. Defines the report time step number.			Optional
1-1	Format	SEQNUM	1	INTE	
2-1	Data	Report time step number.			0
	INITHEAD	Global grid header record for the RESTART Data file that defines the start of a RESTART Global Grid Record for non-unified multiple files, and defines the integer variables for this header record.			Required
1-1	Format	INITHEAD	249	INTE	
2-1	Data	Undefined			0
2-2	Data	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	Data	Undefined.			0
2-9	Data	The number of grid blocks in the x-direction (NX).			
2-10	Data	The number of grid blocks in the y-direction (NY).			
2-11	Data	The number of grid blocks in the z-direction (NZ).			
2-12	Data	Number of global active cells in the model			
2-13	Data	Not used.			0
2-14	Data	Not used.			0

[164] *The RESTART Data – Header Record format is 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 head records and terminated by an LGR termination record.*

No.	Record Name	RESTART Data - Header Record			Status Or Value
		Keyword	No. of Entries	Data Type	
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.			
2-16	Data	Undefined.			0
2-17	Data	Number of wells in the model maximum number of grid block connections per well for this model.			NWELLS
2-18	Data	Maximum number of grid block connections per well.			MXCONS
2-19	Data	Not used.			
2-20	Data	Maximum number of wells belonging to a group in the model.			MXGRPW
2-21	Data	Maximum number of groups for this mode.			MXGRPS
2-22 to 2-24	Data	Not used.			
2-25	Data	Number of values per well in the IWEL array.			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 values per well in the ZWEL array.			NZWELZ
2-29 to 2-32	Data	Not used.			
2-33	Data	Number of values per well in the ICON array.			NICONZ
2-34	Data	Number of values per well in the SCON array.			NSCONZ
2-35	Data	Number of values per well in the XCON array.			NXCONZ
2-36	Data	Not used.			0
2-37	Data	Number of values per well in the IGRP array.			NIGRPZ
2-38	Data	Number of values per well in the SGRP array.			NISGRPZ
2-39	Data	Number of values per well in the XGRP array.			NIXGRPZ
2-40	Data	Number of values per well in the ZGRP array.			NIZGRPZ
2-41	Data	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

No.	Record Name	RESTART Data - Header Record			Status Or Value
		Keyword	No. of Entries	Data Type	
2-49 to 2-65	Data	Not used.			
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
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 2019.			YEAR
2-68 to 2-94	Data	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.			
2-96 to 2-131	Data	Undefined.			0
2-132	Data	Maximum number of nodes in an extended network model or zero if the extended network option has not been activated.			NODMAX
2-133	Data	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
2-139	Data	Number of items in the INOBR array.			NINOBR

No.	Record Name	RESTART Data - Header Record			Status Or Value
		Keyword	No. of Entries	Data Type	
2-140 to 2-162	Data	Undefined.			0
2-163	Data	Maximum number of used aquifer connections.			NGCAUS
2-164 to 2-175	Data	Undefined.			0
2-176	Data	Maximum number of multi-segmented wells in the model, for when multi-segment wells have been activated, or zero otherwise.			NSWLMX
2-177	Data	Maximum number of segments per multi-segment well, for when multi-segment wells have been activated, or zero otherwise.			NSEGMX
2-178	Data	Maximum number of branches per multi-segment well, including the main branch. groups for the model, for when multi-segment wells have been activated, or zero otherwise.			NLBRMZ
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.			NRSEGZ
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	Data	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.			HH
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.			MM
2-209 to 2-223	Data	Undefined.			0
2-224	Data	Number of integer lines in the ACTNUM data set.			NILAQN
2-225	Data	Number of integer lines in the ACTNUM data set.			NIRAQN
2-226	Data	Undefined.			0
2-227	Data	Number of entries in the ACTNUM data set.			NUMAQN
2-228 to 2-234	Data	Undefined.			0
2-235	Data	Number of items in the ICOT array.			NICOTZ

No.	Record Name	RESTART Data - Header Record			Status Or Value
		Keyword	No. of Entries	Data Type	
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 items in the model plus 2.			NSTRA2
2-242 to 2-252	Data	Undefined.			0
2-253	Data	Maximum umber of analytical aquifers in the model.			MAAQID
2.254 to 2-270	Data	Undefined.			0
2.271	Data	Maximum number of segment links per multi-segment well for when multi-segment wells have been activated, or zero otherwise.			
2-272 to 2-410	Data	Undefined.			0
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.			SS
2-412 to 249	Data	Undefined.			0
	LOGIHEAD	Global grid record for the RESTART Data file that defines the logical variables (T for true and F for false) for this record.			Required
1-1	Format	LOGIHEAD	79	LOGI	
2-1 to 2-3	Data	Not used.			F
2-4	Data	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.			F
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.			F

No.	Record Name	RESTART Data - Header Record			Status Or Value
		Keyword	No. of Entries	Data Type	
2.6 to 2-14	Data	Undefined.			0
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.			F
2-16 to 2-30	Data	Undefined.			0
2-31	Data	Coal Bed Methane option has been activated in the composition model indicator, set to T or F. Note that currently OPM Flow does not support the Coal Be Methane option and therefore this item should be set to F.			F
2-32 to 2-79	Data	Undefined.			0
	DOUBHEAD	Global grid record for the RESTART Data file that defines the double precision REAL variables for this record.			Required
1-1	Format	DOUBHEAD	185	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 to 2-160	Data	Undefined.			0
2-160	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 data elements (2-1) and (2-161).			
2-163 to 2-185	Data	Undefined.			0

Table D.22: RESTART Data - Header Record

Example: RESTART Data - Header Record

The following example shows a typical formatted RESTART Data - Header Record data set.

```
'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

D.7.4 RESTART DATA – GROUP, WELL AND CONNECTION DATA RECORDS

The Restart Data – Well and Group Completion Record[165] defines various parameters associated with groups, wells and connection records in the RESTART file 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:

1) Well, group and connection records for the grid.
2) Grid SOLUTION data records that defined the various grid solution arrays for each grid cell in the model.
3) GridINNC solution arrays.

The structure for this type of record is defined in Table D.23.

No.	Record Name	RESTART Data – Group, Well And Connection Records			Status Or Value
		Keyword	No. of Entries	Data Type	
	IGRP	RESTART Data file record that defines the integer Group, well, and connection data for this record.			Required
1-1	Format	IGRP	NSGRPZ x NGMAXZ	DOUB	
2-1	Data	Integer group data array IGRP(NIGRPZ,NGMAXZ) where NIGRPZ and NGMAXZ are defined on the INTEHEAD record. 1) 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. 2) Entries (NWGMAX + 1) define the number of wells or child groups belonging to this group. 3) Entries (NWGMAX + 27) define the group type: with 0 for a well group, 1 for a node group, 2 for a satellite group, and 3 for a slave group. 4) Entries (NWGMAX + 28) define the level of the group, with 0 representing the Field group. 5) Entries (NWGMAX + 29) define the index of the parent group Note that undefined items in this array may be set to zero.			
	SGRP	RESTART Data file record that defines the real GROUP data for this record this record.			Required
1-1	Format	SGRP	NSGRPZ x NGMAXZ	REAL	
2-1	Data	Real group data array SGRP(NIGRPZ,NGMAXZ) where NIGRPZ and NGMAXZ are defined on the INTEHEAD record.			
	XGRP	RESTART Data file record that defines the double precision GROUP data for this record.			Required
1-1	Format	XGRP	NSGRPZ x NGMAXZ	DOUB	
2-1	Data	Double precision group data array XGRP(NIGRPZ,NGMAXZ) where NIGRPZ and NGMAXZ are defined 0n the INTEHEAD record.			
	ZGRP	RESTART Data file record that defines the character GROUP data for this record.			Required

[165] *The RESTART Data – Group, Well and Connection Record format is used both for global and LGR grids with groups, well and connection dats 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 head records and terminated by an LGR termination record.*

No.	Record Name	RESTART Data – Group, Well And Connection Records			Status Or Value
		Keyword	No. of Entries	Data Type	
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 record.			
	ISEG	Integer multi-segment well data array ISEG (NISEGZ, NSEGMX, NSWLMX) where NISEGZ, NSEGMX, and NSWLMX are defined on the INTEHEAD record.			Optional
1-1	Format	ISEG	NISEGZ x NSEGMX x NSWLMX	INTE	
2-1	Data	Undefined.			
2-2	Data	Defines the multi-segment outlet segment number starting with 0 for the segment nearest wellhead (NISEGZ = 2).			
2-3	Data	Undefined.			
2-4	Data	Defines the multi-segment branch for this segment number 1 for main stem and 0 if not active (NISEGZ = 4).			
	RSEG	Double precision multi-segment well data array RSEG (NISEGZ, NSEGMX, NSWLMX) where NISEGZ, NSEGMX, and NSWLMX are defined on the INTEHEAD record.			Optional
1-1	Format	RSEG	NISEGZ x NSEGMX x NSWLMX	DOUB	
2-1	Data	RSEG (NISEGZ, NSEGMX, NSWLMX) array			
	ILBS	Integer multi-segment well data array for lateral branches ILBR (NLBRMX,NSWLMX) where NLBRMX and NSWLMX are defined on the INTEHEAD record.			Optional
1-1	Format	ILBS	NLBRMX x NSWLMX	INTE	
2-1	Data	ILBR (NLBRMX,NSWLMX) array.			
	ILBR	Integer multi-segment well data array for chords ILBR (NILBRZ, NLBRMX, NSWLMX) where NILBRZ, NLBRMX, and NSWLMX are defined on the INTEHEAD record.			Optional
1-1	Format	ILBR	NILBRZ x NLBRMX x NSWLMX	INTE	
2-1	Data	ILBR (NILBRZ, NLBRMX, NSWLMX) array.			
	ICRD	Integer multi-segment well data array for chords ICRD (2, NCRDMX, NSWLMX) where NCRDMX, and NSWLMX are defined on the INTEHEAD record.			Optional
1-1	Format	ILBR	2 x NCRDMX x NSWLMX	INTE	
2-1	Data	ICRD (2, NCRDMX, NSWLMX) array.			
	IWEL	Well integer data array IWEL(NIWELZ, NWELLS) where NIWELX and NWELS are defined on the INTEHEAD record.			
1-1	Format	IWEL	NIWELZ*NWELLS	INTE	Required

No.	Record Name	RESTART Data – Group, Well And Connection Records			Status Or Value
		Keyword	No. of Entries	Data Type	
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	Defines the location of the wellhead in the grid for the 4 dimension. (NIWELZ = 3)			
2-4	Data	Undefined.			
2-5	Data	Defines the number of well connections in the grid (NIWELZ = 5).			
2-6	Data	Defines the group index that the well belongs to (NIWELZ = 6).			
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 to 2-10	Data	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 to 42	Data	Undefined.			
2-43	Data	Defines the LGR index for a well with local completions in a LGR (NIWELZ = 43).			
2-44 to 2-48	Data	Undefined.			
2-49	Data	Defines the well friction indicator, set to non-zero for horizontal wells (NIWELZ = 49).			
2-50 to 2-70	Data	Undefined.			
2-71	Data	Defines the segmented well number, set to zero for ordinary wells (NIWELZ = 71).			
	SWEL	Well real data array SWEL(NSWELZ, NWELLS) where NSWELX and NWELS are defined on the INTEHEAD record.			
1-1	Format	SWEL	NSWELZ*NWELLS	REAL	Required
2-1	Data	SWEL(NSWELZ, NWELLS) array.			
	XWEL	Well double precision data array XWEL(NXWELZ, NWELLS) where NXWELX and NWELS defined on the INTEHEAD record.			
1-1	Format	XWEL	NSWELZ*NWELLS	DOUB	Required

No.	Record Name	RESTART Data – Group, Well And Connection Records			Status Or Value
		Keyword	No. of Entries	Data Type	
2-1	Data	XWEL(NXWELZ, NWELLS) array.			
	ZWEL	Well character data array ZWEL(NXWELZ, NWELLS) where NXWELX and NWELS are defined on the INTEHEAD record.			
1-1	Format	ZWEL	NSWELZ*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).			
2-3	Data	Defines the end of time step action for the well (NXWELZ = 3).			
	ICON	Connection integer data array ICON(NICONZ, NCWMAX, NWELLS) where NICONZ, NCWMAX, and NWELLS are defined on the INTEHEAD record. Data records are required for each completion in the well and the number of data items, N, should be NICONZ x NCWMAX x NWELLS.			
1-1	Format	ICON	N	INTE	
2-1	Data	Defines the well connect index (NICONZ = 1).			
2-2	Data	Defines the location of connection in the grid for the I dimension (NICONZ = 2).			
2-3	Data	Defines the location of connection in the grid for the J dimension (NICONZ = 3).			
2-4	Data	Defines the location of connection in the grid for the K dimension (NICONZ = 4).			
2-5	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).			
2-6 2-13	Data	Undefined.			
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).			
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 record. Data records 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 factor NSCONZ = 1 in the SCON array.			
2-2 to 2-3	Data	Undefined.			

No.	Record Name	RESTART Data – Group, Well And Connection Records			Status Or Value
		Keyword	No. of Entries	Data Type	
2-4	Data	Connection KH, that is the net thickness times the permeability, NSCONZ = 4 in the SCON array.			
	XCON	Connection double precision data array XCON(NXCONZ, NCWMAX, NWELLS) where NXCONZ, NCWMAX, and NWELLS are defined on the INTEHEAD record. The number of data items, N, should be NXCONZ x NCWMAX x NWELLS			Required
1-1	Format	ICON	N	DOUB	
2-1	Data	XCON(NXCONZ, NCWMAX, NWELLS) array.			
	ICOT	Tracer connection integer data array ICOT(NICOTZ, NSTRA2, NCWMAX, NWELLS) where NICOTZ, NSTRA2, NCWMAX, and NWELLS are defined on the INTEHEAD record. 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 record. 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.			
	IWET	Tracer well integer data array IWET(NIWETZ, NSTRA2, NWMAXZ, NWELLS) where NIWETZ, NSTRA2, and NWMAXZ are defined on the INTEHEAD record. The number of records, 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 record. The number of records, 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 record. The number of records, 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 record. The number of records, N, should be NXGRTNZ x NSTRA2 x NWMAXZ.			Optional
1-1	Format	XGRT	N	DOUB	

No.	Record Name	RESTART Data – Group, Well And Connection Records			Status Or Value
		Keyword	No. of Entries	Data Type	
2-1	Data	XWET(NXWETZ, NSTRA2, NWMAXZ) array.			
	INODE	Network node integer data array INODE(NINODE, NODMAX) where NINODE and NODMAX are defined in the INTEHEAD record.			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 record.			Optional
1-1	Format	IBRAN	NIBRAN x NBRMAX	INTE	
2-1	Data	IBRAN(NIBRAN, NBRMAX)			
	INOBR	Network node-branch integer data array INOBR(NIOBR) where NIOBR are defined in the INTEHEAD record.			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 record.			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 record.			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 record.			Optional
1-1	Format	ZNODE	NZNODE x NODMAX	CHAR	
2-1	Data	ZNODE(NZNODE, NODMAX) array.			

Notes:

1) Rows shaded in gray indicate sub-records in this header record type.
2) Item No. cells colored orange indicate that this item is not supported by OPM Flow and those colored red are either "Not Used" or "Undefined".
3) Note all character variables, including the Record Name should be enclosed in single quotations, for example the ZNODE record should be written out as 'ZNODE '.

Table D.23: RESTART Data – Group, Well and Connection Records

Example: RESTART Data – Group, Well and Connection Records

The following example shows a typical formatted RESTART Data – Group, Well and Connection Record data set, showing the IGRP, SGRP, XGRP and ZGRP records.

```
 '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-B2 ' '        ' '        '
'        ' '        ' 'MANI-B1 ' '        ' '        ' '        ' '        '
'MANI-D2 ' '        ' '        ' '        ' '        ' 'MANI-E1 ' '        '
'        ' '        ' '        ' 'MANI-E2 ' '        ' '        ' '        '
'        ' 'MANI-K1 ' '        ' '        ' '        ' '        ' 'MANI-K2 '
'        ' '        ' '        ' '        ' 'MANI-F  ' '        ' '        '
'        ' '        ' 'WI-GSEG ' '        ' '        ' '        ' '        '
'D2-DUMMY' '        ' '        ' '        ' '        ' 'FIELD   ' '        '
'        ' '        ' '        '
```

End of Example

D.7.5 RESTART DATA – AQUIFER DATA RECORDS

The RESTART Data – Aquifer Data Records defines the various aquifer arrays written out by OPM Flow. If there are no aquifers in the model then this set of records are skipped. If an analytical or numerical model aquifer is present in the model, a complete set of records, 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 records should be written to the file.

The structure for this type of record is dependent on the aquifer type, for analytical aquifers the format is outlined in Table D.24 and for numerical aquifers in characterized in Table D.25.

No.	Record Name	RESTART Data – Aquifer Data Records Analytical Aquifer Records			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 record.			Optional
1-1	Format	IAAQ	NIAAQZ x MAAQID	INTE	
2-1	Data	IAAQ(NIAAQZ, MAAQID) array.			
	SAAQ	Analytical aquifers real data array SAAQ(NSAAQZ, MAAQID) where NSAAQZ and MAAQIDR are defined in the INTEHEAD record.			Optional
1-1	Format	SAAQ	NSAAQZ x MAAQID	REAL	
2-1	Data	SAAQ(SIAAQZ, MAAQID) array.			
	XAAQ	Analytical aquifers double precision data array XAAQ(NXAAQZ, MAAQID) where NXAAQZ and MAAQIDR are defined in the INTEHEAD record.			Optional
1-1	Format	XAAQ	NXAAQZ x MAAQID	DOUB	
2-1	Data	XAAQ(NXAAQZ, MAAQID) array.			
	IAQL	Analytical aquifers integer aquifer list data array IAQL(NIAQLX, MXNALI, MXAAQL) where NIAQLX, MXNALI, and MXAAQL are defined in the INTEHEAD record.			Optional
1-1	Format	IAQL	NIAQLX x MXNALI x MXAAQL	INTE	
2-1	Data	IAQL(NIAQLX, MXNALI, MXAAQL) array.			
	ZAQL	Analytical aquifers integer aquifer list data array ZAQL(NZAQLX, MXNALI) where NZAQLX, and MXNALI are defined in the INTEHEAD record.			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 record.			Optional
1-1	Format	ICAQ	NICAQZ x NGCAUS x MAAQID	INTE	
2-1	Data	ICAQ(NICAQZ, NGCAUS, MAAQID) array.			

No.	Record Name	RESTART Data – Aquifer Data Records Analytical Aquifer Records			Status Or Value
		Keyword	No. of Entries	Data Type	
	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 record.			Optional
1-1	Format	SCAQ	NICAQZ x NGCAUS x MAAQID	REAL	
2-1	Data	SCAQ(NSCAQZ, NGCAUS, MAAQID) array.			
	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 record.			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 sub-records in this header record type.
2) Note all character variables, including the Header Record Name should be enclosed in single quotations, for example the ACAQ record should be written out as 'ACAQ '.

Table D.24: RESTART Data – Analytical Aquifer Data Records

And for numerical aquifers the format is outlined in Table D.25.

No.	Record Name	RESTART Data – Aquifer Data Records Numerical Aquifer Records			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 record.			Optional
1-1	Format	IAAQ	NIIAQN x NUMAQN	INTE	
2-1	Data	IAQN (NIIAQN, NUMAQN) array.			
	RAQN	Numerical aquifers doubl eprecision data array RAQN (NIRAQN, NUMAQN) where NIIAQN and NUMAQN are defined in the INTEHEAD record.			Optional
1-1	Format	RAAQ	NIRAQN x NUMAQN	REAL	
2-1	Data	RAQN (NIIAQN, NUMAQN) array.			

Notes:

1) Rows shaded in gray indicate sub-records in this header record type.
2) Note all character variables, including the Header Record Name should be enclosed in single quotations, for example the RAQN record should be written out as 'RAQN '.

Table D.25: RESTART Data – Numerical Aquifer Data Records

Example: RESTART Data – Aquifer Data Records

The following example shows a typical formatted RESTART Data – Aquifer Data Record data set for an analtytical 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

D.7.6 RESTART DATA – HIDDEN RECORD

The Restart Data – Hidden Record [166] defines SOLUTION arrays that have not been exported by OPM Flow,

The structure for this type of record is defined in Table D.26.

No.	Record Name	RESTART Data – Hidden Record			Status Or Value
		Keyword	No. of Entries	Data Type	
	HIDDEN	RESTART Data – Hidden Record 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 charaters 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:

3) Note all character variables, including the Record Name should be enclosed in single quotations, for example the SOMAX record should be written out as 'SOMAX '.

Table D.26: RESTART Data – Hidden Record

Example: RESTART Data – Group, Well and Connection Records

The following example shows a typical formatted RESTART Data – Hidden Record 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

[166] *The RESTART Data – Hidden Record format is used both for global and LGR grids.*

D.7.7 RESTART DATA - SOLUTION DATA RECORD

The Restart Data – Solution Data Record[167] defines 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 record is defined in TableD.27 and a list of SOLUTION names is tabulated in

No.	Record Name	RESTART Data - Solution Data Record			Status Or Value
		Keyword	No. of Entries	Data Type	
	STARTSOL	This record marks the start of the solution variable section for both global and local grids. Note that there is no data associated with this record.			Required
1-1	Format	STARTSOL	0	MESG	
	LGRNAMES	This record 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 quotations and each having a length of eight characters.			
	SOLUTION	The SOLUTION record defines the start of a SOLUTION array data set, where SOLUTION is the name of the array being written out, as outlined in Table D.29. For example, PRESSURE, SWAT, SGAS, RS, RV etc. The SOLUTION record (items 1-1, 1-2, 1-3, 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 record marks the end of the solution variable section for both global and local grids. Note that there is no data associated with this record.			Required
1-1	Format	ENDSOL	0	MESG	

Notes:

1) Rows shaded in gray indicate sub-records in this header record type.
2) Note all character variables, including the Record Name should be enclosed in single quotations, for example the SOLUTION array for PRESSURE should be written out as 'PRESSURE '.

TableD.27: RESTART Data - Solution Data Record

For SOLUTION tracer concentration arrays the ZTRACER keyword must precede the SOLUTION tracer data array – see Table D.28 for a description of ZTRACER record.

No.	Record Name	RESTART Data – Solution Data Record For Tracer Concentration Name			Status Or Value
		Keyword	No. of Entries	Data Type	
	ZTRACER	This record marks the start of the solution variable section for both global and local grids.			Optional

[167] *The RESTART Data - Solution Data Record format is used both for global and LGR grids SOLUTION arrrays. The LGR data is preceded by a series of LGR head records and terminated by an LGR termination record.*

No.	Record Name	RESTART Data – Solution Data Record For Tracer Concentration Name			Status Or Value
		Keyword	No. of Entries	Data Type	
1-1	Format	ZTRACER	2	CHAR	
2-1	Data Format	Tracer name (TRACER)			
		TRACER	' '		
	TRACER	TRACER is the name of the tracer concentration that is contained on the next record. 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) Note all character variables, including the Record Name should be enclosed in single quotations, including the tracer name.

Table D.28: RESTART Data – Solution Data Record for Tracer Concentration Name

The SOLUTION record keywords are listed in Table D.29.

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.	
		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.	

Model Type		Model Option	Solution Data Keywords	Solution Variable Description	OPM Output Status
Black-Oil	Compositional				
		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.	
		STANDARD	RFIPWAT	Water fluid-in-place at reservoir conditions.	

Model Type		Model Option	Solution Data Keywords	Solution Variable Description	OPM Output Status
Black-Oil	Compositional				
		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.	
		ALKALINE	ALKALINE	Alkaline concentration.	
		ALKALINE	ALKMAX	Alkaline maximum historic concentration.	

Model Type		Model Option	Solution Data Keywords	Solution Variable Description	OPM Output Status
Black-Oil	Compositional				
		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.	
		FOAM	FOAM	Foam concentration.	
		FOAM	FOAM_HL	Foam half-life.	
		FOAM	FOAMADS	Foam adsorption.	

Model Type		Model Option	Solution Data Keywords	Solution Variable Description	OPM Output Status
Black-Oil	Compositional				
		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.	
		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.	

Model Type		Model Option	Solution Data Keywords	Solution Variable Description	OPM Output Status
Black-Oil	Compositional				
		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 D.29: RESTART Data - Solution Data Keywords

Example: RESTART Data – Solution Data Record

The following example shows a typical formatted global grid RESTART Data – Solution Data Record.

```
 '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 'DOUBHEAD'
……..……..
 '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
 '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
……..……..
etc.
```

End of Example

D.7.8 RESTART DATA - LGR GRID HEADER RECORD

The RESTART Data - LGR Grid Header Record defines an LGR's properties, LGR Name for example, and always proceeds an LGR data set for a given LGR; that is the record is repeated for each LGR data set in the model. Note that a given LGR data set is terminated by the ENDLGR record and the SEQNUM records terminate the input for a time step which switches the input back to the global grid properties. The record description is outlined in Table D.30.

Note that currently OPM Flow does not support LGR grids and therefore this record format cannot be used by OPM Flow.

No.	Record Name	RESTART Data - LGR Grid Header Record (Repeated For Each LGR In The Model)			Status Or Value
		Keyword	No. Of Entries	Data Type	
	LGR	LGR grid header record for the RESTART 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 record.			Always Required
1-1	Format	LGRHEADI	45	INTE	
2-1 to 2-45	Data	Undefined.			
	LGRHEADQ	LGR grid record for the RESTART Data file that defines the logical variables (T for true and F for false) for this record.			Always Required
1-1	Format	LGRHEADQ	5	LOGI	
2-1 to 2-5	Data	Undefined			
	LGRHEADD	LGR grid record for the RESTART Data file that defines the double precision REAL variables for this record.			Always Required
1-1	Format	LGRHEADD	5	DOUB	5
2-1 to 2-5	Data	Undefined.			
	LGRSGONE	The ENDLGR record 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	ENDLGR	0	MESG	

No.	Record Name	RESTART Data - LGR Grid Header Record (Repeated For Each LGR In The Model)			Status Or Value
		Keyword	No. Of Entries	Data Type	

Notes:

1) Rows shaded in gray indicate sub-records in this header record type.
2) Item No. cells colored orange indicate that this item is not supported by OPM Flow and those colored red are either "Not Used" or "Undefined".
3) Note all character variables, including the Header Record Name should be enclosed in single quotations, for example the LGR header Record Name should be written out as 'LGR '.

Table D.30: RESTART Data - LGR Grid Header Record

This record type is repeated for each LGR in the model with the last LGR data set terminated by the LGSGONE record to mark the end of the LGR input; see the following example.

Note

The LGR, LGRHEADI, LGRHEADQ and LGRHEADD always proceed LGR data sets and the ENDLGR record terminates the end of a data section of LGR data.

Example: RESTART Data - LGR Grid Header Record

The following example shows a typical formatted RESTART Data Record for 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
'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

D.7.9 RESTART DATA - LGR SOLUTION DATA RECORD

The file structure for RESTART Data - LGR Solution Data Record is similar to the global grid RESTART Data Grids record described previously. However, additional data that the defines the LGR properties (LGR Name for example) are included in this record definition and the LGR record is repeated for each LGR in the model. The record description is outlined in Table D.31.

Note that currently OPM Flow does not support LGR grids and therefore this record format cannot be used by OPM Flow.

No.	Record Name	RESTART Data - LGR Grid Property Data Record (Repeated For Each LGR In The Model)	Status Or Value
	LGR	LGR Grid Header Record See section D.7.8 RESTART Data - LGR Grid Header Record	Required
	LGRHEADI		
	LGRHEADQ		
	LGRHEADD		
	INITHEAD	Header Record See section D.7.3 RESTART Data - Header Record	
	LOGIHEAD		
	DOUBHEAD		
	IGRP SGRP etc.	Group, well, and connection data status records for this reporting time step. See section D.7.4 RESTART Data – Group, Well and Connection Data Records	Required
	HIDDEN	HIDDEN solution data, that is a list of solution arrays NOT to be read by post processing software.	
	SOLUTION	SOLUTION array section (PRESSURE, SGAS, SOILS, WAT, etc.) for the Global grid. See section D.7.7 RESTART Data - Solution Data Record	Required
	ENDSOL ENDLGR	LGR grid header section termination records See section D.7.8 RESTART Data - LGR Grid Header Record	Required

Notes:

1) Rows shaded in gray indicate sub-records in this header record type.
2) Item No. cells colored orange indicate that this item is not supported by OPM Flow and those colored red are either "Not Used" or "Undefined".
3) Note all character variables, including the Header Record Name should be enclosed in single quotations, for example the LGR header Record Name should be written out as 'LGR '.

Table D.31: RESTART Data – LGR Solution Data Record

This record type is repeated for each LGR in the model with the last LGR data set terminated by the ENDSOL and ENDLGR records to mark the end of the LGR input; see the following example.

Example: RESTART Data - LGR Solution Data Record

The following example shows a typical formatted RESTART Data - LGR Solution Data Record 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
 'INTEHEAD'         249 'INTE'
  -954922105      200400           1       -2345       -2345       -2345
       -2345       -2345           3           3           1           9
           3       -2345           7       -2345           0          36
           1          84           2           0           0           0
……..……..
           1           6           1          10           1          10
           9           1           7           0          30           3
          18          10           9
 'LOGIHEAD'          79 'LOGI'
  T   T   F   T   F   F   T   F   T   F   F   F   F   F   F   F   T   F   T   T   F   F   F   F   F
  F   F   F   F   F   F   T   F   F   F   F   F   F   F   F   F   F   F   F   F   F   F   F   F   F
  F   F   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
 'DOUBHEAD'         185 'DOUB'
   0.00000000000000D+00   0.10000000000000D+01   0.10000000000000D+02
……..……..
  -0.10000000200409D+21   0.10000000000000D+01   0.10000000000000D+01
   0.10000000000000D+01   0.00000000000000D+00   0.00000000000000D+00
   0.10000000000000D+01   0.10000000000000D-03
 'IGRP    '          358 'INTE'
           0           0           0           0           0           0
           0           0           0           0           0           0
……..……..
           0           0           0           2           2           0
           0           0           0           0
 'SGRP    '          186 '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   0.10000000E+01
 'XGRP    '          348 '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
 'ZGRP    '           10 'CHAR'
 '        ' '        ' '        ' '        ' '        ' 'FIELD   ' '        '
 '        ' '        ' '        '
 'ICOT    '            0 'INTE'
```

```
 'XCOT    '           0 'DOUB'
 'IWET    '           0 'INTE'
 'XWET    '           0 'DOUB'
 'IGRT    '          18 'INTE'
           0           0           0           0           0           0
           0           0           0           0           0           0
           0           0           0           0           0           0
 'XGRT    '         180 'DOUB'
   0.00000000000000D+00   0.00000000000000D+00   0.00000000000000D+00
   0.00000000000000D+00   0.00000000000000D+00   0.00000000000000D+00
   0.10000000200409D+21   0.10000000200409D+21   0.10000000200409D+21
   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
.....................................
   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   '
'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    '
'STARTSOL'           0 'MESS'
'PRESSURE'           9 'REAL'
  0.26823886E+03   0.26824997E+03   0.26826108E+03   0.26822879E+03
  0.26824088E+03   0.26825293E+03   0.26821875E+03   0.26823175E+03
  0.26824478E+03
 'SWAT    '           9 'REAL'
  0.10500000E+00   0.10500000E+00   0.10500000E+00   0.10500000E+00
  0.10500000E+00   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
  0.89499998E+00   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
  0.12021925E+03   0.12022604E+03   0.12020680E+03   0.12021413E+03
  0.12022144E+03
 'RV      '           9 'REAL'
  0.57467099E-04   0.57473411E-04   0.57479720E-04   0.57461380E-04
  0.57468234E-04   0.57475085E-04   0.57455662E-04   0.57463058E-04
  0.57470450E-04
.....................................
 'ENDSOL  '           0 'MESS'
 'ENDLGR  '           1 'INTE'
           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       -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'
  -954922105      200400           1       -2345       -2345       -2345
       -2345       -2345           3           3           1           9
           3       -2345           7       -2345           0          36
           9           1           7           0          30           3
          18          10           9
 'LOGIHEAD'          79 'LOGI'
  T  T  F  T  F  F  T  F  T  F  F  F  F  F  F  F  T  F  T  T  F  F  F  F  F
  F  F  F  F  F  F  T  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F  F
  F  F  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
 'DOUBHEAD'         185 'DOUB'
   0.00000000000000D+00   0.10000000000000D+01   0.10000000000000D+02
   0.10000000000000D-05   0.15000000000000D+00   0.30000000000000D+01
……..……..
   0.10000000000000D+01   0.00000000000000D+00   0.00000000000000D+00
   0.10000000000000D+01   0.10000000000000D-03
 'IGRP    '         358 'INTE'
           0           0           0           0           0           0
           0           0           0           0           0           0
……..……..
           0           0           0           2           2           0
           0           0           0           0
 'SGRP    '         186 '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   0.10000000E+01
 'XGRP    '         348 '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
 'ZGRP    '          10 'CHAR'
 '        ' '        ' '        ' '        ' '        ' 'FIELD   ' '        '
 '        ' '        ' '        '
 'ICOT    '           0 'INTE'
 'XCOT    '           0 'DOUB'
 'IWET    '           0 'INTE'
 'XWET    '           0 'DOUB'
 'IGRT    '          18 'INTE'
           0           0           0           0           0           0
           0           0           0           0           0           0
           0           0           0           0           0           0
 'XGRT    '         180 'DOUB'
   0.00000000000000D+00   0.00000000000000D+00   0.00000000000000D+00
   0.00000000000000D+00   0.00000000000000D+00   0.00000000000000D+00
……..……..
   0.10000000200409D+21   0.10000000200409D+21   0.10000000200409D+21
   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
……..……..
   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   '
 '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    '
 'STARTSOL'           0 'MESS'
 'PRESSURE'           9 'REAL'
   0.26813840E+03   0.26813367E+03   0.26812894E+03   0.26812656E+03
   0.26813242E+03   0.26813831E+03   0.26811469E+03   0.26813120E+03
   0.26814767E+03
 'SWAT    '           9 'REAL'
   0.10500000E+00   0.10500000E+00   0.10500000E+00   0.10500000E+00
   0.10500000E+00   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
   0.89499998E+00   0.89499998E+00   0.89499998E+00   0.89499998E+00
   0.89499998E+00
 'RS      '           9 'REAL'
   0.12016161E+03   0.12015896E+03   0.12015630E+03   0.12015495E+03
   0.12015826E+03   0.12016157E+03   0.12014828E+03   0.12015756E+03
   0.12016683E+03
 'RV      '           9 'REAL'
   0.57410016E-04   0.57407331E-04   0.57404646E-04   0.57403286E-04
   0.57406625E-04   0.57409969E-04   0.57396555E-04   0.57405923E-04
   0.57415291E-04
.....................................
 'ENDSOL  '           0 'MESS'
 'ENDLGR  '           1 'INTE'
           2
 'SEQNUM  '           1 'INTE'
            4
```

End of Example

D.8 RFT FILES – PRESSURE, SATURATION AND PRODUCTION LOG DATA

To be written in a future release of the manual.

D.9 SUMMARY FILES – PRODUCTION DATA

To be written in a future release of the manual.

D.10 SAVE FILES - INITIALIZATION AND SOLUTION DATA

This file format is currently not supported by OPM Flow.

End of Document