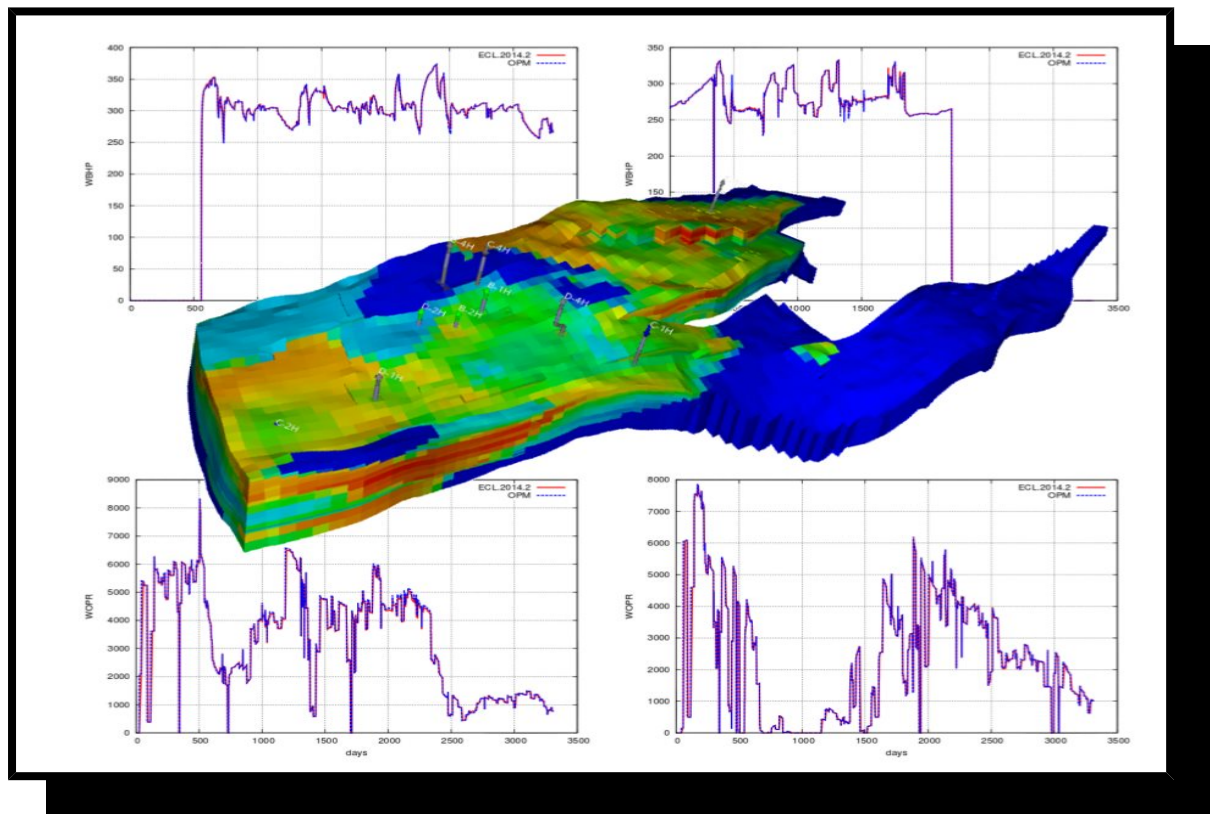


OPEN POROUS MEDIA

Flow Documentation Manual



OPM FLOW VERSION: 2017-10
MANUAL REVISION: Rev-0

OPEN POROUS MEDIA

Flow Documentation Manual

(2017-10)

DOCUMENT CONTROL			
Company	Statoil Petroleum AS		
Confidentiality, copyright and reproduction	<p>This document is Copyright 2017 by the OPEN POROUS MEDIA team.</p> <p><u>Documentation Contributors:</u></p> <p>David Baxendale Atgeirr Rasmussen Alf Birger Rustad Torbjørn Skille Tor Harald Sandve</p> <p>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/).</p> <p>All trademarks within this guide belong to their legitimate owners.</p>		
Document Type	OPEN POROUS MEDIA		
Document Title	Flow Documentation Manual (2017-10)		
Document Ref.	2017-03-RPT		
Document Revision	Date	Revision	Status
	October 3, 2017	Rev-0	Final
	August 30, 2017	Rev-F	Complete
	July 31, 2017	Rev-E	Additional Keywords
	June 26, 2017	Rev-D	Added Status to Keyword List
	June 2, 2017	Rev-C	Working Draft OPM format
	May 24, 2017	Rev-B	Working Draft OPM format
	May 17, 2017	Rev-A	Working Draft EIPC format
Editor	Equinox International Petroleum Consultants Pte. Ltd. 51 Goldhill Plaza, #07-10/11, Singapore 308900		
	David Baxendale		
	david.baxendale@eipc.co		
Document Name	D:\Projects\Statoil\Miscellaneous\M70 Technical Reports and Notes\OPM Flow Documentation 2017-10 Rev-0.odt		

TABLE OF CONTENTS

1 INTRODUCTION.....	21
2 INSTALLING AND RUNNING FLOW.....	22
2.1 Installing Flow.....	22
2.1.1 Ubuntu Linux 16.04 (64-bit version only).....	22
2.1.2 Red Hat Enterprise or CentOS (version 6 or 7).....	23
2.1.3 Using a Docker container.....	23
2.1.4 Installing from source.....	23
2.2 Running Flow.....	24
2.2.1 Command Line Options.....	24
3 KEYWORD DOCUMENTATION STRUCTURE.....	30
3.1 Keyword Definitions.....	30
3.2 Multi-Section Keywords.....	30
3.3 Keyword Formats.....	31
3.3.1 Keyword Format Type - Comment.....	31
3.3.2 Keyword Format Type - Activation.....	31
3.3.3 Keyword Format Type - Vector (Row Vector).....	32
3.3.4 Keyword Format Type - Vector (Columnar Vector).....	34
3.3.5 Keyword Format Type - Array.....	35
3.4 Input File Structure.....	36
4 GLOBAL SECTION KEYWORDS.....	37
4.1 Keyword Definitions.....	37
4.1.1 DEBUG - Define the Debug Data to be Printed to File.....	37
4.1.2 ECHO - Activate Echoing of User Input Files to the Print File.....	38
4.1.3 END - Define the End of the Input File.....	39
4.1.4 ENDINC - Define the End of an Include File.....	40
4.1.5 ENDSKIP - Deactivates Skipping of Keywords and Input Data.....	41
4.1.6 EXTRAPMS - Activate Extrapolation Warning Messages.....	42
4.1.7 INCLUDE - Load Another Data File at the Current Position.....	43
4.1.8 MESSAGES - Define Message Print Limits and Stop Limits.....	44
4.1.9 NOECHO - Deactivate Echoing of User Input Files to the Print File.....	46
4.1.10 NOWARN - Deactivate Warning Messages.....	47
4.1.11 SKIP - Activate Skipping of All Keywords and Input Date.....	48
4.1.12 SKIP100 - Activate Skipping of "Black-Oil" Keywords and Input Date.....	49
4.1.13 SKIP300 - Activate Skipping of "Compositional" Keywords and Input Date.....	50
4.1.14 WARN - Activate Warning Messages.....	51
5 RUNSPEC SECTION.....	52
5.1 Introduction.....	52
5.2 Keyword Definitions.....	53
5.2.1 ACTDIMS - ACTION Keyword Dimensions.....	53
5.2.2 AITS - Intelligent Time Stepping Activation.....	54
5.2.3 API - Activate API Tracking.....	55
5.2.4 AQUODIMS - Aquifer Dimensions.....	56
5.2.5 CPR - Activate Constrained Pressure Residual ("CPR") Linear Solver.....	57
5.2.6 DIMENS - Define the Dimension of the Model.....	58
5.2.7 DISGAS - Define Dissolved Gas to be Present in the Model.....	59
5.2.8 ENDSCALE - Activate Relative Permeability End-Point Scaling Option.....	60
5.2.9 EQLDIMS - Define the Equilibration Data Dimensions.....	62
5.2.10 EQLOPTS - Activates the Equilibration Options.....	63
5.2.11 FAULTDIM - Define the Number of Fault Segments.....	64

5.2.12 FIELD - Activates the Oil Field System of Units for the Model.....	65
5.2.13 FMTIN - Activates The Format Input File Option.....	66
5.2.14 FMTOU - Activates The Format Output File Option.....	68
5.2.15 FULLIMP - Activates Fully Implicit Solution Option.....	70
5.2.16 GAS - Define the Gas Phase to be Present in the Model.....	71
5.2.17 GRIDOPTS - Grid Processing Options.....	72
5.2.18 IMPES - Activates Implicit Pressure Explicit Saturation Solution Option.....	74
5.2.19 LAB - Activates the Laboratory System of Units for the Model.....	75
5.2.20 LGR - Define Local Grid Refinement Parameters.....	76
5.2.21 MEMORY - Define Allocated Memory.....	77
5.2.22 METRIC - Activates the Metric System of Units for the Model.....	78
5.2.23 MISCIBLE - Define Miscibility Todd-Longstaff Parameters.....	79
5.2.24 MONITOR - Activates Output of the Monitoring Data and File.....	80
5.2.25 MSGFILE - Active or Deactivate Message File Output.....	81
5.2.26 NOCASC - Activate Linear Solver Tracer Algorithm.....	82
5.2.27 NOINSPEC - Deactivate Output of the INIT Index File.....	83
5.2.28 NOMONITO - Deactivates Output of the Monitoring Data and File.....	84
5.2.29 NONNC - Deactivates Non-Neighbor Connections.....	85
5.2.30 NORSSPEC - Deactivate Output of the RESTART Index File.....	86
5.2.31 NOSIM - Activates the No Simulation Mode for Data File Checking.....	87
5.2.32 NSTACK - Define the Stack Length for the Iterative Linear Solver.....	88
5.2.33 NUMRES - Define the Number of Reservoir Grids.....	89
5.2.34 NUPCOL - Define the Number of Newtonian Iterations Used to Update Well Targets.....	90
5.2.35 OIL - Define the Oil Phase to be Present in the Model.....	91
5.2.36 OPTIONS - Activate Various Program Options.....	92
5.2.37 PARALLEL - Define Run Configuration.....	93
5.2.38 PATHS - Define Filename Directory Path Aliases.....	94
5.2.39 POLYMER - Define the POLYMER Phase to be Present in the Model.....	95
5.2.40 PIMTDIMS - Define Well Productivity Scaling Table Dimensions.....	96
5.2.41 RADIAL - Radial Grid Activation Option.....	97
5.2.42 REGDIMS - Define the Maximum Number of Regions for a Region Array.....	98
5.2.43 ROCKCOMP - Activate Rock Compaction.....	100
5.2.44 RPTRUNSP - Activates RUNSPEC Reporting.....	102
5.2.45 RUNSPEC - Define the Start of the RUNSPEC Section of Keywords.....	103
5.2.46 SATOPTS - Activate Relative Permeability Assignment Options.....	104
5.2.47 SAVE - Activates Output of a SAVE File for Fast Restarts.....	108
5.2.48 SMRYDIMS - Define Maximum Number of Summary Vectors to be Written.....	109
5.2.49 SOLVENT - Define the SOLVENT Phase to be Present in the Model.....	110
5.2.50 START - Simulation Start Date.....	111
5.2.51 TABDIMS - Define the Number of Tables and the Table Dimensions.....	112
5.2.52 TEMP - Activate the Temperature Modeling Option.....	115
5.2.53 TITLE - Define the Title for the Input Deck.....	116
5.2.54 TRACERS - Activate Tracer Options and Set Tracer Array Dimensions.....	117
5.2.55 UDADIMS - Define the Dimensions of the User Defined Arguments Feature.....	119
5.2.56 UDQDIMS - Define the Dimensions of the User Defined UDQ Feature.....	120
5.2.57 UNIFIN - Activates The Unified Input File Option.....	121
5.2.58 UNIFOUT - Activates The Unified Output File Option.....	123
5.2.59 VAPOIL - Define Vaporize Oil in Wet Gas to be Present in the Model.....	125
5.2.60 VFPIDIMS - Injection Vertical Flow Performance Table Dimensions.....	126
5.2.61 VFPPDIMS - Production Vertical Flow Performance Table Dimensions.....	127
5.2.62 WATER - Define the Water Phase to be Present in the Model.....	128
5.2.63 WELLDIMS - Define the Wells and Group Dimensions.....	129
5.2.64 WSEGDIMS - Define Multi-Segment Well Dimensions.....	131

6 GRID SECTION..... 132

6.1 Introduction.....	132
6.2 Data Requirements.....	133
6.2.1 Cartesian Regular Grid.....	133
6.2.2 Radial Grid.....	134
6.2.3 Irregular Corner-Point Grids.....	135
6.2.4 Rock Properties.....	138

6.3 Keyword Definitions.....	139
6.3.1 ACTNUM - Set the Status of a Grid Block To Active or Inactive.....	139
6.3.2 ADD - Add a Constant to a Specified Array.....	140
6.3.3 ADDREG - Add a Constant to an Array based on a Region Number.....	142
6.3.4 AQUANCON - Analytic Aquifer Assignment.....	145
6.3.5 AQUCON - Numerical Aquifers Connection to the Grid.....	146
6.3.6 AQUNUM - Numerical Aquifer Assignment.....	147
6.3.7 BOX - Define a Range of Grid Blocks to Enter Property Data.....	148
6.3.8 CIRCLE - Completion of Radial Grid Circle Activation.....	150
6.3.9 COALNUM - Define the Coal Region Numbers.....	151
6.3.10 COORD - Define a Set of Coordinates Lines for a Reservoir Grid.....	152
6.3.11 COORDSYS - Define Coordinate Grid Options.....	154
6.3.12 COPY - Copy Array Data to Another Array.....	155
6.3.13 COPYREG - Copy an Array to Another Array based on a Region Number.....	157
6.3.14 DRV - Define the Size of Grid Blocks in the R Direction via a Vector.....	159
6.3.15 DTHETAV - Sets the Size of Grid Blocks in THETA Direction via a Vector.....	160
6.3.16 DUMPFLUX - Activate Writing Out of a Flux File.....	161
6.3.17 DX - Define the Size of Grid Blocks in the X Direction for All Cells.....	162
6.3.18 DXV - Define the Size of Grid Blocks in the X Direction via a Vector.....	163
6.3.19 DY - Define the Size of Grid Blocks in the Y Direction for All Cells.....	164
6.3.20 DYV - Define the Size of Grid Blocks in the Y Direction via a Vector.....	165
6.3.21 DZ - Define the Size of Grid Blocks in the Z Direction for All Cells.....	166
6.3.22 DZV - Define the Size of Grid Blocks in the Z Direction via a Vector.....	167
6.3.23 ENDBOX - Define the End of the BOX Defined Grid.....	168
6.3.24 EQUALREG - Sets an Array to a Constant by Region Number.....	169
6.3.25 EQUALS - Sets a Specified Array to a Constant.....	172
6.3.26 FAULTS - Define Faults in the Grid Geometry.....	174
6.3.27 FLUXNUM - Define the Flux Regions.....	176
6.3.28 GDORIENT - Define Grid Orientation Parameters.....	177
6.3.29 GRID - Define the Start of the GRID Section of Keywords.....	178
6.3.30 GRIDFILE - Set the Grid File Output Options.....	179
6.3.31 GRIDUNIT - Define the Grid Units.....	180
6.3.32 INIT - Activates the INIT File Option.....	181
6.3.33 INRAD - Define the Inner Radius of a Radial Grid.....	182
6.3.34 ISOLNUM - Define the Independent Reservoir Regions.....	183
6.3.35 JFUNC - Activates the Leverett J-function Option.....	184
6.3.36 MAPAXES - Define the Map Origin Input Data.....	187
6.3.37 MAPUNITS - Define the Map Axes Units.....	188
6.3.38 MAXVALUE - Sets a Maximum Value for an Array Element.....	189
6.3.39 MINPV - Set a Minimum Grid Block Pore Volume Threshold.....	191
6.3.40 MINVALUE - Set a Minimum Value for an Array Element.....	192
6.3.41 MULTFLT - Multiply the Transmissibility of a Defined Fault by a Constant.....	194
6.3.42 MULTIPLY - Multiply a Specified Array by a Constant.....	195
6.3.43 MULTIREG - Multiply an Array by a Constant based on a Region Number.....	197
6.3.44 MULTNUM - Define the Multiple Transmissibility Regions.....	200
6.3.45 MULTPV - Multiply Cell Pore Volumes by a Constant.....	201
6.3.46 MULTREGP - Multiply Pore Volumes Based On Region Number.....	202
6.3.47 MULTREGT - Multiply Transmissibilities Between Regions.....	203
6.3.48 MULTX - Multiply Cell Transmissibility in the +X Direction.....	205
6.3.49 MULTX - Multiply Cell Transmissibility in the -X Direction.....	206
6.3.50 MULTY - Multiply Cell Transmissibility in the +Y Direction.....	207
6.3.51 MULTY - Multiply Cell Transmissibility in the -Y Direction.....	208
6.3.52 MULTZ - Multiply Cell Transmissibility in the +Z Direction.....	209
6.3.53 MULTZ - Multiply Cell Transmissibility in the -Z Direction.....	210
6.3.54 NEWTRAN - Activate Irregular Corner-Point Grid Transmissibilities.....	211
6.3.55 NNC - Define Non-Neighbor Connections Between Cells Manually.....	212
6.3.56 NOGGF - Deactivate Output of Grid Geometry File.....	214
6.3.57 NTG - Define the Net-to-Gross Ratio for All the Cells.....	215
6.3.58 OLDTRAN - Activate Cartesian Regular Grid Transmissibilities.....	216
6.3.59 OPERATE - Define Mathematical Operations on Arrays.....	217
6.3.60 OPERNUM - Define Regions for Mathematical Operations on Arrays.....	218
6.3.61 OUTRAD - Define the Outer Radius of a Radial Grid.....	219
6.3.62 PERMR - Define the Permeability for Each Cell in the R Direction.....	220

6.3.63 PERMX - Define the Permeability in the X Direction for All the Cells.....	221
6.3.64 PERMXX - Define the Permeability Tensor in the XX Direction for All the Cells.....	222
6.3.65 PERMXY - Define the Permeability Tensor in the XY Direction for All the Cells.....	223
6.3.66 PERMY - Define the Permeability in the Y Direction for All the Cells.....	224
6.3.67 PERMY - Define the Permeability Tensor in the YY Direction for All the Cells.....	225
6.3.68 PERMYZ - Define the Permeability Tensor in the YZ Direction for All the Cells.....	226
6.3.69 PERMZ - Define the Permeability in the Z Direction for All the Cells.....	227
6.3.70 PERMZ - Define the Permeability Tensor in the ZX Direction for All the Cells.....	228
6.3.71 PERMZZ - Define the Permeability Tensor in the ZZ Direction for All the Cells.....	229
6.3.72 PINCH - Define Pinch-Out Layer Options.....	230
6.3.73 PINCHNUM - Define Pinch-Out Regions for the PINCHREG Keyword.....	232
6.3.74 PINCHREG - Define Pinch-Out Region Options.....	233
6.3.75 PORO - Define the Porosity Values for All the Cells.....	235
6.3.76 RADFIN - Define a Radial Local Grid Refinement with One Column.....	236
6.3.77 RADFIN4 - Define a Radial Local Grid Refinement with Four Columns.....	237
6.3.78 RPTGRID - Define GRID Section Reporting.....	238
6.3.79 SPECGRID - Define the Dimensions of a Corner-Point Grid.....	239
6.3.80 SWATINIT - Define the Initial Water Saturation Array for Capillary Pressure Scaling.....	240
6.3.81 THCONR - Define Rock and Fluid Thermal Conductivity.....	241
6.3.82 TOPS - Define the Depth at the Center of the Top Face for Each Cell.....	242
6.3.83 ZCORN - Define the Depth of Each Corner-Point of a Grid Block.....	244

7 EDIT SECTION.....245

7.1 Introduction.....	245
7.2 Data Requirements.....	245
7.3 Keyword Definitions.....	246
7.3.1 ADD - Add a Constant to a Specified Array.....	246
7.3.2 ADDREG - Add a Constant to an Array based on a Region Number.....	246
7.3.3 BOX - Define a Range of Grid Blocks to Enter Property Data.....	246
7.3.4 COPY - Copy Array Data to Another Array.....	246
7.3.5 COPYREG - Copy an Array to Another Array based on a Region Number.....	246
7.3.6 DEPTH - Edits the Depth at the Center of Each Cell.....	247
7.3.7 EDIT - Define the Start of the EDIT Section of Keywords.....	248
7.3.8 ENDBOX - Define the End of the BOX Defined Grid.....	249
7.3.9 EDITNNC - Edit Non-Neighbor Connections Between Cells Manually.....	250
7.3.10 EQUALREG - Sets an Array to a Constant by Region Number.....	252
7.3.11 EQUALS - Sets a Specified Array to a Constant.....	252
7.3.12 MAXVALUE - Sets a Maximum Value for an Array Element.....	252
7.3.13 MINVALUE - Set a Minimum Value for an Array Element.....	252
7.3.14 MULTFLT - Multiply the Transmissibility of a Defined Fault by a Constant.....	252
7.3.15 MULTIPLY - Multiply a Specified Array by a Constant.....	252
7.3.16 MULTIREG - Multiply an Array by a Constant based on a Region Number.....	253
7.3.17 MULTPV - Multiply Cell Pore Volumes by a Constant.....	253
7.3.18 MULTREGP - Multiply Pore Volumes Based On Region Number.....	253
7.3.19 MULTREGT - Multiply Transmissibilities Between Regions.....	253
7.3.20 MULTX - Multiply Cell Transmissibility in the +X Direction.....	253
7.3.21 MULTX- - Multiply Cell Transmissibility in the -X Direction.....	253
7.3.22 MULTY - Multiply Cell Transmissibility in the +Y Direction.....	254
7.3.23 MULTY- - Multiply Cell Transmissibility in the -Y Direction.....	254
7.3.24 MULTZ - Multiply Cell Transmissibility in the +Z Direction.....	254
7.3.25 MULTZ- - Multiply Cell Transmissibility in the -Z Direction.....	254
7.3.26 OPERATE - Define Mathematical Operations on Arrays.....	254
7.3.27 PORV - Define the Pore Volumes for All the Cells.....	255
7.3.28 TRANX - Define the Transmissibility in the X Direction for All the Cells.....	256
7.3.29 TRANY - Define the Transmissibility in the Y Direction for All the Cells.....	257
7.3.30 TRANZ - Define the Transmissibility in the Z Direction for All the Cells.....	258

8 PROPS SECTION.....259

8.1 Introduction.....	259
-----------------------	-----

8.2 Data Requirements.....	259
8.2.1 Fluid Property Tables.....	259
8.2.2 Saturation Tables (Relative Permeability and Capillary Pressure Tables).....	260
8.3 Keyword Definitions.....	263
8.3.1 ADD - Add a Constant to a Specified Array.....	263
8.3.2 ADDREG - Add a Constant to an Array based on a Region Number.....	263
8.3.3 ADSALNOD - Salt Concentration Based on SATNUM Array.....	264
8.3.4 ASPKDAM - Define Asphaltene Permeability Damage.....	266
8.3.5 ASPPW2D - Define Asphaltene Two Parameters Precipitation Data.....	267
8.3.6 ASPREWG - Define Asphaltene as Percentage Weight.....	268
8.3.7 ASPWETF - Define Asphaltene Wettability Factor Data.....	269
8.3.8 BOX - Define a Range of Grid Blocks to Enter Property Data.....	270
8.3.9 COPY - Copy Array Data to Another Array.....	270
8.3.10 COPYREG - Copy an Array to Another Array based on a Region Number.....	270
8.3.11 DENSITY - Define the Surface Oil, Water Gas Densities for the Fluids.....	271
8.3.12 EHYSTR - Define Hysteresis Model and Parameters.....	273
8.3.13 ENDBOX - Define the End of the BOX Defined Grid.....	276
8.3.14 ENKRVD - Define Relative Permeability End-Points versus Depth Functions.....	277
8.3.15 ENPTVD - Define Relative Permeability Saturation End-Points versus Depth.....	278
8.3.16 EQUALREG - Sets an Array to a Constant by Region Number.....	279
8.3.17 EQUALS - Sets a Specified Array to a Constant.....	279
8.3.18 FILLEPS - Activate Saturation End-Point Export to the INIT File.....	280
8.3.19 GRAVITY- Define the Surface Oil, Water Gas Gravities for the Fluids.....	281
8.3.20 IMKRVD - Imbibition Relative Permeability End-Points versus Depth Functions.....	283
8.3.21 IMPTVD - Imbibition Relative Permeability Saturation End-Points versus Depth.....	284
8.3.22 IPCG - End-Point Scaling of Grid Cell Gas Capillary Pressure (Imbibition).....	285
8.3.23 IPCW - End-Point Scaling of Grid Cell Water Capillary Pressure (Imbibition).....	286
8.3.24 ISGCR - End-Point Scaling of Grid Cell Critical Gas Saturation (Imbibition).....	287
8.3.25 ISGL - End-Point Scaling of Grid Cell Connate Gas Saturation (Imbibition).....	288
8.3.26 ISGU - End-Point Scaling of Grid Cell Maximum Gas Saturation (Imbibition).....	289
8.3.27 ISOGCR - End-Point Scaling of Grid Cell Critical Oil Saturation to Gas (Imbibition).....	290
8.3.28 ISOWCR - End-Point Scaling of Grid Cell Critical Oil Saturation to Water (Imbib.).....	291
8.3.29 ISWCR - End-Point Scaling of Grid Cell Critical Water Saturation (Imbibition).....	292
8.3.30 ISWL - End-Point Scaling of Grid Cell Connate Water Saturation (Imbibition).....	293
8.3.31 ISWU - End-Point Scaling of Grid Cell Maximum Water Saturation (Imbibition).....	294
8.3.32 KRG - End-Point Scaling of Grid Cell Krg(Sgu) (Drainage).....	295
8.3.33 KRGR - End-Point Scaling of Grid Cell Krgr(1-Sogcr) (Drainage).....	297
8.3.34 KRO - End-Point Scaling of Grid Cell Kro(Swl) (Drainage).....	299
8.3.35 KRORG - End-Point Scaling of Grid Cell Kro(Sgcr) (Drainage).....	301
8.3.36 KRORW - End-Point Scaling of Grid Cell Kro(Swcr) (Drainage).....	303
8.3.37 KRW - End-Point Scaling of Grid Cell Krw(Sw =1.0) (Drainage).....	305
8.3.38 KRWR - End-Point Scaling of Grid Cell KRWR(Sw =1.0) (Drainage).....	307
8.3.39 MAXVALUE - Sets a Maximum Value for an Array Element.....	309
8.3.40 MINVALUE - Set a Minimum Value for an Array Element.....	309
8.3.41 MSFN - Miscible Normalized Relative Permeability Tables.....	310
8.3.42 MULTIPLY - Multiply a Specified Array by a Constant.....	311
8.3.43 MULTIREG - Multiply an Array by a Constant based on a Region Number.....	311
8.3.44 OILVISCT - Oil Viscosity versus Temperature Functions.....	312
8.3.45 OPERATE - Define Mathematical Operations on Arrays.....	313
8.3.46 PCG - End-Point Scaling of Grid Cell Maximum Gas Capillary Pressure (Drainage).....	314
8.3.47 PCW - End-Point Scaling of Grid Cell Water Capillary Pressure (Drainage).....	315
8.3.48 PLMIXPAR - Define the Polymer Todd-Longstaff Mixing Parameters.....	316
8.3.49 PLYADS - Define Polymer Rock Adsorption Tables.....	317
8.3.50 PLYADSS - Define Polymer Rock Adsorption with Salt Dependence Tables.....	319
8.3.51 PLYDHFLF - Define Polymer Thermal Degradation Half-Life Tables.....	321
8.3.52 PLYMAX - Define Polymer-Salt Viscosity Mixing Concentrations.....	323
8.3.53 PLYROCK - Define Polymer-Rock Properties.....	324
8.3.54 PLYSHEAR - Activate and Define Polymer Shearing Parameters.....	326
8.3.55 PLYSHLOG - Activate and Define the Polymer Shearing Logarithmic Parameters.....	327
8.3.56 PLYVISC - Define Polymer Viscosity Scaling Factors.....	328
8.3.57 PMISC - Define Miscibility versus Pressure Tables.....	330
8.3.58 PROPS - Define the Start of the PROPS Section of Keywords.....	331

8.3.59 PVCDO - Oil PVT Properties for Dead Oil (Constant Compressibility).....	332
8.3.60 PVDG - Gas PVT Properties for Dry Gas.....	334
8.3.61 PVDO - Oil PVT Properties for Dead Oil.....	336
8.3.62 PVDS - Solvent PVT Properties for the Solvent Model.....	338
8.3.63 PVTG - Gas PVT Properties for Wet Gas.....	340
8.3.64 PVTO - Oil PVT Properties for Live Oil.....	342
8.3.65 PVTW - Define Water Fluid Properties for Various Regions.....	345
8.3.66 RKTRMDIR - Activate ROCKTAB Keyword Directional Transmissibility Multipliers.....	347
8.3.67 ROCK - Define the Rock Compressibility for Various Regions.....	348
8.3.68 ROCKOPTS - Define Rock Compaction and Compressibility Options.....	350
8.3.69 ROCKTAB - Rock Compaction Tables.....	352
8.3.70 RPTPROPS - Define PROPS Section Reporting.....	355
8.3.71 RTEMP - Define the Initial Reservoir Temperature for the Model.....	356
8.3.72 RTEMPA - Define the Initial Reservoir Temperature for the Model.....	357
8.3.73 SALNODE - Salt Concentration Based PVTNUM Array.....	358
8.3.74 SCALECRS - Set End-Point Scaling Option.....	359
8.3.75 SDENSITY - Define the Miscible or Solvent Surface Gas Density.....	360
8.3.76 SGCR - End-Point Scaling Grid Cell Critical Gas Saturations.....	361
8.3.77 SGCWMIS - Miscible Critical Gas versus Water Saturation Functions.....	362
8.3.78 SGFN - Gas Saturation Tables (Format Type 2).....	363
8.3.79 SGL - End-Point Scaling Grid Cell Connate Gas Saturations.....	364
8.3.80 SGOF - Gas-Oil Saturation Tables versus Gas (Format Type 1).....	365
8.3.81 SGU - End-Point Scaling Grid Cell Gas Saturation.....	367
8.3.82 SGWFN - Gas-Water Saturation Tables (Format Type 2).....	368
8.3.83 SHRATE - Activate and Define the Polymer Shearing Logarithmic Parameters.....	370
8.3.84 SLGOF - Gas-Oil Saturation Tables versus Gas (Format Type 1).....	371
8.3.85 SOF2 - Oil Saturation Tables with Respect to Gas or Water (Format Type 2).....	373
8.3.86 SOF3 - Oil Saturation Tables with Respect to Gas and Water (Format Type 2).....	375
8.3.87 SOGCR - End-Point Scaling Grid Cell Critical Oil Saturation with Respect to Gas.....	377
8.3.88 SORWMIS - Miscible Residual Oil versus Water Saturation Functions.....	378
8.3.89 SOWCR - End-Point Scaling Grid Cell Critical Oil Saturation with Respect to Water.....	379
8.3.90 SPECHEAT - Define the Specific Heat of Oil, Water and Gas.....	380
8.3.91 SPECROCK - Define the Specific Heat of the Reservoir Rock.....	382
8.3.92 SSFN - Solvent and Gas Relative Permeability Tables.....	384
8.3.93 SWCR - End-Point Scaling Grid Cell Critical Water Saturation.....	386
8.3.94 SWFN - Water Saturation Tables (Format Type 2).....	387
8.3.95 SWL - End-Point Scaling Grid Cell Connate Water Saturation.....	389
8.3.96 SWOF - Water-Oil Saturation Tables (Format Type 1).....	390
8.3.97 SWU - End-Point Scaling Grid Cell Gas Saturation.....	392
8.3.98 TLMIXPAR - Define the Miscible Todd-Longstaff Mixing Parameters.....	393
8.3.99 TRACER - Define Passive Tracer Variables.....	394
8.3.100 VISCREF - Define Viscosity-Temperature Reference Conditions.....	396
8.3.101 WATVISCT - Water Viscosity versus Temperature Functions.....	397

9 REGIONS SECTION.....398

9.1 Introduction.....	398
9.2 Data Requirements.....	398
9.3 Keyword Definitions.....	399
9.3.1 ADD - Add a Constant to a Specified Array.....	399
9.3.2 ADDREG - Add a Constant to an Array based on a Region Number.....	399
9.3.3 BOX - Define a Range of Grid Blocks to Enter Property Data.....	399
9.3.4 COPY - Copy Array Data to Another Array.....	399
9.3.5 COPYREG - Copy an Array to Another Array based on a Region Number.....	399
9.3.6 ENDBOX - Define the End of the BOX Defined Grid.....	400
9.3.7 ENDNUM - Define the End-Point Scaling Depth Region Numbers.....	401
9.3.8 EQLNUM - Define the Equilibration Region Numbers.....	402
9.3.9 EQUALREG - Sets an Array to a Constant by Region Number.....	403
9.3.10 EQUALS - Sets a Specified Array to a Constant.....	403
9.3.11 FIPNUM - Define the Fluid In-Place Region Numbers.....	404
9.3.12 FIPOWG - Activate Oil, Gas, and Water FIP Zone Reporting.....	406
9.3.13 IMBNUM - Define the Imbibition Saturation Table Region Numbers.....	407

9.3.14 MISNUM - Define the Miscibility Region Numbers.....	408
9.3.15 MULTIPLY - Multiply a Specified Array by a Constant.....	409
9.3.16 MULTIREG - Multiply an Array by a Constant based on a Region Number.....	409
9.3.17 OPERATE - Define Mathematical Operations on Arrays.....	409
9.3.18 OPERNUM - Define Regions for Mathematical Operations on Arrays.....	409
9.3.19 PLMIXNUM - Define the Polymer Region Numbers.....	410
9.3.20 PVTNUM - Define the PVT Regions.....	411
9.3.21 REGIONS - Define the Start of the REGIONS Section of Keywords.....	413
9.3.22 ROCKNUM - Define Rock Compaction Table Region Numbers.....	414
9.3.23 RPTREGS - Define REGIONS Section Reporting.....	415
9.3.24 SATNUM - Define the Saturation Table Region Numbers.....	416
9.3.25 TNUM - Define Passive Tracer Concentration Regions.....	417

10 SOLUTION SECTION.....419

10.1 Introduction.....	419
10.2 Data Requirements.....	419
10.3 Keyword Definitions.....	420
10.3.1 ADD - Add a Constant to a Specified Array.....	420
10.3.2 ADDREG - Add a Constant to an Array based on a Region Number.....	420
10.3.3 AQUFETP - Fetkovich Aquifer Definition.....	421
10.3.4 BOX - Define a Range of Grid Blocks to Enter Property Data.....	422
10.3.5 COPY - Copy Array Data to Another Array.....	422
10.3.6 COPYREG - Copy an Array to Another Array based on a Region Number.....	422
10.3.7 DATUM - Define the Datum Depth for the Model.....	423
10.3.8 DATUMR - Define Datum Depths for the FIPNUM Regions.....	424
10.3.9 ENDBOX - Define the End of the BOX Defined Grid.....	425
10.3.10 EQUALREG - Sets an Array to a Constant by Region Number.....	425
10.3.11 EQUALS - Sets a Specified Array to a Constant.....	425
10.3.12 EQUIL - Define the Equilibration Initialization Data.....	426
10.3.13 MULTIPLY - Multiply a Specified Array by a Constant.....	429
10.3.14 MULTIREG - Multiply an Array by a Constant based on a Region Number.....	429
10.3.15 OPERATE - Define Mathematical Operations on Arrays.....	429
10.3.16 PBUB - Define the Initial Equilibration Bubble-Point Pressure for All Grid Blocks.....	430
10.3.17 PBVD - Equilibration Bubble-Point versus Depth Tables.....	431
10.3.18 PDEW - Define the Initial Equilibration Dew-Point Pressure for All Grid Blocks.....	432
10.3.19 PDVD - Equilibration Dew-Point versus Depth Tables.....	433
10.3.20 PRESSURE - Define the Initial Equilibration Pressures for All Grid Blocks.....	435
10.3.21 RESTART - Restart Run From an Existing Restart File.....	436
10.3.22 RPTRST - Define Data to be Written to the RESTART File.....	438
10.3.23 RPTSOL - Define SOLUTION Section Reporting.....	441
10.3.24 RS - Define the Initial Equilibration GOR (Rs) for All Grid Blocks.....	442
10.3.25 RSVD - Equilibration Dissolved Gas-Oil Ratio (Rs) versus Depth Tables.....	443
10.3.26 RTEMP - Define the Initial Reservoir Temperature for the Model.....	444
10.3.27 RTEMPA - Define the Initial Reservoir Temperature for the Model.....	444
10.3.28 RTEMPVD - Define the Initial Reservoir Temperature versus Depth Tables.....	445
10.3.29 RV - Define the Initial Equilibration CGR (Rv) for All Grid Blocks.....	447
10.3.30 RVVD - Equilibration Vaporized Oil-Gas Ratio (Rv) versus Depth Tables.....	448
10.3.31 SGAS - Define the Initial Equilibration Gas Saturation for All Grid Blocks.....	449
10.3.32 SOIL - Define the Initial Equilibration Oil Saturation for All Grid Blocks.....	450
10.3.33 SOLUTION - Define the Start of the SOLUTION Section of Keywords.....	451
10.3.34 SSOL - Define the Initial Equilibration Solvent Saturation for All Grid Blocks.....	452
10.3.35 STONE1 - Activate Stone's First Three Phase Oil Relative Permeability Model.....	453
10.3.36 STONE1EX - Define Stone's First Three Phase Oil Relative Permeability Parameter.....	454
10.3.37 STONE2 - Activate Stone's Second Three Phase Oil Relative Permeability Model.....	455
10.3.38 SWAT - Define the Initial Equilibration Water Saturation for All Grid Blocks.....	456
10.3.39 TEMPVD - Define the Initial Reservoir Temperature versus Depth Tables.....	457
10.3.40 THPRES - Define Equilibration Region Threshold Pressures.....	459
10.3.41 TVDP - Define the Initial Equilibration Tracer Saturation versus Depth Functions.....	461
10.3.42 VAPPARS - Oil Vaporization Parameters.....	462

11 SUMMARY SECTION.....464

11.1 Introduction.....	464
11.2 Data Requirements.....	464
11.3 Keyword Definitions.....	465
11.3.1 ALL - Export Standard Summary Variable Vectors to File.....	465
11.3.2 DATE - Activate the DATE Option for the SUMMARY File.....	466
11.3.3 EXCEL - Activate the EXCEL Option for the SUMMARY File.....	467
11.3.4 FWSET - Export Well Status Vectors for the Field to File.....	468
11.3.5 GWSET - Export Well Status Vectors by Group to File.....	469
11.3.6 MONITOR - Activates Output of the Monitoring Data and File.....	470
11.3.7 NOMONITO - Deactivates Output of the Monitoring Data and File.....	470
11.3.8 RPTONLY - Activate the Report Time Steps Only Option for the SUMMARY File.....	471
11.3.9 RPTONLYO - Deactivate the Report Time Steps Only Option for the SUMMARY File.....	472
11.3.10 RUNSUM - Activate RSM File Output of the SUMMARY Data.....	473
11.3.11 SEPARATE - Activate the Separate RSM File Output Option.....	474
11.3.12 SUMMARY - Define the Start of the SUMMARY Section of Keywords.....	475
11.3.13 SUMTHIN - Define SUMMARY DATA Reporting Time Steps.....	476
12 SCHEDULE SECTION.....	477
12.1 Introduction.....	477
12.2 Data Requirements.....	477
12.3 Keyword Definitions.....	478
12.3.1 BOX - Define a Range of Grid Blocks to Enter Property Data.....	478
12.3.2 CECON - Define Well Connections Economic Limit Criteria.....	479
12.3.3 COMPDAT - Define Well Connections to the Grid.....	480
12.3.4 COMPIMB - Assign Imbibition Saturation Tables to Well Connections.....	484
12.3.5 COMPLUMP - Assign Well Connections to Completions.....	486
12.3.6 COMPORD - Define Well Connection Ordering.....	488
12.3.7 COMPSEGS - Define Well Connections for Multi-Segment Wells.....	490
12.3.8 DATES - Advance Simulation by Reporting Date.....	491
12.3.9 DRSDT - Solution Gas (Rs) Maximum Rate of Increase Parameters.....	494
12.3.10 DRVDT - Solution Oil (Rv) Maximum Rate of Increase Parameters.....	496
12.3.11 ENDBOX - Define the End of the BOX Defined Grid.....	497
12.3.12 GCONINJE - Group Injection Targets and Constraints.....	498
12.3.13 GCONPROD - Group Production Targets and Constraints.....	501
12.3.14 GECON - Group Economic Criteria for Production Groups.....	505
12.3.15 GEFAC - Define Group Efficiency.....	508
12.3.16 GRUPNET - Define Group Standard Network Parameters.....	509
12.3.17 GRUPRIG - Group Drilling and Workover Rig Specifications.....	513
12.3.18 GRUPTREE - Define Group Tree Hierarchy.....	514
12.3.19 MULTFLT - Multiply the Transmissibility of a Defined Fault by a Constant.....	516
12.3.20 MULTPV - Multiply Cell Pore Volumes by a Constant.....	516
12.3.21 MULTX - Multiply Cell Transmissibility in the +X Direction.....	516
12.3.22 MULTX- - Multiply Cell Transmissibility in the -X Direction.....	516
12.3.23 MULTY - Multiply Cell Transmissibility in the +Y Direction.....	516
12.3.24 MULTY- - Multiply Cell Transmissibility in the -Y Direction.....	516
12.3.25 MULTZ - Multiply Cell Transmissibility in the +Z Direction.....	517
12.3.26 MULTZ- - Multiply Cell Transmissibility in the -Z Direction.....	517
12.3.27 NETBALAN - Network Balancing Parameters.....	518
12.3.28 NEXTSTEP - Maximum Next Time Step Size.....	519
12.3.29 NUPCOL - Define the Number of Newtonian Iterations Used to Update Well Targets.....	522
12.3.30 PIMULTAB - Define Well Productivity Index versus Water Cut Tables.....	523
12.3.31 PLYADS - Define Polymer Rock Adsorption Tables.....	526
12.3.32 PLYDHFLF - Define Polymer Thermal Degradation Half-Life Tables.....	526
12.3.33 PLYMAX - Define Polymer-Salt Viscosity Mixing Concentrations.....	526
12.3.34 PLYSHEAR - Activate and Define Polymer Shearing Parameters.....	526
12.3.35 PLYSHLOG - Activate and Define the Polymer Shearing Logarithmic Parameters.....	526
12.3.36 PLYVISC - Define Polymer Viscosity Scaling Fact.....	526
12.3.37 PRORDER - Define a Group Production Rules Sequence.....	527
12.3.38 RPTONLY - Activate the Report Time Steps Only Option for the SUMMARY File.....	528

12.3.39 RPTONLYO - Deactivate the Report Time Steps Only Option for the SUMMARY File.....	528
12.3.40 RPTRST - Define Data to be Written to the RESTART File.....	528
12.3.41 RPTSCHED - Define SCHEDULE Section Reporting.....	529
12.3.42 SAVE - Activates Output of a SAVE File for Fast Restarts.....	531
12.3.43 SCHEDULE - Define the Start of the SCHEDULE Section of Keywords.....	532
12.3.44 SHRATE - Activate and Define the Polymer Shearing Logarithmic Parameters.....	533
12.3.45 SKIPREST - Activate Skipping of Restart Schedule Data.....	534
12.3.46 SUMTHIN - Define SUMMARY DATA Reporting Time Steps.....	535
12.3.47 TSTEP - Advance Simulation by Reporting Time.....	536
12.3.48 TUNING - Numerical Tuning Control.....	538
12.3.49 TUNINGDP - Numerical Tuning Control for High Throughput Cases.....	539
12.3.50 VAPPARS - Oil Vaporization Parameters.....	540
12.3.51 VFPIIJ - Define Injection Vertical Flow Performance Tables.....	541
12.3.52 VFPPROD - Define Production Vertical Flow Performance Tables.....	544
12.3.53 WCONHIST - Define Well Historical Production Rates and Pressures.....	550
12.3.54 WCONINJ - Well Injection Targets and Constraints.....	554
12.3.55 WCONINJE - Well Injection Targets and Constraints.....	555
12.3.56 WCONINJH - Well Historical Observed Injection Rates and Pressures.....	558
12.3.57 WCONPROD - Define Well Production Targets and Constraints.....	561
12.3.58 WDFACCOR - Gas Flow Dependent Skin Factor.....	565
12.3.59 WDRILTIM - Define Drilling Parameters for Automatic Drilling of New Wells.....	566
12.3.60 WECON - Well Economic Criteria for Production Wells.....	567
12.3.61 WEFAC - Define Well Efficiency.....	570
12.3.62 WELCNTL - Modify Well Control and Targets.....	571
12.3.63 WELOPEN - Define Well and Well Connections Flowing Status.....	573
12.3.64 WELSEGS - Define Multi-Segment Wells and Their Segment Structure.....	576
12.3.65 WELSPECS - Define Well Specifications.....	577
12.3.66 WELTARG - Modify Well Targets and Constraints Values.....	581
12.3.67 WGRUPCON - Define Well Guides for Group Control.....	583
12.3.68 WHISTCTL - Define Well Historical Target Phase.....	585
12.3.69 WLIFT - Define Well Re-Tubing, THP and Lift Switching Workover Operations.....	587
12.3.70 WORKLIM - Define Workover Time for a Well.....	588
12.3.71 WPAVE - Define Well Block Average Pressure Calculation Parameters.....	589
12.3.72 WPAVEDEP - Define Well Reference Depth for Pressure Calculations.....	591
12.3.73 WPIMULT - Define Well Connection Multipliers.....	593
12.3.74 WPITAB - Assign Well Productivity Index versus Water Cut Tables.....	595
12.3.75 WPOLYMER - Define Water Injection Well Polymer and Salt Concentrations.....	596
12.3.76 WRFT - Activate Well RFT Reporting to the RFT File.....	598
12.3.77 WRFTPLT - Activate Well RFT and PLT Reporting to the RFT File.....	600
12.3.78 WSEGITER - Define Multi-Segment Wells Iteration Parameters.....	602
12.3.79 WSOLVENT - Define Gas Injection Well Solvent Fraction.....	603
12.3.80 WTEMP - Define An Injection Well's Fluid Temperature.....	604
12.3.81 WTEST - Well Testing Criteria for Re-Opening Closed Wells.....	605
12.3.82 WTRACER - Define An Injection Well's Tracer Concentration.....	607
12.3.83 ZIPPY2 - Activate Automatic Time Step Control.....	608

13 KEYWORD INDEX - ALPHABETIC LISTING.....609

A.....	609
B.....	610
C.....	611
D.....	612
E.....	613
F.....	615
G.....	616
H.....	617
I.....	618

J.....	620
K.....	621
L.....	623
M.....	624
N.....	625
O.....	626
P.....	627
Q.....	629
R.....	630
S.....	631
T.....	635
U.....	636
V.....	637
W.....	638
X.....	640
Y.....	641
Z.....	642
14 OPM FLOW RELEASE HISTORY.....	643
14.1 Release 2017-04.....	643

INDEX OF TABLES

Table 2.1: OPM Flow Command Line Options.....	28
Table 3.1: Example Keyword Table Section.....	30
Table 3.2: ADD Keyword Table Section.....	30
Table 3.3: PORO Keyword Description.....	35
Table 3.4: OPM Flow Input Deck Sections.....	36
Table 4.1: EXTRAPMS Keyword Description.....	42
Table 4.2: INCLUDE Keyword Description.....	43
Table 4.3: MESSAGES Keyword Description.....	45
Table 5.1: ACTDIMS Keyword Description.....	53
Table 5.2: AQDIMS Keyword Description.....	56
Table 5.3: DIMENS Keyword Description.....	58
Table 5.4: ENDSCALE Keyword Description.....	61
Table 5.5: EQLDIMS Keyword Description.....	62
Table 5.6: EQLOPTS Keyword Description.....	63
Table 5.7: FAULTDIM Keyword Description.....	64
Table 5.8: FMTOU Keyword Description.....	66
Table 5.9: FMOUT Keyword Description.....	68
Table 5.10: GRIDOPTS Keyword Description.....	72
Table 5.11: MISCIBLE Keyword Description.....	79
Table 5.12: MSGFILE Keyword Description.....	81
Table 5.13: NSTACK Keyword Description.....	88
Table 5.14: NUMRES Keyword Description.....	89
Table 5.15: NUPCOL Keyword Description.....	90
Table 5.16: OPTIONS Keyword Description.....	92
Table 5.17: PARALLEL Keyword Description.....	93
Table 5.18: PATHS Keyword Description.....	94
Table 5.19: PIMTDIMS Keyword Description.....	96
Table 5.20: REGDIMS Keyword Description.....	98
Table 5.21: ROCKCOMP Keyword Description.....	101
Table 5.22: SATOPTS Keyword Description.....	106
Table 5.23: SATOPTS Relative Permeability Function Allocation Keywords.....	106
Table 5.24: SMRYDIMS Keyword Description.....	109
Table 5.25: START Keyword Description.....	111
Table 5.26: TABDIMS Keyword Description.....	113

Table 5.27: Reservoir Temperature Keywords.....	115
Table 5.28: TITLE Keyword Description.....	116
Table 5.29: TRACERS Keyword Description.....	117
Table 5.30: UNIFIN Keyword Description.....	121
Table 5.31: UNIFOUT Keyword Description.....	123
Table 5.32: VFPIDIMS Keyword Description.....	126
Table 5.33: VFPPDIMS Keyword Description.....	127
Table 5.34: WELLDIMS Keyword Description.....	130
Table 6.1: ACTNUM Keyword Description.....	139
Table 6.2: ADD Keyword Description.....	140
Table 6.3: ADD Keyword Applicable Arrays by Section.....	141
Table 6.4: ADDREG Keyword Description.....	142
Table 6.5: ADDREG Keyword Applicable Arrays by Section.....	143
Table 6.6: BOX Keyword Description.....	148
Table 6.7: COALNUM Keyword Description.....	151
Table 6.8: COORD Keyword Description.....	152
Table 6.9: COPY Keyword Description.....	155
Table 6.10: COPY Keyword Applicable Arrays by Section.....	156
Table 6.11: COPYREG Keyword Description.....	157
Table 6.12: COPYREG Keyword Applicable Arrays by Section.....	158
Table 6.13: DRV Keyword Description.....	159
Table 6.14: DTHETA Keyword Description.....	160
Table 6.15: DX Keyword Description.....	162
Table 6.16: DXV Keyword Description.....	163
Table 6.17: DY Keyword Description.....	164
Table 6.18: DYV Keyword Description.....	165
Table 6.19: DZ Keyword Description.....	166
Table 6.20: DZV Keyword Description.....	167
Table 6.21: EQUALREG Keyword Description.....	169
Table 6.22: EQUALREG Keyword Applicable Arrays by Section.....	170
Table 6.23: EQUALS Keyword Description.....	172
Table 6.24: EQUALS Keyword Applicable Arrays by Section.....	173
Table 6.25: FAULTS Keyword Description.....	174
Table 6.26: FLUXNUM Keyword Description.....	176
Table 6.27: GRIDFILE Keyword Description.....	179

Table 6.28: GRIDUNIT Keyword Description.....	180
Table 6.29: INRAD Keyword Description.....	182
Table 6.30: ISOLNUM Keyword Description.....	183
Table 6.31: JFUNC Keyword Description.....	184
Table 6.32: MAPAXES Keyword Description.....	187
Table 6.33: MAPUNITS Keyword Description.....	188
Table 6.34: MAXVALUE Keyword Description.....	189
Table 6.35: MAXVALUE Keyword Applicable Arrays by Section.....	190
Table 6.36: MINPV Keyword Description.....	191
Table 6.37: MINVALUE Keyword Description.....	192
Table 6.38: MINVALUE Keyword Applicable Arrays by Section.....	193
Table 6.39: MULTFLT Keyword Description.....	194
Table 6.40: MULTIPLY Keyword Description.....	195
Table 6.41: MULTIPLY Keyword Applicable Arrays by Section.....	196
Table 6.42: MULTIREG Keyword Description.....	197
Table 6.43: MULTIREG Keyword Applicable Arrays by Section.....	198
Table 6.44: MULTNUM Keyword Description.....	200
Table 6.45: MULTPV Keyword Description.....	201
Table 6.46: MULTREGP Keyword Description.....	202
Table 6.47: MULTREGT Keyword Description.....	203
Table 6.48: MULTX Keyword Description.....	205
Table 6.49: MULTX- Keyword Description.....	206
Table 6.50: MULTY Keyword Description.....	207
Table 6.51: MULTY- Keyword Description.....	208
Table 6.52: MULTZ Keyword Description.....	209
Table 6.53: MULTZ- Keyword Description.....	210
Table 6.54: NNC Keyword Description.....	213
Table 6.55: NTG Keyword Description.....	215
Table 6.56: OUTRAD Keyword Description.....	219
Table 6.57: PERMX Keyword Description.....	221
Table 6.58: PERMY Keyword Description.....	224
Table 6.59: PERMZ Keyword Description.....	227
Table 6.60: PINCH Keyword Description.....	231
Table 6.61: PINCHNUM Keyword Description.....	232
Table 6.62: PINCHREG Keyword Description.....	234

Table 6.63: PORO Keyword Description.....	235
Table 6.64: RPTGRID Keyword Description.....	238
Table 6.65: SPECGRID Keyword Description.....	239
Table 6.66: SWATINIT Keyword Description.....	240
Table 6.67: THCONR Keyword Description.....	241
Table 6.68: TOPS Keyword Description.....	242
Table 6.69: ZCORN Keyword Description.....	244
Table 7.1: TOPS Keyword Description.....	247
Table 7.2: EDITNNC Keyword Description.....	251
Table 7.3: PORV Keyword Description.....	255
Table 7.4: TRANX Keyword Description.....	256
Table 7.5: TRANY Keyword Description.....	257
Table 7.6: TRANZ Keyword Description.....	258
Table 8.1: Oil-Water Relative Permeability End-Point Data Definitions.....	260
Table 8.2: Gas-Oil Relative Permeability End-Point Data Definitions.....	261
Table 8.3: Saturation Table Formats and Phases.....	262
Table 8.4: ADSALNOD Keyword Description.....	264
Table 8.5: DENSITY Keyword Description.....	271
Table 8.6: EHYSTR Keyword Description.....	275
Table 8.7: GRAVITY Keyword Description.....	281
Table 8.8: IPCG Keyword Description.....	285
Table 8.9: IPCW Keyword Description.....	286
Table 8.10: ISGCR Keyword Description.....	287
Table 8.11: ISGL Keyword Description.....	288
Table 8.12: ISGU Keyword Description.....	289
Table 8.13: ISOGCR Keyword Description.....	290
Table 8.14: ISOGCR Keyword Description.....	291
Table 8.15: ISWCR Keyword Description.....	292
Table 8.16: ISWL Keyword Description.....	293
Table 8.17: ISWU Keyword Description.....	294
Table 8.18: KRG Keyword Description.....	295
Table 8.19: Critical Displacement Relationships.....	296
Table 8.20: KRGR Keyword Description.....	297
Table 8.21: Critical Displacement Relationships.....	298
Table 8.22: KRO Keyword Description.....	299

Table 8.23: Critical Displacement Relationships.....	300
Table 8.24: KRORG Keyword Description.....	301
Table 8.25: Critical Displacement Relationships.....	302
Table 8.26: KRORW Keyword Description.....	303
Table 8.27: Critical Displacement Relationships.....	304
Table 8.28: KRW Keyword Description.....	305
Table 8.29: Critical Displacement Relationships.....	306
Table 8.30: KRWR Keyword Description.....	307
Table 8.31: Critical Displacement Relationships.....	308
Table 8.32: MSFN Keyword Description.....	310
Table 8.33: OILVISCT Keyword Description.....	312
Table 8.34: PCG Keyword Description.....	314
Table 8.35: PCW Keyword Description.....	315
Table 8.36: PLMIXPAR Keyword Description.....	316
Table 8.37: PLYADS Keyword Description.....	317
Table 8.38: PLYADS Keyword Description.....	319
Table 8.39: PLYDHFLF Keyword Description.....	321
Table 8.40: PLYDMAX Keyword Description.....	323
Table 8.41: PLYROCK Keyword Description.....	324
Table 8.42: PLYSHEAR Keyword Description.....	326
Table 8.43: PLYVISC Keyword Description.....	328
Table 8.44: PMISC Keyword Description.....	330
Table 8.45: PVCDO Keyword Description.....	332
Table 8.46: PVDG Keyword Description.....	334
Table 8.47: PVDO Keyword Description.....	336
Table 8.48: PVDS Keyword Description.....	338
Table 8.49: PVTG Keyword Description.....	341
Table 8.50: PVTO Keyword Description.....	343
Table 8.51: PVTW Keyword Description.....	345
Table 8.52: ROCK Keyword Description.....	348
Table 8.53: ROCKOPTS Keyword Description.....	350
Table 8.54: ROCKTAB Keyword Description.....	353
Table 8.55: RPTPROPS Keyword Description.....	355
Table 8.56: RTEMP Keyword Description.....	356
Table 8.57: RTEMPA Keyword Description.....	357

Table 8.58: SALNODE Keyword Description.....	358
Table 8.59: SCALECRS Keyword Description.....	359
Table 8.60: End-Point Arrays Used in the End-Point Scaling Options.....	359
Table 8.61: SDENSITY Keyword Description.....	360
Table 8.62: SGCR Keyword Description.....	361
Table 8.63: SGCWMIS Keyword Description.....	362
Table 8.64: SGFN Keyword Description.....	363
Table 8.65: SGL Keyword Description.....	364
Table 8.66: SGOF Keyword Description.....	365
Table 8.67: SGU Keyword Description.....	367
Table 8.68: SGWFN Keyword Description.....	368
Table 8.69: SLGOF Keyword Description.....	371
Table 8.70: SOF2 Keyword Description.....	373
Table 8.71: SOF3 Keyword Description.....	375
Table 8.72: SOGCR Keyword Description.....	377
Table 8.73: SORWMIS Keyword Description.....	378
Table 8.74: SOGCR Keyword Description.....	379
Table 8.75: SPECHEAT Keyword Description.....	380
Table 8.76: SPECROCK Keyword Description.....	382
Table 8.77: SSFN Keyword Description.....	384
Table 8.78: SWCR Keyword Description.....	386
Table 8.79: SWFN Keyword Description.....	387
Table 8.80: SWL Keyword Description.....	389
Table 8.81: SWOF Keyword Description.....	390
Table 8.82: SWU Keyword Description.....	392
Table 8.83: TLMIXPAR Keyword Description.....	393
Table 8.84: TRACER Keyword Description.....	394
Table 8.85: VISCREF Keyword Description.....	396
Table 8.86: WATVISCT Keyword Description.....	397
Table 9.1: ENDNUM Keyword Description.....	401
Table 9.2: EQLNUM Keyword Description.....	402
Table 9.3: FIPNUM Keyword Description.....	404
Table 9.4: IMBNUM Keyword Description.....	407
Table 9.5: MISNUM Keyword Description.....	408
Table 9.6: PLMIXNUM Keyword Description.....	410

Table 9.7: PVTNUM Keyword Description.....	411
Table 9.8: ROCKNUM Keyword Description.....	414
Table 9.9: RPTREGS Keyword Description.....	415
Table 9.10: SATNUM Keyword Description.....	416
Table 9.11: TNUM Keyword Name Format.....	417
Table 9.12: TNUM Keyword Data Description.....	417
Table 10.1: DATUM Keyword Description.....	423
Table 10.2: DATUMR Keyword Description.....	424
Table 10.3: EQUIL Keyword Description.....	428
Table 10.4: PBUB Keyword Description.....	430
Table 10.5: PBVD Keyword Description.....	431
Table 10.6: PDEW Keyword Description.....	432
Table 10.7: PDVD Keyword Description.....	433
Table 10.8: PRESSURE Keyword Description.....	435
Table 10.9: RESTART Keyword Description.....	436
Table 10.10: RPTRST Keyword Description.....	439
Table 10.11: Data Sets Automatically Written to the RESTART File.....	440
Table 10.12: RPTSOL Keyword Description.....	441
Table 10.13: RS Keyword Description.....	442
Table 10.14: RSVD Keyword Description.....	443
Table 10.15: RTEMPVD Keyword Description.....	445
Table 10.16: RV Keyword Description.....	447
Table 10.17: RVVD Keyword Description.....	448
Table 10.18: SGAS Keyword Description.....	449
Table 10.19: SOIL Keyword Description.....	450
Table 10.20: SSOL Keyword Description.....	452
Table 10.21: STONEIEX Keyword Description.....	454
Table 10.22: SWAT Keyword Description.....	456
Table 10.23: RTEMPVD Keyword Description.....	457
Table 10.24: THPRES Keyword Description.....	459
Table 10.25: VAPPARS Keyword Description.....	462
Table 11.1: SUMTHIN Keyword Description.....	476
Table 12.1: COMPDAT Keyword Description.....	482
Table 12.2: COMPIMB Keyword Description.....	485
Table 12.3: COMPLUMP Keyword Description.....	487

Table 12.4: COMPORD Keyword Description.....	488
Table 12.5: DATES Keyword Description.....	491
Table 12.6: DRSDT Keyword Description.....	494
Table 12.7: DRVDT Keyword Description.....	496
Table 12.8: GCONINJE Keyword Description.....	500
Table 12.9: GCONPROD Keyword Description.....	503
Table 12.10: GECON Keyword Description.....	506
Table 12.11: GEFAC Keyword Description.....	508
Table 12.12: GRUPNET Keyword Description.....	511
Table 12.13: GRUPTREE Keyword Description.....	514
Table 12.14: NEXTSTEP Keyword Description.....	519
Table 12.15: PIMULTAB Keyword Description.....	523
Table 12.16: RPTSCHED Keyword Description.....	529
Table 12.17: TSTEP Keyword Description.....	536
Table 12.18: VFPINJ Keyword Description.....	542
Table 12.19: VFPPROD Keyword Description.....	547
Table 12.20: WCONHIST Keyword Description.....	552
Table 12.21: WCONINJE Keyword Description.....	557
Table 12.22: WCONINJH Keyword Description.....	559
Table 12.23: WCONPROD Keyword Description.....	563
Table 12.24: WECON Keyword Description.....	569
Table 12.25: WELCNTL Keyword Description.....	572
Table 12.26: WELOPEN Keyword Description.....	573
Table 12.27: WELSPECS Keyword Description.....	579
Table 12.28: WELTARG Keyword Description.....	582
Table 12.29: WGRUPCON Keyword Description.....	584
Table 12.30: WHISTCTL Keyword Description.....	585
Table 12.31: WPAVE Keyword Description.....	590
Table 12.32: WPAVDEP Keyword Description.....	591
Table 12.33: WPIMULT Keyword Description.....	593
Table 12.34: WPITAB Keyword Description.....	595
Table 12.35: WPOLYMER Keyword Description.....	596
Table 12.36: WRFT Keyword Description.....	598
Table 12.37: WRFTPLT Keyword Description.....	600
Table 12.38: WSOLVENT Keyword Description.....	603

Table 12.39: WTEST Keyword Description.....	606
---	-----

INDEX OF FIGURES

Figure 6.1: Numerical Model Dimensions.....	132
Figure 6.2: SPE Simulation Case #01 Cartesian Regular Grid.....	133
Figure 6.3: Norne Field Grid Skeleton.....	135
Figure 6.4: Norne Field Corner-Point Geometry Example.....	136
Figure 6.5: Norne Grid Ternary Solution Display.....	137
Figure 8.1: Example Oil-Water Relative Permeability Curves.....	260
Figure 8.2: Example Gas-Oil Relative Permeability Curves.....	261
Figure 12.1: Norne Group Tree Hierarchy Example.....	515

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.

Model formulation:

- black-oil with dissolved gas and vaporized oil
- rock-dependent capillary and relative-permeability curves
- end-point scaling and hysteresis
- oil vaporization controls (VAPPARS)

EOR options:

- Todd-Longstaff type polymer model with adsorption, dead-pore space, permeability reduction, and shear effects ([Flow-polymer](#))
- extra component equation(s), such as a solvent model ([Flow-solvent](#))

Description of geology:

- rectilinear and fully-unstructured grid
- corner-point grids from Eclipse input, including fault and region multipliers, minpv/pinch, etc

Wells and controls:

- bottom-hole pressure and surface/reservoir rate
- group controls
- shut/stop/open individual completions
- history-matching wells

Input/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)

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 FLOW

If OPM Flow is installed on your system typing the following command:

```
flow CASENAME
```

from your terminal 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
```

By default output files are generated in the same folder as the .DATA file.

The user can supply various options to the simulator. For instance

```
flow CASENAME output_dir=foo
```

will send the output files to the foo directory. A list of command line options is given in the following section. Note that there must be no spaces around the equals sign.

If OPM Flow is installed with parallel capabilities.

```
mpirun -np 4 flow CASENAME
```

will start Flow on four nodes etc.

Additional tutorials for running OPM Flow is available on OPM website in the [Tutorials](#) section.

2.2.1 COMMAND LINE OPTIONS

OPM Flow accepts command line options to control various aspects of the simulator's 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.

No.	Variable Name	Description	Default
1	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
2	output	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: <code>output=log</code> or <code>output=false</code>	all
3	output_dir	Set the directory to which output files are written.	deck location

No.	Variable Name	Description	Default
4	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
5	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
6	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
7	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
8	linear_solver_maxiter	A positive integer value that defines the maximum number of linear iterations.	150
9	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
10	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
11	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
12	ilu_relaxation	A real positive double precision value that sets the relaxation parameter for the ilu pre-conditioner.	0.9
13	ilu_fillin_level	A positive integer value that sets the fill in level for the ilu pre-conditioner.	0
14	dp_max_rel	A real positive double precision value that sets the maximum allowed relative change of pressure per iteration.	1
15	ds_max	A real positive double precision value that sets the maximum allowed change in saturation per iteration.	0.2
16	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
17	dbhp_max_rel	A real positive double precision value that sets the maximum allowed relative change in BHP per iteration.	1

No.	Variable Name	Description	Default
18	dwel_fraction_max	A real positive double precision value that sets the maximum allowed change in well component fraction per iteration.	1e+07
19	tolerance_mb	A real positive double precision value that sets the maximum mass balance error.	1e-05
20	tolerance_cnv	A real positive double precision value that sets the maximum non-linear tolerance error.	0.01
21	tolerance_wells	A real positive double precision value that sets the maximum non-linear error for well equations.	0.0001
22	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
23	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
24	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
25	relax_max	A real positive double precision value that is used to tune the stabilized Newton option.	0.5
26	relax_type	A character string that sets relaxation type of the stabilized Newton option.	dampen
27	max_iter	A positive integer that sets the maximum number of non-linear iterations.	10
28	min_iter	A positive integer that sets the minimum number of non-linear iterations.	1
29	output_terminal	A Boolean value set to true or false that turns on (true) or off (false) output to terminal.	true
30	use_TUNING	A Boolean value set to true or false that that instructs OPM Flow to read the time stepping parameters from the <i>TUNING - Numerical Tuning Control</i> keyword in the input deck, if set to true. Note that only the first record of the TUNING keyword is processed.	false
31	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

No.	Variable Name	Description	Default
32	<code>solver.restartfactor</code>	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 <code>solver.restartfactor</code> 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
33	<code>timestep.control.maxgrowth</code>	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 <code>timestep.max_timestep_in_days</code> parameter. For example, if the current time step has converged at 10 days and <code>solver.restartfactor</code> is set to the default value, then the next time step will be $0.3 * 10$ days, that is at 30 days.	3
34	<code>timestep.max_timestep_in_days</code>	A real positive double precision value that sets the maximum allowed time step size in days.	365
35	<code>solver.restart</code>	A positive integer that defines the number of allowed consecutive restarts (or time step chops) before the simulation is terminated.	10
36	<code>solver.verbose</code>	A Boolean value set to true or false that switches on (true) or off (false) solver specific output.	true
37	<code>timestep.verbose</code>	A Boolean value set to true or false that switches on (true) or off (false) time step specific output.	true
38	<code>timestep.initial_timestep</code>	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 <code>solver.restartfactor</code> * the length of the first report step.	-1
39	<code>full_timestep_initially</code>	Try to use the report steps as time steps.	false
40	<code>timestep.timestep_in_days_after_event</code>	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
41	<code>timestep.control</code>	A character string that defines the time stepping control and is set to one of the following: 1) <code>pid</code> : PID controller based adaptive time step control as suggested by Turek and Kuzmin ¹ . 2) <code>pid+iteration</code> : Use PID and linear iteration numbers to guide the time step. 3) <code>pid+newtoniteration</code> : Use PID and non-linear iterations numbers to guide the time step. 4) <code>Hardcoded</code> : Use time steps supplied by user. Via <code>timestep.control.filename</code>	pid

¹ Algebraic Flux Correction III. Incompressible Flow Problems. Uni Dortmund, Turek and Kuzmin, January, 2006 (DOI: 10.1007/3-540-27206-2_8)

No.	Variable Name	Description	Default
42	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 <i>timestep.control</i> option).	0.1
43	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 <i>timestep.control</i> option.	8
44	timestep.control.filename	A character string that defines a filename where time steps are specified. For instance generated by the ecl_summary application in libecl as per the following UNIX command line: <code>path_to_libecl_applications/ecl_summary DECK TIME > filename</code> 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 2.1: OPM Flow Command Line Options

As mentioned previously, it is also possible to put multiple options together in a parameter file, by using a filename ending with the extension “.param” and then passing that filename as a command line parameter to OPM Flow.

Note

As the run parameters will effect the results and run times of a simulation it may be worth considering have a parameter file for each run of the form CASENAME.param. This can be then be used to audit any runs when evaluating the results and run times.

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.

Example

```
// -----
// 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 Parmaters  
//  
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 Fails 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  
// -----
```

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, or only part of the keyword's function is implemented. 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, 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 for the cells colored in light gray.

A complete list of keywords in alphabetic order is given in [SECTION 13 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 EQUIL 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 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 for 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.			Default
		psia 1.032	barsa 1.032	atma 1.032	
2	Cf	Cf is a real number defining the rock compressibility at the reference pressure, Cf(Pref) and is defined as: $C_f = -\frac{1}{V}\left(\frac{dV}{dP}\right)$			Defined
		l/psia 0.0	l/barsa 0.0	l/atma 0.0	

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 stating from 0 and terminating at 1.0, that defines the water saturation.			None
		dimensionless	dimensionless	dimensionless	
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 0.0 and less than or equal to 1.0 that defines the water relative permeability with respect to gas saturation. The first value in the column should be zero.			None
		dimensionless	dimensionless	dimensionless	
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.			None
		psia	bars	atm	

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 "I*" 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 for all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
I	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:

- I) 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 100² simulator. Table 3.4 lists the various section together with a brief description of purpose of the section. As well 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 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: <ol style="list-style-type: none"> 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

² 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 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 it is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness.

4.1.2 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.

4.1.3 END – DEFINE THE END OF THE INPUT FILE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

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

There is no data required for this keyword.

Example

```

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

DATES
1 JAN 2006 /
/

RPTSCHED
'NOTHING'      /

DATES
1 APR 2006 /
1 JUL 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.

4.1.4 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 JUL 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.

4.1.5 ENDSKIP – DEACTIVATES 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 examples skips reading of the of the grid geometry data using the INCLUDE keyword, and then reverts back to reading the input files again.

4.1.6 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
I	EXTRAP	<p>Defines a single integer that activates the extrapolation warning message options for PVT and VFP tables. EXTRAP can have the following values:</p> <ul style="list-style-type: none"> 0) 0 – No warning messages are give (the default). 1) 1 – PVT table extrapolation warnings are printed. 2) 2 – VFP table extrapolation warnings are printed. 3) 3 – PVT and VFP table extrapolation warnings are printed. 4) 4 - PVT and VFP table extrapolation warnings are printed with additional information. 	0
<p>Notes:</p> <ul style="list-style-type: none"> 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.

4.1.7 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 is encountered, 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
I	FILEINC	A character string enclosed in quotes that defines a file to read in and be processed by OPM Flow.	None
Notes: I) The keyword is terminated by “/”.			

Table 4.2: 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 load the data one directory above from where the data file is located.

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

4.1.8 MESSAGES – DEFINE MESSAGE PRINT LIMITS AND STOP LIMITS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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 programming errors, as outlined in Table 4.3.

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 PROBLRM 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: <ol style="list-style-type: none"> 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 4.3: MESSAGES Keyword Description

Examples

```
--      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.

4.1.9 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.

4.1.10 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.

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

4.1.11 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.

4.1.12 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.

4.1.13 SKIP300 – ACTIVATE SKIPPING OF “COMPOSITIONAL” KEYWORDS AND INPUT DATE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

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

There is no data required for this keyword.

This keyword is 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
--
SKIP100
--
--      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.

4.1.14 WARN – ACTIVATE WARNING MESSAGES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

Turns on warning messages to be printed to the print file; 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 subsequent runs, the warning messages can be turned off.

There is no data required for this keyword.

OPM Flow does not recognize this keyword and the keyword may cause the program to terminate, but it 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

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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 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 condition statements in the ACTION statement	3
Notes: 1) The keyword is terminated by “/”.			

Table 5.1: ACTDIMS Keyword Description

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

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 AITS - INTELLIGENT TIME STEPPING ACTIVATION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

Turns on ECLIPSES intelligent time stepping.

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

5.2.3 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.

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

Example

```
--
--      ACTVATE THE API TRACKING OPTION
--
API
```

The above example switches on the API tracking facility.

5.2.4 AQUDIMS – AQUIFER DIMENSIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

The AQDDIMS 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.	36
5	NANAQ	A positive integer value that defines the AQUFET and AQUTAB 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 5.2: AQDIMS Keyword Description

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

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.

5.2.5 CPR – ACTIVATE CONSTRAINED PRESSURE RESIDUAL (“CPR”) LINEAR SOLVER

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

Turns on the CPR linear solver. There is no data required for this keyword.

OPM Flow does not recognize this keyword.

Example

```
--      ACTIVATE CONSTRAINED PRESSURE RESIDUAL LINEAR SOLVER FOR THE RUN  
CPR
```

The above example activates Constrained Pressure Residual (“CPR”) Linear Solver for the run.

5.2.6 DIMENS – DEFINE THE DIMENSION OF THE MODEL

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
----------------	------	------	-------	---------	----------	---------	----------

Description

DIMENS defines the dimensions of the model entered as integer vector. The keyword can be used for 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 5.3: 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 46 cells in the x direction, 112 cells in the y direction and 22 cells in the z direction.

5.2.7 DISGAS – DEFINE DISSOLVED GAS TO BE PRESENT IN THE MODEL

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

This keyword indicates that dissolved gas is present in live³ oil in the model and the keyword should only be used if 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 oil⁴, 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.

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.

³ “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.

⁴ “Dead” oil is oil that it contains no dissolved gas or a relatively thick oil or residue that has lost its volatile components.

5.2.8 ENDSALE – ACTIVATE RELATIVE PERMEABILITY END-POINT SCALING OPTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
----------------	------	------	-------	---------	----------	---------	----------

Description

The ENDSALE 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 endpoint 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-, SWLY-, SWLY-SWLZ and SWLZ-, instead of SWL or the SWLX, SWLY and SWLZ set of keywords.

The keyword also defines the number of saturation end-point tables that allows for the re-scaling of the saturation functions to be a function of depth as oppose to being a grid property array. This is accomplished via the ENKRVD and ENPVT keywords in the PROPS section.

No.	Name	Description	Default
I	DIRECT	<p>A character string that activates or deactivates directional end-point scaling option.</p> <p>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.</p> <p>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.</p> <p>Only the default option is supported by OPM Flow.</p>	NODIR

No.	Name	Description	Default
2	IRREVERS	<p>A character string that activates or deactivates non-reversible end-point scaling option.</p> <p>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 x_i to x_{i+1} direction as for the flow from the x_i to the x_{i-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.</p> <p>Alternatively, if IRREVERS is set to IRREVERS then the end-point scaling is set to non-reversible and results in different sets of end-point arrays being applied for flow from the x_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 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.</p> <p>Only the default option is supported by OPM Flow.</p>	REVERS
3	NTENDP	<p>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.</p> <p>Only the default option is supported by OPM Flow.</p>	1
4	NNODES	<p>A positive integer the defines the maximum number entries for saturation end-point depth tables.</p> <p>Only the default option is supported by OPM Flow.</p>	20
5	MODE	<p>A positive integer that activates the options for temperature dependent saturation end-point scaling.</p> <p>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.</p>	0
<p>Notes:</p> <ol style="list-style-type: none"> 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 5.4: 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.

5.2.9 EQLDIMS – DEFINE THE EQUILIBRATION DATA DIMENSIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

The EQLDIMS keyword defines the maximum number of properties associated with equilibrating the model, that is initializing the model. A reservoir grid can be separated into separate regions in order to apply different pressure regimes and/or fluid contacts. Care should be taken that the different regions are not in communication if the pressures or fluid contacts are different for the various regions, as this would lead to an unstable initialization and would also imply errors in the model description as implemented.

No.	Name	Description	Default
1	NTEQUL	A positive integer value that defines the number of equilibration regions entered using the EQLNUM keyword in the REGIONS section and the number of entries associated with the EQUIL keyword in the SOLUTION section.	1
2	NPRSVD	A positive integer value setting the number of pressure versus depth entries used by OPM Flow in determining equilibration parameters. Unless there is a requirement for a very fine equilibration this parameter should be defaulted.	100
3	NDRXVD	A positive integer value that defines the maximum number of depth entries in equilibration property versus depth tables (RSVD, RVVD, PBVD or PDVD etc.) as defined in the SOLUTION section.	20
4	NTTRVD	A positive integer that defines the maximum number of TVDP tables that describe the initial tracer concentration versus depth. 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 5.5: 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.

5.2.10 EQLOPTS – ACTIVATES THE EQUILIBRATION OPTIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
----------------	------	------	-------	---------	----------	---------	----------

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 I* 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	THPRES	A character string that activates the inter-region equilibration flow option. This option allows for a threshold pressure variable entered via the THPRES keyword to define a pressure which prevents flow between regions until the THPRES value between regions is exceeded. If the THPRES command is stated then this option is activated.	None
4	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 I* on the keyword.	None
Notes: 1) The keyword is terminated by “/”.			

Table 5.6: EQLOPTS Keyword Description

Examples

```
--      ACTIVATE EQUILIBRATION OPTIONS
--      MOBILE ENDPOINT(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.

5.2.11 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
I	MFSEGS	A positive integer value that defines the maximum number of records (segments) for the FAULTS keyword.	0
Notes: I) The keyword is terminated by "/".			

Table 5.7: 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 10,000 segments.

5.2.12 FIELD – ACTIVATES THE OIL FIELD SYSTEM OF UNITS FOR THE MODEL

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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.

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.

5.2.13 FMTIN – ACTIVATES THE FORMAT INPUT FILE OPTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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 5.8.

Process	Option	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. <u>If the keyword is omitted then the default is for binary file input.</u>	*.SMSPEC *.SUMMARY *.RSSPEC *.RESTART
	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. <u>If the keyword is omitted then the default is for one file per report time step.</u>	*.SMSPEC *.SUMMARY *.RSSPEC *.RESTART
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. <u>If the keyword is omitted then the default is for binary file input.</u>	*.SMSPEC *.SUMMARY *.GRID *.INIT *.RSSPEC *.RESTART
	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.. <u>If the keyword is omitted then the default is for one file per report time step input.</u>	*.SMSPEC *.SUMMARY *.RSSPEC *.RESTART

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 5.8: FMTOUT Keyword Description

There is no data required for this keyword.

Example

```
--          SWITCH ON THE FORMAT INPUT FILES OPTION
FMTIN
```

The above example switches on the format input file option.

5.2.14 FMTOUT – ACTIVATES THE FORMAT OUTPUT FILE OPTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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 5.9.

Process	Option	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. <u>If the keyword is omitted then the default is for binary file input.</u>	*.SMSPEC *.SUMMARY *.RSSPEC *.RESTART
	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. <u>If the keyword is omitted then the default is for one file per report time step.</u>	*.SMSPEC *.SUMMARY *.RSSPEC *.RESTART
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. <u>If the keyword is omitted then the default is for binary file input.</u>	*.SMSPEC *.SUMMARY *.GRID *.INIT *.RSSPEC *.RESTART
	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. <u>If the keyword is omitted then the default is for one file per report time step input.</u>	*.SMSPEC *.SUMMARY *.RSSPEC *.RESTART

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 5.9: FMOUT Keyword Description

There is no data required for this keyword.

Example

```
--          SWITCH ON THE FORMAT OUTPUT FILES OPTION
FMTOUT
```

The above example switches on the format output file option.

5.2.15 FULLIMP – ACTIVATES FULLY IMPLICIT SOLUTION OPTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

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.

There is no data required for this keyword.

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.

5.2.16 GAS – DEFINE THE GAS PHASE TO BE PRESENT 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.

Example

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

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

5.2.17 GRIDOPTS - GRID PROCESSING OPTIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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	<p>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.</p> <p>OPM Flow uses a positive directional dependent transmissibility formulation to describe the flow between two cells, that is for cell (I, J, K) OPM Flow calculates the x face transmissibility between (I, J, K) and (I + 1, J, K) cell face. Modification to the transmissibilities in this case is accomplished by the MULTX, MULTY and MULTZ. Keywords.</p> <p>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.</p> <p>This option is not supported and should be defaulted with I* or NO on the keyword.</p>	NO
2	NRMULT	<p>A positive integer value that defines the maximum number of MULTNUM regions for the MULTNUM array.</p> <p>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.</p> <p>Inter-region MULTNUM transmissibility multipliers can be defined using the MULTREGT and regional pore volumes multipliers can be set using the MULTREGP keyword.</p>	0
3	NRPINC	<p>A positive integer value that defines the maximum number of PINCHNUM regions for the PINCHNUM array.</p> <p>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.</p> <p>Each regions criteria for setting the pinch out criteria is set by the PINCHREG keyword.</p>	0

Table 5.10: 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.

5.2.18 IMPES – ACTIVATES IMPLICIT PRESSURE EXPLICIT SATURATION SOLUTION OPTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

IMPES keyword activates on Implicit Pressure Explicit Saturation formulation and solution options, commonly known as IMPES.

OPM Flow uses a different numerical scheme which makes this keyword redundant; hence, OPM Flow ignores this keyword. It is documented here for completeness.

There is no data required for this keyword.

Example

```
--
--      ACTIVATES THE IMPES SOLUTION OPTION
--
IMPES
```

The above example switches on the fully implicit solution option; however, this has no effect in OPM Flow input decks.

5.2.19 LAB - ACTIVATES THE LABORATORY SYSTEM OF UNITS FOR THE MODEL

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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.

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.

5.2.20 LGR – DEFINE LOCAL GRID REFINEMENT PARAMETERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

This keyword defines various parameters for the local grid refinement option.

Currently, OPM Flow does not support the local grid refinement feature.

5.2.21 MEMORY – DEFINE ALLOCATED MEMORY

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

This keyword defines the memory allocation for the run.

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

5.2.22 METRIC – ACTIVATES THE METRIC SYSTEM OF UNITS FOR THE MODEL

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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.

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.

5.2.23 MISCIBLE – DEFINE MISCIBILITY TODD-LONGSTAFF PARAMETERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
----------------	------	------	-------	---------	----------	---------	----------

Description

The MISCIBLE keyword defines the options associated with the Todd-Longstaff⁵ mixing parameters used for when polymer flooding or CO₂ EOR simulation cases are being run

No.	Name	Description	Default
1	NTMISC	A positive integer value that declares the number miscible residual oil saturations versus water saturations tables for SORWMIS keyword and the number Todd-Longstaff mixing parameters entries on the TLMIXPAR keyword.	1
2	NSMISC	A positive integer value that sets the maximum number of entries (or rows) for each SORWMIS table defined by the SORWMIS keyword.	20
3	MISOPT	A character string that defines the numerical dispersion control options for the oil and gas relative permeability curves, set to either NONE or TWOPOINT: 1) NONE – standard single point up streaming, that is using the immediate neighbor 2) TWOPOINT – two-point up streaming, that is using the immediate neighbor plus one cell for better numerical dispersion control but with a higher computational cost. <u>Only the default value of NONE is supported.</u>	NONE
Notes: 1) The keyword is terminated by “/”.			

Table 5.11: 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.

⁵ 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.

5.2.24 MONITOR – ACTIVATES OUTPUT OF THE MONITORING DATA AND FILE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

The MONITOR keyword activates the writing out of the run time monitoring information used by post-processing graphics software to display run time information, for example the simulated production and injection rates and cumulative values. OPM Flow does not have this functionality.

Hence, OPM Flow ignores this keyword but it is documented here for completeness.

There is no data required for this keyword.

Example

```
--
--      ACTIVATES 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.

5.2.25 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
I	MSGOPT	A positive integer set to 0 for to deactivate message file output or 1 to activate message file output.	1
Notes: I) The keyword is terminated by “/”.			

Table 5.12: MSGFILE Keyword Description

Example

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

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

5.2.26 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.

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.

5.2.27 NOINSPEC – DEACTIVATE OUTPUT OF THE INIT INDEX FILE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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 i(*.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.

Example

```
--
--      DEACTIVATES 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.

5.2.28 NOMONITO – DEACTIVATES OUTPUT OF THE MONITORING DATA AND FILE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

The NOMONITO keyword deactivates the writing out of the run time monitoring information used by post-processing graphics software to display run time information, for example the simulated production and injection rates and cumulative values. OPM Flow does not have this functionality.

Hence, OPM Flow ignores this keyword but it is documented here for completeness.

There is no data required for this keyword.

Example

```
--
--      DEACTIVATES 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.

5.2.29 NONNC – DEACTIVATES 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.

Example

```
--
--      DEACTIVATES NON-NEIGHBOR CONNECTIONS
--
NONNC
```

The above example switches off the NNCs; however, this has no effect in OPM Flow input decks.

5.2.30 NORSSPEC – DEACTIVATE OUTPUT OF THE RESTART INDEX FILE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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) is 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.

Example

```
--
--      DEACTIVATES 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.

5.2.31 NOSIM – ACTIVATES 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.

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.

5.2.32 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; hence, OPM Flow ignores this keyword. It 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 5.13: 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 endpoint scaling, etc., then resolve these messages first before adjusting the numerical controls.

5.2.33 NUMRES – DEFINE THE NUMBER OF RESERVOIR GRIDS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

The NUMRES keyword defines the number of reservoir grids (COORD data sets) that the simulator should process. Currently, this should only be set to one in OPM Flow

No.	Name	Description	Default
I	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.	I
Notes: I) The keyword is terminated by “/”.			

Table 5.14: 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.

5.2.34 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.

OPM Flow uses a different numerical scheme which makes this keyword redundant; hence, OPM Flow ignores this keyword. It is documented here for completeness. See section [2.2 Running Flow](#) on how to set the numerical control parameters for OPM Flow.

No.	Name	Description	Default
I	NUPCOL	A positive integer that defines the maximum number of Newtonian iterations used to update well targets within a time step.	3
Notes: I) The keyword is terminated by “/”.			

Table 5.15: NUPCOL Keyword Description

Example

```
--
--      DEFINE THE NUMBER OF ITERATIONS TO UPDATE WELL FLOW TARGETS
--
NUPCOL      3
```

The above example sets the default NUPCOL value; however, this has no effect in OPM Flow input decks.

5.2.35 OIL – DEFINE THE OIL PHASE TO BE PRESENT IN THE MODEL

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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.

Example

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

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

5.2.36 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.

No.	Description	Default
I - 273	Undefined.	0
Notes: I) The keyword is terminated by “/”.		

Table 5.16: OPTIONS Keyword Description

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

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.

5.2.37 PARALLEL – DEFINE 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 on how to run OPM Flow in parallel.

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 5.17: PARALLEL Keyword Description

OPM Flow uses a different numerical scheme which makes this keyword redundant; hence, OPM Flow ignores this keyword. It is documented here for completeness.

See section [2.2 Running Flow](#) on how to run OPM Flow in parallel mode.

Example

```
--
--      PARALLEL MULTI-CORE OPTIONS
--      NDMAIN      MACHINE TYPE
PARALLEL      2      DISTRIBUTED      /
```

The above example sets the number of domains (or processor)s to two and for the simulation to run in parallel mode. this has no effect in OPM Flow input decks.

5.2.38 PATHS – DEFINE FILENAME DIRECTORY PATH ALIASES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

PATHS allows the user to define alias directory filenames to avoid long filenames with the INCLUDE, IMPORT, RESTART or GDFILE. To use the alias, should be prefixed with the \$ symbol.

OPM Flow currently does not support this feature and ignores this keyword. It is documented here for completeness.

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 5.18: 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.

5.2.39 POLYMER – DEFINE THE POLYMER PHASE TO BE PRESENT IN THE MODEL

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

This keyword indicate that the polymer phase is present in the model and to activate the polymer flooding mode. | 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.

Example

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

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

5.2.40 PIMTDIMS – DEFINE WELL PRODUCTIVITY SCALING TABLE DIMENSIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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 5.19: 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.

5.2.41 RADIAL – RADIAL GRID ACTIVATION OPTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

RADIAL⁶ activates the radial grid geometry option for the model, if this keyword is omitted then Cartesian geometry is assumed by OPM Flow.

⁶ Radial grids are not currently implemented in OPM Flow version 2017-10 but is expected to be incorporated in the next release.

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

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

The REGDIMS keyword defines the maximum number of regions for various region arrays used in the model. Note that the maximum number of FIPNUM regions can be defined both on this keyword and the TABDIMS keyword, if it set in both locations the maximum value is used. The reason for this type of inconsistency is due to the commercial simulator evolving with time as new features were added, but at the same time having to maintain backward input deck compatibility.

No.	Name	Description	Default
1	NTFIP	A positive integer defining the maximum number of regions in the FIPNUM region array. 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 5.20: 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.

5.2.43 ROCKCOMP – ACTIVATE ROCK COMPACTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

The ROCKCOMP keyword activates rock compaction and defines various rock compaction options for the run. By default OPM Flow models rock compaction via pore volume compressibility as entered on the ROCK keyword in the PROPS section. This keyword enables pressure dependent pore volume and transmissibility multipliers for rock compaction that are entered in the PROPS section using the ROCKTAB keyword. Currently OPM Flow only supports the default options for rock compaction.

No.	Name	Description	Default
1	ROCKOPT	A character string that defines the rock compaction option based on one of the following character strings: <ol style="list-style-type: none"> 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 formulation⁷. 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-Mansoori⁸ 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. 	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).	NO

⁷ 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.

⁸ 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	<p>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:</p> <p>1) EXP: An exponential porosity-transmissibility relationship should be used.</p> <p>2) CZ: The Carmen-Kozeny⁹, ¹⁰ and ¹¹ porosity-transmissibility relationship should be used.</p> <p>This option is currently ignored by OPM Flow.</p>	1*
Notes: 1) The keyword is terminated by "/".			

Table 5.21: 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.

⁹ J. Kozeny, "Ueber kapillare Leitung des Wassers im Boden." Sitzungsber Akad. Wiss., Wien, 136(2a): 271-306, 1927.

¹⁰ P.C. Carman, "Fluid flow through granular beds." Transactions, Institution of Chemical Engineers, London, 15: 150-166, 1937.

¹¹ P.C. Carman, "Flow of gases through porous media." Butterworths, London, 1956

5.2.44 RPTRUNSP – ACTIVATES RUNSPEC REPORTING

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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

```
--
--      SWITCH ON RUNSPEC SECTION REPORTING
--
RPTRUNSP
```

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

5.2.45 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 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.

5.2.46 SATOPTS – ACTIVATE RELATIVE PERMEABILITY ASSIGNMENT OPTIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

SATOPTS keyword activates OPM Flow's relative permeability assignment options. The relative permeability functions are defined using the either the:

- 1) SWOF, SGOF, SLGOF series of saturation functions, or the
- 2) SWFN, SGFN, SGWFN, SOF2, SOF3, SOF32D series of functions.

The allocation of the relative permeability tables to the grid cells is dependent on the options selected on this keyword (SATOPTS).

If the DIRECT option as been activated and the IRREVERS **has not been invoked** on the SATOPTS keyword, then **different relative permeability functions are used for each x, y, and z directions**. Here the KRNUMX, KRNUMY and KRNUMZ keywords are used for Cartesian grids to allocate the relative permeability tables to the cells. For Radial grids the KRNUMR, KRNUMT and KRNUMZ keywords should be used. This results in the same relative permeability curves being used in both the 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 Carlson¹² is used to describe relative permeability hysteresis and the Killough¹³ 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 5.23 for the various relative permeability table allocation keywords for the various combination of DIRECT, IRREVERS and HYSTER command options.

¹² Carlson, F. M. (1981) SPE 10157, presented at the 56th Annual SPE Fall Meeting, San Antonio, 1981

¹³ Killough, J. E. "Reservoir Simulation with History-dependent Saturation Functions," paper SPE 5106, Society of Petroleum Engineers Journal (1976) 16, No. 1, 37-48.

The keyword should be followed by one or more of the following keyword options.

No.	Name	Description	Default
1	DIRECT	<p>A character string that activates the directional relative permeability assignment option.</p> <p>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.</p> <p>Only the default option is supported by OPM Flow.</p>	None
2	IRREVERS	<p>A character string the activates reversible directional relative permeability assignment option.</p> <p>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).</p> <p>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.</p> <p>Only the default option is supported by OPM Flow.</p>	None
3	HYSTER	<p>A character string that activates the hysteresis option.</p> <p>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, KRNUMY 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.</p> <p>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.</p> <p>Only the default option is supported by OPM Flow.</p>	None
4	SURFTENSA	<p>A character string that activates the capillary pressure surface tension pressure dependency option.</p> <p>Only the default option is supported by OPM Flow.</p>	None

No.	Name	Description	Default
Notes: <ol style="list-style-type: none"> 1) Note that the IRREVERS command can only been activated if the DIRECT command is activated at the same time. 2) See Table 5.23 for the various relative permeability table allocation keywords. 3) The keyword is terminated by “/”. 			

Table 5.22: 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: <ol style="list-style-type: none"> 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 5.23: SATOPTS Relative Permeability Function Allocation Keywords.

Examples

The first example activates 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
--      DIRECTIONAL(DIRECT) IRREVERSIBLE(IRREVERS) HYSTERESIS(HYSTER)
SATOPTS      'DIRECT' /
```

The next example activates 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
--      DIRECTIONAL(DIRECT) IRREVERSIBLE(IRREVERS) HYSTERESIS(HYSTER)
SATOPTS      'DIRECT' 'IRREVERS' /
```

Finally, the last option invokes all three assignment options.

```
--
--      ACTIVATE RELATIVE PERMEABILITY ASSIGNMENT HYSTERESIS OPTIONS
--      DIRECTIONAL(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 ENDSIZE 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.

5.2.47 SAVE – ACTIVATES 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.

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

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

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

OPM Flow 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
I	NSUMMX	A positive integer that defines the maximum number of summary vectors to be written out to the SUMMARY file (*.SUMMARY).	10000
Notes: I) The keyword is terminated by “/”.			

Table 5.24: 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.

5.2.49 SOLVENT – DEFINE THE SOLVENT PHASE TO BE PRESENT IN THE MODEL

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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.

Example

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

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

5.2.50 START – SIMULATION START DATE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

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 5.25: 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.

5.2.51 TABDIMS – DEFINE THE NUMBER OF TABLES AND THE TABLE DIMENSIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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 not been activated then this parameter is ignored.	20
7	NRVPVT	A positive integer that defines the maximum number of Rv entries in the PVT tables for the 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 ENDSKALE keyword, and if specified on both here and on the ENDSKALE 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.	I*
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	I*
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	I*

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 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 5.26: 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 of 40, and the maximum number of PVT tables to be nine with a maximum number of rows of 30.

5.2.52 TEMP – ACTIVATE THE TEMPERATURE MODELING OPTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
----------------	------	------	-------	---------	----------	---------	----------

Description

This keyword activates the temperature modeling option. There is no data required for this keyword.

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

Table 5.27: Reservoir Temperature Keywords

The specific heat of the fluids and rock are given by the SPECHEAT and SPECROCK keywords in the PROPS section. Oil and water viscosity and temperature relationships are entered via the OILVISCT and WATVISCT keywords in the PROPS section and the reference conditions for these tables is given by the VISCREF keyword, which is also defined 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

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

The above example activates the temperature modeling option.

5.2.53 TITLE – DEFINE THE TITLE FOR THE INPUT DECK

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

The TITLE keyword defines the title for the input deck. The title text will be printed on all reports so as to act as a reference for the run.

No.	Name	Description	Default
I	TITLE	A character string that defines the TITLE for the input deck	None
Notes: <ol style="list-style-type: none"> 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 5.28: TITLE Keyword Description

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

Note

It is good practice to include the name of the input file in the title (without the extension) for when cross checking results from multiple cases.

Examples

```
--
--      DEFINE THE TITLE FOR THE RUN
TITLE
CASE OPM-SPE01 - SIMULATION RUN TITLE
```

The above example defines the title for the run to be "CASE OPM-SPE01 - SIMULATION RUN TITLE".

5.2.54 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.

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 5.29: TRACERS Keyword Description

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

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.

5.2.55 UDADIMS – DEFINE THE DIMENSIONS OF THE USER DEFINED ARGUMENTS FEATURE

<i>RUNSPEC</i>	<i>GRID</i>	<i>EDIT</i>	<i>PROPS</i>	<i>REGIONS</i>	<i>SOLUTION</i>	<i>SUMMARY</i>	<i>SCHEDULE</i>
----------------	-------------	-------------	--------------	----------------	-----------------	----------------	-----------------

Description

This keyword defines the dimensions of the user defined arguments used in the commercial simulator to calculate various user defined quantities in the SCHEDULE section, this feature is not supported by OPM Flow.

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

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

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

This keyword defines the dimensions associated with the UDQ keyword used in the commercial simulator to calculate various user defined quantities in the SCHEDULE section, this feature is not supported by OPM Flow.

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

5.2.57 UNIFIN – ACTIVATES THE UNIFIED INPUT FILE OPTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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 5.30.

Process	Option	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. <u>If the keyword is omitted then the default is for binary file input.</u>	*.SMSPEC *.SUMMARY *.RSSPEC *.RESTART
	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. <u>If the keyword is omitted then the default is for one file per report time step.</u>	*.SMSPEC *.SUMMARY *.RSSPEC *.RESTART
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. <u>If the keyword is omitted then the default is for binary file input.</u>	*.SMSPEC *.SUMMARY *.GRID *.INIT *.RSSPEC *.RESTART
	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. <u>If the keyword is omitted then the default is for one file per report time step input.</u>	*.SMSPEC *.SUMMARY *.RSSPEC *.RESTART

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 5.30: UNIFIN Keyword Description

There is no data required for this keyword.

Example

```
--  
--      SWITCH ON THE UNIFIED INPUT FILES OPTION  
--  
UNIFIN
```

The above example switches on the unified input file option.

5.2.58 UNIFOUT – ACTIVATES THE UNIFIED OUTPUT FILE OPTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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 5.31.

Process	Option	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. <u>If the keyword is omitted then the default is for binary file input.</u>	*.SMSPEC *.SUMMARY *.RSSPEC *.RESTART
	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. <u>If the keyword is omitted then the default is for one file per report time step.</u>	*.SMSPEC *.SUMMARY *.RSSPEC *.RESTART
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. <u>If the keyword is omitted then the default is for binary file input.</u>	*.SMSPEC *.SUMMARY *.GRID *.INIT *.RSSPEC *.RESTART
	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. <u>If the keyword is omitted then the default is for one file per report time step input.</u>	*.SMSPEC *.SUMMARY *.RSSPEC *.RESTART

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 5.31: UNIFOUT Keyword Description

There is no data required for this keyword.

Example

```
--  
--      SWITCH ON THE UNIFIED OUTPUT FILES OPTION  
--  
UNIFOUT
```

The above example switches on the unified output file option.

5.2.59 VAPOIL – DEFINE VAPORIZE OIL IN WET GAS TO BE PRESENT 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 wet¹⁴ gas in the model and the keyword should only be used if 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 gas¹⁵, 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.

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.

¹⁴ 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.

¹⁵ 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.

5.2.60 VFPIIDIMS – INJECTION VERTICAL FLOW PERFORMANCE TABLE DIMENSIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
----------------	------	------	-------	---------	----------	---------	----------

Description

VFPIIDIMS 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 VFPPIDIMS 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 5.32: VFPIIDIMS Keyword Description

Example

```
--      INJECTING VFP TABLES
--      VFP      VFP      VFP
--      MXMFLO  MXMTHP  NMMVFT
VFPIIDIMS
      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.

5.2.61 VFPPDIMS – PRODUCTION VERTICAL FLOW PERFORMANCE TABLE DIMENSIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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 5.33: 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.

5.2.62 WATER – DEFINE THE WATER PHASE TO BE PRESENT IN THE MODEL

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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 oil phase input parameters are defined in the input deck.

There is no data required 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.

5.2.63 WELLDIMS – DEFINE THE WELLS AND GROUP DIMENSIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
----------------	------	------	-------	---------	----------	---------	----------

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	<p>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.</p> <p>If the generalized pseudo-pressure option has not been activated then this is ignored.</p> <p>This option is ignored by OPM Flow.</p>	201
Notes: <ol style="list-style-type: none"> Only parameters (1) to (4) are used by OPM Flow. The keyword is terminated by “/”. 			

Table 5.34: 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.

5.2.64 WSEGDIMS – DEFINE MULTI-SEGMENT WELL DIMENSIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

The WSEGDIMS keyword defines the multi-segment well dimensions for the multi-segment well model.

OPM Flow has not implemented multi-segment wells and therefore this keyword is ignored by OPM Flow and has no effect on the simulation.

6 GRID SECTION

6.1 INTRODUCTION

The GRID section defines the basic grid properties, including structure, faults and various static rock properties (porosity, permeability etc.). The information in this section will be used by the software to calculate the pore volume (PORV) for each cell, the cell mid-point depths, and the regular transmissibilities (TRANX, TRANY and TRANZ) 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 Mattax¹⁶). 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.

OPM Flow enables the user to define 1D, 2D and 3D models using three types of grids: Cartesian Regular Grid, Radial Grid¹⁷, 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.

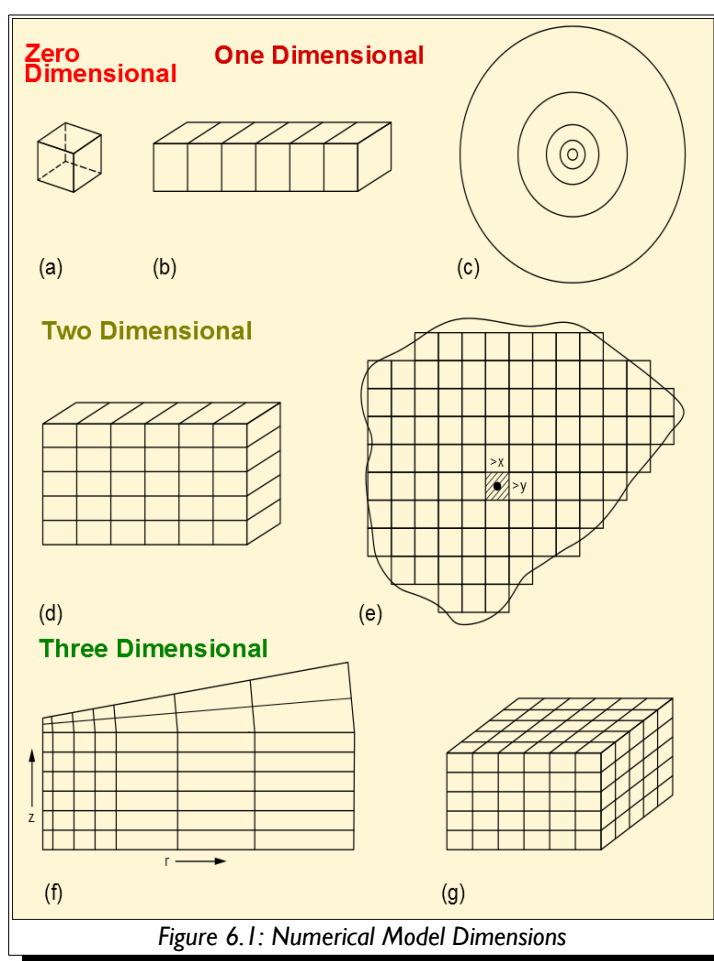


Figure 6.1: Numerical Model Dimensions

¹⁶ Mattax, C.C. and Dalton R.L. 1990. *Reservoir Simulation*. Society of Petroleum Engineers, Henry L. Doherty Series, Monograph Vol. 13

¹⁷ Radial grids are not currently implemented in OPM Flow version 2017-10 but is expected to be incorporated in the next 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 Odeh¹⁸.

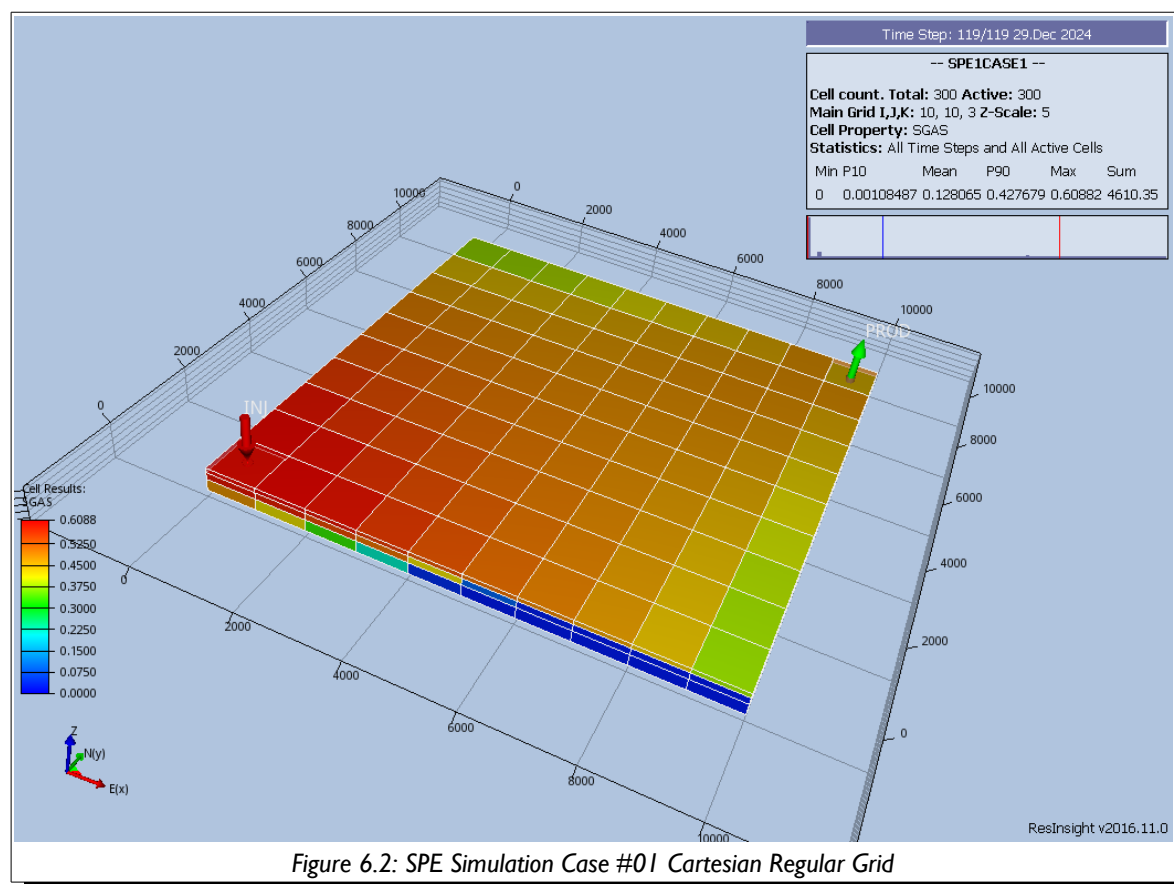


Figure 6.2: SPE Simulation Case #01 Cartesian Regular Grid

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

```
--
-- 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
```

¹⁸ 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.

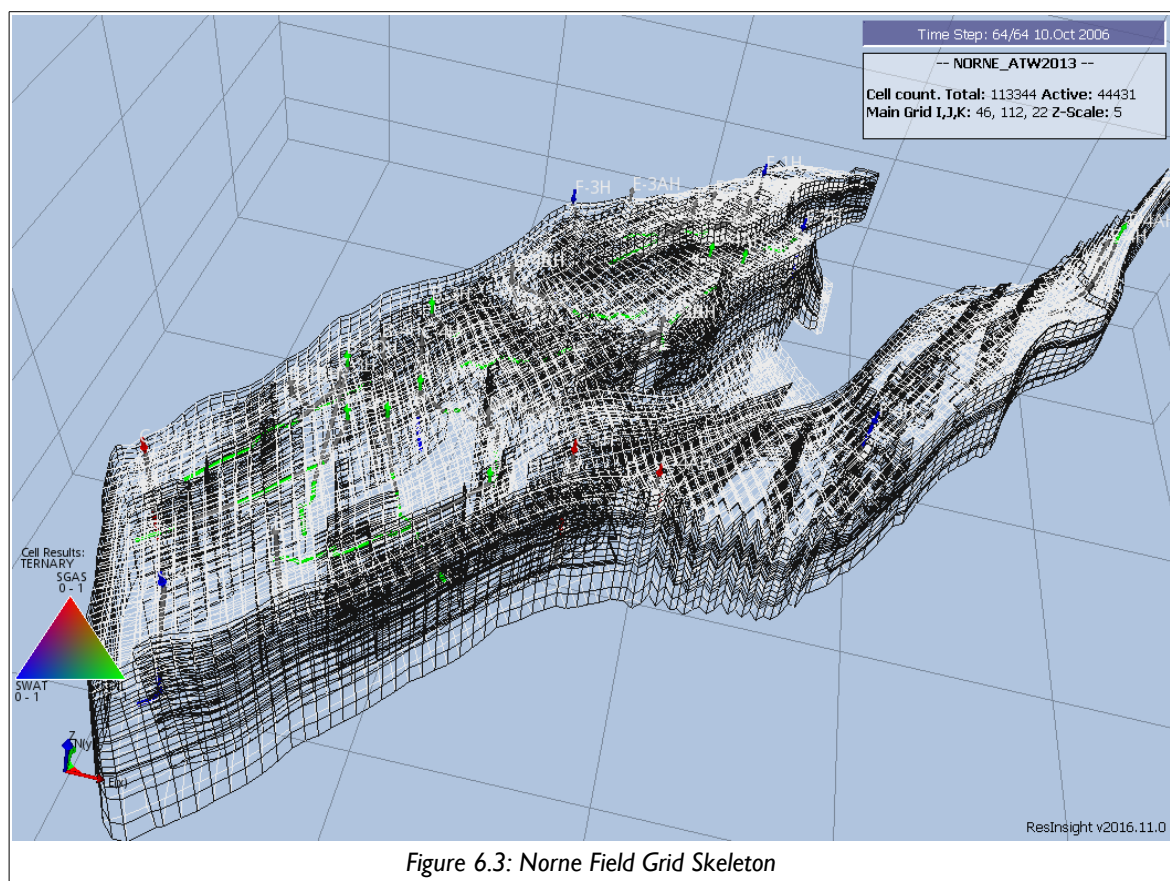


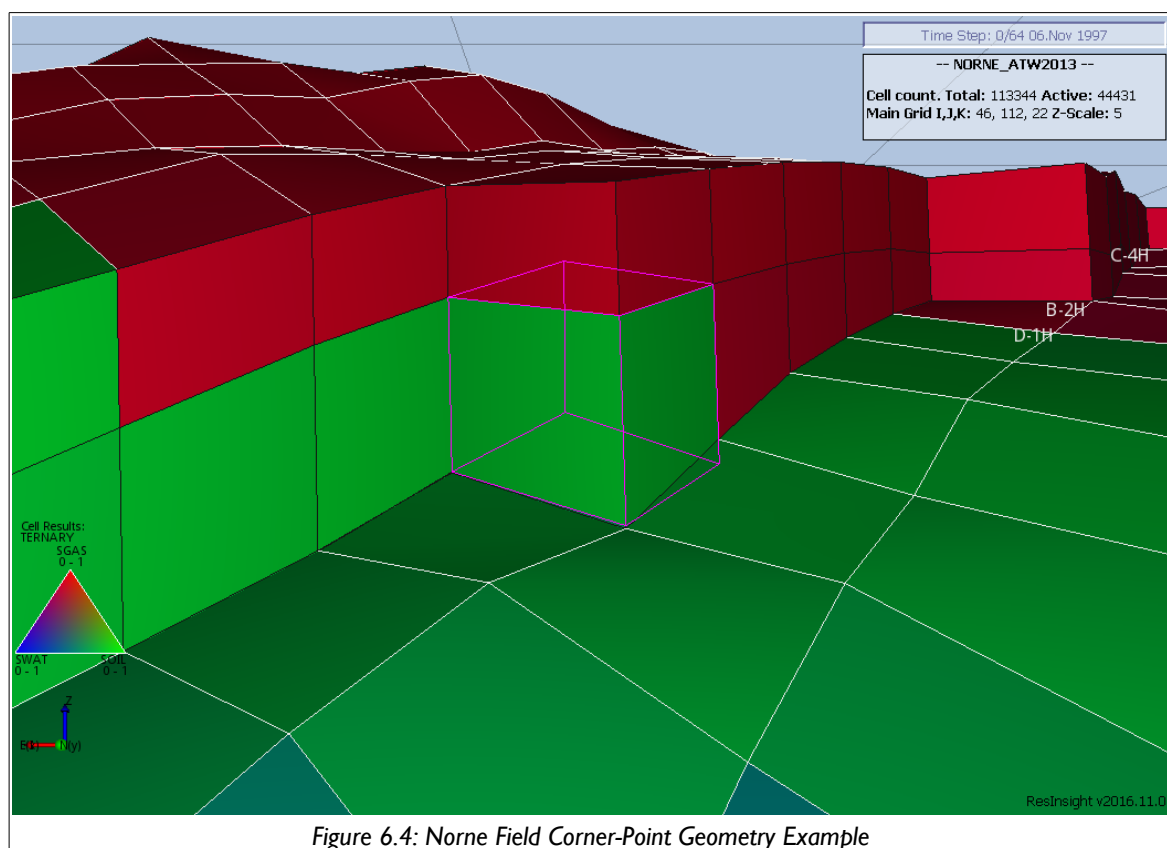
Figure 6.3: Norne Field Grid Skeleton

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.



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

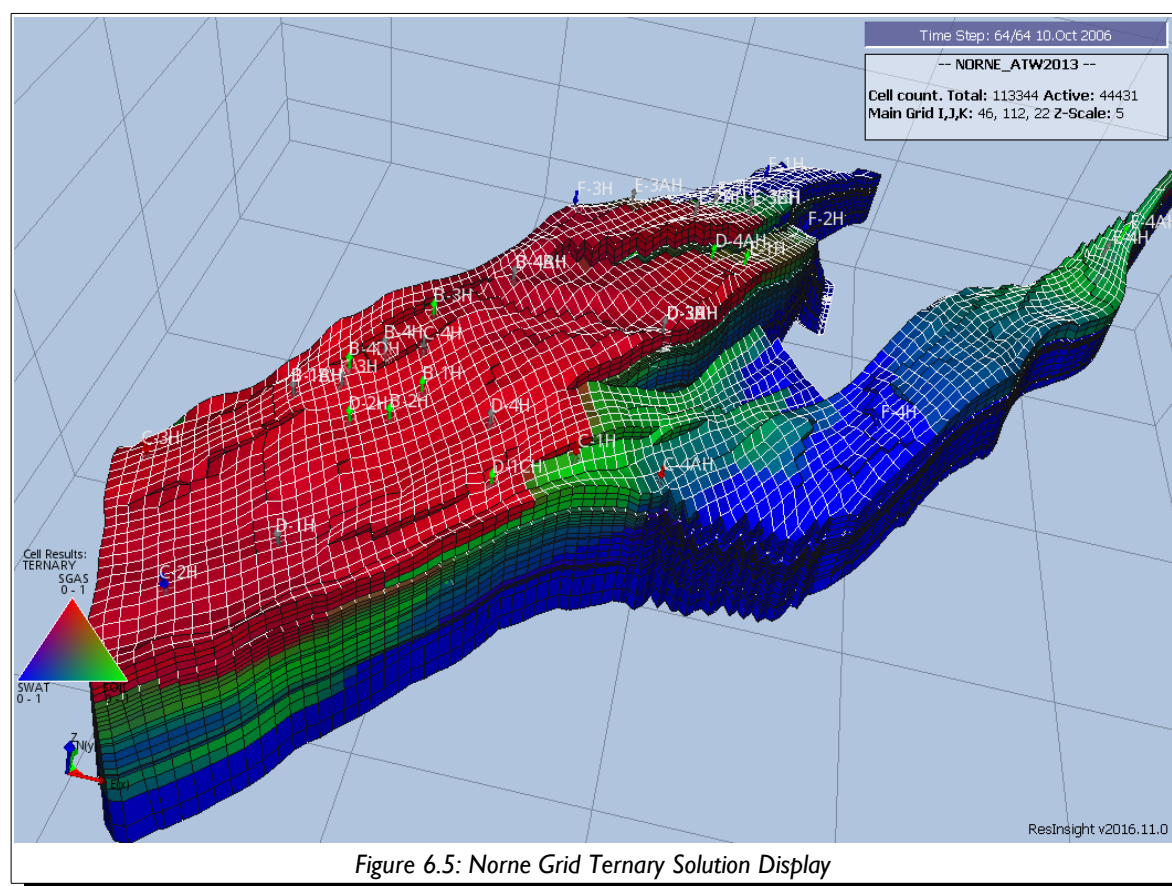


Figure 6.5: Norne Grid Ternary Solution Display

6.2.4 ROCK PROPERTIES

To be written in a future release of the manual.

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*I; however the full array must be specified.

Note that the 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
I	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 6.1: 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

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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 6.2: ADD Keyword Description

The applicable arrays for each section are defined in Table 6.3 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	TRANX	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	TRANHT	PCG		GASCONC		
THETA	DIFFR	PCW		SOLVCONC		
PERMR	DIFFHT			SOLVFRAC		
PERMHT				SFOAM		
DZNET				SPOLY		
PORO						
NTG						
FLUXNUM						
MULTNUM						
MPANUM						
DIFFX						
DIFFY						
DIFFZ						
DIFFR						
DIFFHT						

Table 6.3: ADD Keyword Applicable Arrays by Section

Example

```
--
-- -- ARRAY      CONSTANT -- ----- BOX -----
--                               I1  I2   J1  J2   K1  K2
ADD
'PERMX'      20.000      1*  1*   1*  1*   1*  1* / ADD 20 mD TO PERMX
/
```

The above example ADDS 20 units to the PERMX array in the GRID section to all grid blocks in the model.

6.3.3 ADDREG – ADD A CONSTANT TO AN ARRAY BASED ON A REGION NUMBER

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

The ADDREG keyword adds a constant to a specified array or part of an array based on cells with a specific region number. The region number array can be FLUXNUM, MULTNUM or OPERNUM and these arrays must be defined and be available before the ADDREG keyword is read by the simulator. The constant can be real or integer depending on the property array type; however, the property arrays that can be operated on is dependent on which section the ADDREG keyword is being used.

No.	Name	Description	Default
1	ARRAY	The name of the array to be modified. This is the keyword name identifying the property and is up to eight characters in length and enclosed in quotes.	None
2	CONSTANT	An integer or real value to be added to the ARRAY in the same units as the ARRAY property for a given REGION	0
3	REGION NUMBER	REGION NUMBER is a positive integer representing the region for which the CONSTANT in (2) should be applied	None
4	REGION ARRAY	The REGION ARRAY to use for applying the CONSTANT in (2) based on the REGION NUMBER in (3). REGION ARRAY can have the following values: 1) F for the FLUXNUM array 2) M for the MULTNUM array 3) O for the OPERNUM array	M
Notes: 1) Where the REGION NUMBER should be less than or equal to the maximum number of regions as defined on the REGDIMS keyword for the FIPNUM and OPERNUM arrays or the GRIDOPTS keyword for the MULTNUM array in the RUNSPEC section. 2) Each record must be terminated by a "/" and the keyword is terminated by "/".			

Table 6.4: ADDREG Keyword Description

The applicable arrays for each section are defined in Table 6.5 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	TRANX	SGL	IMBNUM	RV		
PERMY	TRANZ	SOCR	MISCNUM	RS		
PERMZ	DIFFX	SGU	PVTNUM	TBLK		
MULTX	DIFFY	KRW	ROCKNUM	GI		
MULTY	DIFFZ	KRO	SATNUM	OILAPI		
MULTZ	TRANR	KRG	WH2NUM	SALT		
DR	TRANHT	PCG		GASCONC		
THETA	DIFFR	PCW		SOLVCONC		
PERMR	DIFFHT			SOLVFRAC		
PERMHT				SFOAM		
DZNET				SPOLY		
PORO						
NTG						
FLUXNUM						
MULTNUM						
MPANUM						
DIFFX						
DIFFY						
DIFFZ						
DIFFR						
DIFFHT						

Table 6.5: 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 AQUANCON - ANALYTIC AQUIFER ASSIGNMENT

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

Description

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

6.3.5 AQUICON – NUMERICAL AQUIFERS CONNECTION TO THE GRID

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

Description

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

6.3.6 AQUNUM – NUMERICAL AQUIFER ASSIGNMENT

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

Description

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

6.3.7 BOX - DEFINE A RANGE OF GRID BLOCKS TO ENTER PROPERTY DATA

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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 6.6: 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 be 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.

6.3.8 CIRCLE – COMPLETION OF RADIAL GRID CIRCLE ACTIVATION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

Description

CIRCLE activates the completion of the circle for a radial grid by generating non-neighboring connections between the first and last cells in the radial plane and for all layers in the model. This is only activated if the radial grid extends 360 degrees in the radial plane. 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.

6.3.9 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. OPM Flow does not have a coal bed methane option and therefore this keyword is ignored by OPM Flow; however, it is documented here for completeness.

No.	Name	Description	Default
I	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.	I
Notes: <ol style="list-style-type: none"> 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 cell is not assigned a COALNUM region then the default value of I will be used. 3) COALNUM value of 0 sets the cell be a non-coal region. 4) The keyword is terminated by “/”. 			

Table 6.7: 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.

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

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	-------------	------	-------	---------	----------	---------	----------

Description

COORD defines a set of coordinate lines or pillars for a reservoir grid via an array. A total of (NX+1) x (NY+1) lines must be specified for each coordinate data set.

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			None
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	

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 6.8: 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.

6.3.11 COORDSYS – DEFINE COORDINATE GRID OPTIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

Description

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

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

6.3.12 COPY – COPY ARRAY DATA TO ANOTHER ARRAY

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

The COPY keyword copies an array (or part of an array) to another array or part of an array. The arrays can be real or integer depending on the array type; however, the arrays that can be operated on is dependent on which section the COPY keyword is being used.

No.	Name	Description	Default
1	ARRAY-1	The name of the array to be copied from. This is the keyword name identifying the property and is up to eight characters in length and enclosed in quotes.	None
2	ARRAY-2	The name of the array to be copied to. This is the keyword name identifying the property and is up to eight characters in length and enclosed in quotes.	None
3	I1	A positive integer that defines the lower bound of the array in the I-direction to be modified must be greater than or equal 1 and less than or equal to I2 and NX.	1
4	I2	A positive integer that defines the upper bound of the array in the I-direction to be modified must be greater than or equal to I1 and less than or equal to NX	NX
5	J1	A positive integer that defines the lower bound of the array in the J-direction to be modified must be greater than or equal 1 and less than or equal to J2 and NY.	1
6	J2	A positive integer that defines the upper bound of the array in the J-direction to be modified must be greater than or equal to J1 and less than or equal to NY.	NY
7	K1	A positive integer that defines the lower bound of the array in the K-direction to be modified must be greater than or equal to one and less than or equal to K2 and NZ.	1
8	K2	A positive integer that defines the upper bound of the array in the K-direction to be modified must be greater than or equal to K1 and less than or equal to NZ.	NZ
Notes: 1) Where NX, NY and NZ are the dimensions of the model as defined on the DIMENS keyword in the RUNSPEC section. 2) Each record must be terminated by a "/" and the keyword is terminated by "/".			

Table 6.9: COPY Keyword Description

The applicable arrays for each section are defined in Table 6.10 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	TRANX	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	TRANHT	PCG		GASCONC		
THETA	DIFFR	PCW		SOLVCONC		
PERMR	DIFFHT			SOLVFRAC		
PERMHT				SFOAM		
DZNET				SPOLY		
PORO						
NTG						
FLUXNUM						
MULTNUM						
MPANUM						
DIFFX						
DIFFY						
DIFFZ						
DIFFR						
DIFFHT						

Table 6.10: 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.

6.3.13 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.

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 6.11: COPYREG Keyword Description

The applicable arrays for each section are defined in Table 6.12 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	TRANX	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	TRANHT	PCG		GASCONC		
THETA	DIFFR	PCW		SOLVCONC		
PERMR	DIFFHT			SOLVFRAC		
PERMHT				SFOAM		
DZNET				SPOLY		
PORO						
NTG						
FLUXNUM						
MULTNUM						
MPANUM						
DIFFX						
DIFFY						
DIFFZ						
DIFFR						
DIFFHT						

Table 6.12: 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
--   FROM      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.

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

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

DRV¹⁹ 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 should be activated to indicate that radial geometry is being used.

No.	Name	Description			Default
		Field	Metric	Laboratory	
I	DRV	DRV is a vector of real numbers describing the cell size for the grid blocks in the R direction. Repeat counts may be used, for example 10*100.0.			None
		feet	m	cm	

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 6.13: DRV Keyword Description

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

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

Example

```
--
-- DEFINE GRID BLOCK SIZES IN THE R DIRECTION (BASED ON NX = 10)
--
DRV
1.0 3.0 10.0 30.0 50.0 100.0 200.0 500.0 750.0 1000.0 /
```

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.

¹⁹ Radial grids are not currently implemented in OPM Flow version 2017-10 but is expected to be incorporated in the next release.

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

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

DTHETAV²⁰ 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.

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. Repeat counts may be used, for example 10*100.0.			None
		degrees	degrees	degrees	
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 6.14: DTHETA Keyword Description

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

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

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.

²⁰ Radial grids are not currently implemented in OPM Flow version 2017-10 but is expected to be incorporated in the next release.

6.3.16 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.

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.

6.3.17 DX - DEFINE THE SIZE OF GRID BLOCKS IN THE X DIRECTION FOR ALL CELLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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	
I	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.			None
		feet	m	cm	

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 6.15: 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.

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

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

DXV defines the size of grid blocks in the X direction via a vector as opposed to defining the X direction cell size for each cell for a Cartesian Regular Grid.

No.	Name	Description			Default
		Field	Metric	Laboratory	
I	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.			None
		feet	m	cm	
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 6.16: 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.

6.3.19 DY - DEFINE THE SIZE OF GRID BLOCKS IN THE Y DIRECTION FOR ALL CELLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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	
I	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.			None
		feet	m	cm	

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 6.17: 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.

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

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

DYV defines the size of grid blocks in the Y direction via a vector as opposed to defining the Y direction cell size for each cell for a Cartesian Regular Grid.

No.	Name	Description			Default
		Field	Metric	Laboratory	
I	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.			None
		feet	m	cm	
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 6.18: 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.

6.3.21 DZ - DEFINE THE SIZE OF GRID BLOCKS IN THE Z DIRECTION FOR ALL CELLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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	
I	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.			None
		feet	m	cm	

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: 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.

6.3.22 DZV - DEFINE THE SIZE OF GRID BLOCKS IN THE Z DIRECTION VIA A VECTOR

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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	
I	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.			None
		feet	m	cm	
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 6.20: 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.

6.3.23 ENDBOX – DEFINE THE END OF THE BOX DEFINED GRID

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

This keyword marks the end of a previously defined BOX sub-grid as defined by a previously entered BOX keyword. The keyword resets the input grid to be the full grid as defined by the NX, NY, and NZ variables on the DIMENS keyword in the RUNSPEC section.

There is no data required for this keyword.

Example

```
--
--      DEFINE INPUT BOX FOR EDITING INPUT ARRAYS
--
--      ----- BOX -----
--      I1  I2   J1  J2   K1  K2
BOX      10  10   1   6   1   1
--
--                                     / DEFINE BOX AREA
--
--      DEFINE GRID BLOCK PERMZ DATA FOR THE INPUT BOX
--
--      PERMZ
--      6*0.01
--
--      DEFINE END OF INPUT BOX EDITING OF INPUT ARRAYS
--
--      ENDBOX
```

The above example defines a subset of the grid and sets the cells PERMZ values to 0.01 for that area. After which the ENDBOX keyword resets the input to be the full grid.

Note

It is good practice to always use the ENDBOX keyword to reset the input back to the full grid when all the modifications for a sub-grid have been completed.

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

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

The EQUALREG keyword sets a specified array to a constant for cells with a specific region number. The region number array can be FLUXNUM, MULTNUM or OPERNUM and these arrays must be defined and be available before the EQUALREG keyword is read by the simulator. The constant can be real or integer depending on the property array type; however, the property arrays that can be operated on is dependent on which section the EQUALREG keyword is being used.

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 6.21: EQUALREG Keyword Description

The applicable arrays for each section are defined in Table 6.22 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	TRANX	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	TRANHT	PCG		GASCONC		
THETA	DIFFR	PCW		SOLVCONC		
PERMR	DIFFHT			SOLVFRAC		
PERMHT				SFOAM		
DZNET				SPOLY		
PORO						
NTG						
FLUXNUM						
MULTNUM						
MPANUM						
DIFFX						
DIFFY						
DIFFZ						
DIFFR						
DIFFHT						

Table 6.22: 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.

6.3.25 EQUALS – SETS A SPECIFIED ARRAY TO A CONSTANT

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

The EQUALS keyword sets a specified array or part of an array to a constant. The constant can be real or integer depending on the array type; however, the arrays that can be operated on is dependent on which section the EQUALS keyword is being used.

No.	Name	Description	Default
1	ARRAY	The name of the array to be modified. This is the keyword name identifying the property and is up to eight characters in length and enclosed in quotes.	None
2	CONSTANT	An integer or real value that the ARRAY will be set to in the same units as the ARRAY property.	None
3	I1	A positive integer that defines the lower bound of the array in the I-direction to be modified must be greater than or equal to one and less than or equal to I2 and NX.	1
4	I2	A positive integer that defines the upper bound of the array in the I-direction to be modified must be greater than or equal to I1 and less than or equal to NX	NX
5	J1	A positive integer that defines the lower bound of the array in the J-direction to be modified must be greater than or equal to one and less than or equal to J2 and NY.	1
6	J2	A positive integer that defines the upper bound of the array in the J-direction to be modified must be greater than or equal to J1 and less than or equal to NY.	NY
7	K1	A positive integer that defines the lower bound of the array in the K-direction to be modified must be greater than or equal to one and less than or equal to K2 and NZ.	1
8	K2	A positive integer that defines the upper bound of the array in the K-direction to be modified must be greater than or equal to K1 and less than or equal to NZ.	NZ

Notes:

- 1) Where NX, NY and NZ are the dimensions of the model as defined on the DIMENS keyword in the RUNSPEC section.
- 2) Each record must be terminated by a "/" and the keyword is terminated by "/".

Table 6.23: EQUALS Keyword Description

The applicable arrays for each section are defined in Table 6.24 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	TRANX	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	TRANHT	PCG		GASCONC		
THETA	DIFFR	PCW		SOLVCONC		
PERMR	DIFFHT			SOLVFRAC		
PERMHT				SFOAM		
DZNET				SPOLY		
PORO						
NTG						
FLUXNUM						
MULTNUM						
MPANUM						
DIFFX						
DIFFY						
DIFFZ						
DIFFR						
DIFFHT						

Table 6.24: EQUALS Keyword Applicable Arrays by Section

Examples

```
--
-- -- 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 above example resets the PERMX, PERMY and PERMZ, arrays to 0.10, 0.10, and 0.01 for all cells in layer five, respectively.

6.3.26 FAULTS – DEFINE FAULTS IN THE GRID GEOMETRY

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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 6.25: 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'           /
.....
/
```

6.3.27 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 regions²¹. 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.

No.	Name	Description	Default
I	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.	I
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 6.26: 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
/

```

²¹ Although the FLUXNUM keyword is supported, OPM Flow currently does not support the flux boundary option.

6.3.28 GDORIENT - DEFINE GRID ORIENTATION PARAMETERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

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.

6.3.29 GRID - DEFINE THE START OF THE GRID SECTION OF KEYWORDS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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.

6.3.30 GRIDFILE – SET THE GRID FILE OUTPUT OPTIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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: 0 - for no GRID file to be written out. 1 - for the standard GRID file to be written out. 2 - for the extended GRID file to be written out. Only the default value of zero is supported.	0
2	NEGRID	A positive integer that controls the output of the 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.	1
Notes: 1) The keyword is terminated by “/”.			

Table 6.27: 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

6.3.31 GRIDUNIT – DEFINE THE GRID UNITS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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.	I*
Notes: 1) Note the alternative spelling METRES, that is METERS is not recognized. 2) The keyword is terminated by "/".			

Table 6.28: GRIDUNIT Keyword Description

6.3.32 INIT – ACTIVATES THE INIT FILE OPTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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 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 FMTOU keyword has been activated, or binary format if the FMTOU 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.

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.

6.3.33 INRAD – DEFINE THE INNER RADIUS OF A RADIAL GRID

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

INRAD²² 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	
I	INRAD	A single real positive number defining the inner radius of a radial grid.			None
		feet	m	cm	
Notes: I) The keyword is terminated by “/”.					

Table 6.29: INRAD Keyword Description

See also the DRV, DTHETA 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.

²² Radial grids are not currently implemented in OPM Flow version 2017-10 but is expected to be incorporated in the next release.

6.3.34 ISOLNUM – DEFINE THE INDEPENDENT RESERVOIR REGIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

The ISOLNUM keyword defines areas of the grid that consists of isolated reservoirs where the only form of communication between the reservoirs is via wellbore connections. This enables the reservoir flow equations to be solved independently for greater computational efficiency.

No.	Name	Description	Default
I	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.	I
Notes: <ol style="list-style-type: none"> 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 6.30: 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.

6.3.35 JFUNC - ACTIVATES THE LEVERETT J-FUNCTION OPTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

JFUNC keyword activates Leverett-J-Function²³ 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.

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: <div><div>1)</div>WATER: apply the J-Function option to the water-oil capillary pressure data only. <div>2)</div>GAS: apply the J-Function option to the gas-oil capillary pressure data only. <div>3)</div>BOTH: apply the J-Function option to the water-oil and the gas-oil capillary pressure data.</div>			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..			None
		dynes/cm	dynes/cm	dynes/cm	
3	OGSTEN	A positive real number that defines oil-gas surface tension used to de-normalized J-Function data entered in the PROPS section..			None
		dynes/cm	dynes/cm	dynes/cm	
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 $\sqrt{\frac{k}{\phi}}$ use $\frac{k^{0.5}}{\phi^\alpha}$ 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 $\sqrt{\frac{k}{\phi}}$ use $\frac{k^\beta}{\phi^{0.5}}$ instead in the transformation.			0.5
6	PERM	PERM is a character string that sets the permeability array to be used in the transform, based on the following options: <div><div>1)</div>X: use the PERMX array. <div>2)</div>XY: use the average of the PERMX and PERMY arrays. <div>3)</div>Y: use the PERMY array. <div>4)</div>Z: use the PERMZ array.</div>			XY

Notes:

1)

The keyword is terminated by “/”.

Table 6.31: JFUNC Keyword Description

²³ Leverett, M. C.; “Capillary Behaviour in Porous Solids”, Trans. AIME (1941) 142, 152-168.

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-function²⁴, which is defined as:

$$J(S_w) = \frac{P_{c,res}(S_w) \sqrt{\frac{k}{\phi}}}{\sigma} \quad (6.1)$$

Where:

- $J(S_w)$ = dimensionless function of water saturation
- $P_c(S_w)$ = capillary pressure (kPa)
- k = permeability, (m²)
- ϕ = porosity (fraction)
- σ = interfacial tension (mN/m)
- Θ = contact angle

Sometimes the equation is stated with the $\cos \Theta$ term included, that is:

$$J(S_w) = \frac{P_{c,res}(S_w) \sqrt{\frac{k}{\phi}}}{\sigma \cos \Theta} \quad (6.2)$$

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:

$$J(S_w) = P_{c,res}(S_w) \sqrt{\frac{k}{\phi}} \quad (6.3)$$

However, in the simulator it is necessary to use the formal definition as outlined in equation (6.1). 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 (6.1), that is:

$$J(S_w) = \frac{P_{c,res}(S_w) \left(\frac{k^\beta}{\phi^\alpha} \right)}{\sigma} \quad (6.4)$$

Where:

- $J(S_w)$ = dimensionless function of water saturation
- $P_c(S_w)$ = capillary pressure (kPa)
- k = permeability, (m²)
- ϕ = porosity (fraction)
- σ = interfacial tension (mN/m)
- Θ = contact angle
- α = porosity power value
- β = permeability value

²⁴ Leverett, M. C.; "Capillary Behaviour in Porous Solids", *Trans. AIME (1941)* 142, 152-168.

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.

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.

6.3.36 MAPAXES- DEFINE THE MAP ORIGIN INPUT DATA

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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 6.32: 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.

6.3.37 MAPUNITS – DEFINE THE MAP AXES UNITS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

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.

No.	Name	Description	Default
I	MAPUNITS	<p>A character string that defines the units of the coordinates stated on the MAPAXES keyword, and should be set to:</p> <ol style="list-style-type: none"> 1) FIELD for field units 2) METRES for metric units, or 3) LAB for laboratory units 	METRES
<p>Notes:</p> <ol style="list-style-type: none"> 1) Note the alternative spelling of METRES, that is METERS is not recognized. 2) The keyword is terminated by “/”. 			

Table 6.33: 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.

6.3.38 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.

No.	Name	Description	Default
1	ARRAY	The name of the array to be modified. This is the keyword name identifying the property and is up to eight characters in length and enclosed in quotes.	None
2	CONSTANT	CONSTANT is a positive integer or positive real value that an ARRAY element will be reset to if an element in the defined input BOX, as defined by items (3) to (8), is greater than CONSTANT. CONSTANT has in the same units as the ARRAY property.	None
3	I1	The lower bound of the array in the I-direction to be modified must be greater than or equal to one and less than or equal to I2 and NX.	I
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.	I
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.	I
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 6.34: MAXVALUE Keyword Description

The applicable arrays for each section are defined in Table 6.35 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	TRANX	SGL				
PERMY	TRANZ	SGCR				
PERMZ	DIFFX	SGU				
MULTX	DIFFY	KRW				
MULTY	DIFFZ	KRO				
MULTZ	TRANR	KRG				
DR	TRANHT	PCG				
THETA	DIFFR	PCW				
PERMR	DIFFTHT					
PERMTHT						
DZNET						
PORO						
NTG						
FLUXNUM						
MULTNUM						
MPANUM						
DIFFX						
DIFFY						
DIFFZ						
DIFFR						
DIFFTHT						

Table 6.35: 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.

6.3.39 MINPV – SET A MINIMUM GRID BLOCK PORE VOLUME THRESHOLD

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

MINPV defines a minimum threshold pore volume that makes all grid blocks whose pore volume is below this value inactive in the model. Inactive cells are not used in OPM Flow calculations.

No.	Name	Description			Default
		Field	Metric	Laboratory	
I	MPVTHRES	MPVTHRES is a real positive number that defines the minimum pore volume for a cell to be active in the model.			Defined
		rb 1.0e-6	rm ³ 1.0e-6	rcc 1.0e-6	
Notes: I) The keyword is terminated by “/”.					

Table 6.36: 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 m³) as minimum pore volume for a cell to be active in the model.

6.3.40 MINVALUE – SET A MINIMUM VALUE FOR AN ARRAY ELEMENT

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

The MINVALUE keyword sets a minimum value for the specified array or part of an array. The constant can be real or integer depending on the array type; however, the arrays that can be operated on is dependent on which section the MINVALUE keyword is being used.

No.	Name	Description	Default
1	ARRAY	The name of the array to be modified. This is the keyword name identifying the property and is up to eight characters in length and enclosed in quotes.	None
2	CONSTANT	CONSTANT is a positive integer or positive real value that an ARRAY element will be reset to if an element in the defined input BOX, as defined by items (3) to (8), is less than CONSTANT. CONSTANT has in the same units as the ARRAY property.	None
3	I1	A positive integer that defines the lower bound of the array in the I-direction to be modified must be greater than or equal to one and less than or equal to I2 and NX.	I
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.	I
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.	I
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 6.37: MINVALUE Keyword Description

The applicable arrays for each section are defined in Table 6.38 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	TRANX	SGL				
PERMY	TRANZ	SGCR				
PERMZ	DIFFX	SGU				
MULTX	DIFFY	KRW				
MULTY	DIFFZ	KRO				
MULTZ	TRANR	KRG				
DR	TRANHT	PCG				
THETA	DIFFR	PCW				
PERMR	DIFFHT					
PERMHT						
DZNET						
PORO						
NTG						
FLUXNUM						
MULTNUM						
MPANUM						
DIFFX						
DIFFY						
DIFFZ						
DIFFR						
DIFFHT						

Table 6.38: 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.

6.3.41 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. The 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 6.39: 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.

6.3.42 MULTIPLY – MULTIPLY A SPECIFIED ARRAY BY A CONSTANT

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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 6.40: MULTIPLY Keyword Description

The applicable arrays for each section are defined in Table 6.41 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	TRANX	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	TRANHT	PCG		GASCONC		
THETA	DIFFR	PCW		SOLVCONC		
PERMR	DIFFHT			SOLVFRAC		
PERMHT				SFOAM		
DZNET				SPOLY		
PORO						
NTG						
FLUXNUM						
MULTNUM						
MPANUM						
DIFFX						
DIFFY						
DIFFZ						
DIFFR						
DIFFHT						

Table 6.41: 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 multiplies the PERMZ property array by 0.5 throughout the model.

6.3.43 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 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 6.42: MULTIREG Keyword Description

The applicable arrays for each section are defined in Table 6.43 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	TRANX	SGL	IMBNUM	RV		
PERMY	TRANZ	SOCR	MISCNUM	RS		
PERMZ	DIFFX	SGU	PVTNUM	TBLK		
MULTX	DIFFY	KRW	ROCKNUM	GI		
MULTY	DIFFZ	KRO	SATNUM	OILAPI		
MULTZ	TRANR	KRG	WH2NUM	SALT		
DR	TRANHT	PCG		GASCONC		
THETA	DIFFR	PCW		SOLVCONC		
PERMR	DIFFHT			SOLVFRAC		
PERMHT				SFOAM		
DZNET				SPOLY		
PORO						
NTG						
FLUXNUM						
MULTNUM						
MPANUM						
DIFFX						
DIFFY						
DIFFZ						
DIFFR						
DIFFHT						

Table 6.43: 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.

6.3.44 MULTNUM – DEFINE THE MULTIPLE TRANSMISSIBILITY REGIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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
I	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.	I
Notes: <ol style="list-style-type: none"> 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 6.44: 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          /
/
```


6.3.45 MULTPV – MULTIPLY CELL PORE VOLUMES BY A CONSTANT

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

MULTPV multiplies 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
I	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 6.45: MULTPV Keyword Description

See also the MULTREGP for scaling the cell pore volumes by region numbers.

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
--
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.

6.3.46 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 NUMBER	REGION NUMBER 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	REGION ARRAY	The REGION ARRAY to use for applying the CONSTANT in (2) based on the REGION NUMBER in (1). 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 6.46: 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.

6.3.47 MULTREGT– MULTIPLY TRANSMISSIBILITIES BETWEEN REGIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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.	0
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 YZ.	XYZ
	TYPE	A character string that defines the type of connections the transmissibility multiplier should be applied to, should be one of the following: 1) NNC – Only apply the transmissibility multiplier between REGION1 and REGION2 to non-neighbor connections. 2) NONNC – Do not apply the transmissibility multiplier between REGION1 and REGION2 to non-neighbor connections. 3) ALL - Apply the transmissibility multiplier between REGION1 and REGION2 to all connections.	ALL
4	REGION ARRAY	The REGION ARRAY to use for applying the CONSTANT in (2) based on the REGION NUMBER in (1). 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 6.47: MULTREGT Keyword Description

Example

```
--  
-- 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  
/
```

The above example isolates all regions from one another by setting the transmissibility for the MULTNUM regions to zero for all directions and for connections types.

6.3.48 MULTX - MULTIPLY CELL TRANSMISSIBILITY IN THE +X DIRECTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

MULTX multiplies 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).

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
I	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: <ol style="list-style-type: none"> 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 6.48: 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.

6.3.49 MULTX- - MULTIPLY CELL TRANSMISSIBILITY IN THE -X DIRECTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

MULTX- multiplies 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
I	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 6.49: 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.

6.3.50 MULTY - MULTIPLY CELL TRANSMISSIBILITY IN THE +Y DIRECTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

MULTY multiplies 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+I, 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
I	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: <ol style="list-style-type: none"> 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 6.50: 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.

6.3.51 MULTY- - MULTIPLY CELL TRANSMISSIBILITY IN THE -Y DIRECTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

MULTY- multiplies 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
I	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 6.51: 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.

6.3.52 MULTZ - MULTIPLY CELL TRANSMISSIBILITY IN THE +Z DIRECTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

MULTZ multiplies 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
I	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 6.52: 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.

6.3.53 MULTZ- - MULTIPLY CELL TRANSMISSIBILITY IN THE -Z DIRECTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

MULTZ- multiplies 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-I) 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
I	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 6.53: 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.

6.3.54 NEWTRAN – ACTIVATE IRREGULAR CORNER-POINT GRID TRANSMISSIBILITIES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

This keyword switches on Irregular Corner-Point Grid geometry transmissibility calculation, which is the default option for this type of grid. Grids defined with the COORD and ZCORN keywords will always invoke this option by default.

For Cartesian Regular Grids defined by the DX, DY, and DZ series of keywords the block center geometry transmissibility calculations should be activated via the OLDTRAN keyword. Again this is automatically invoked if this type of grid is being employed.

There is no data required for this keyword.

Example

```
--
--      ACTIVATE IRREGULAR CORNER-POINT GRID TRANSMISSIBILITIES
--
NEWTRAN
```

The above example manually activates Irregular Corner-Point Grid transmissibility calculations.

6.3.55 NNC – DEFINE NON-NEIGHBOR CONNECTIONS BETWEEN CELLS MANUALLY

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

NNC enables non-neighbor connections 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.			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.			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.			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.			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.			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.			None
7	TRANSNNC	TRANSNNC is a positive real number that defines the transmissibility between the first grid block (I1, J1, K1) and the second grid block (I2, J2, K2).			0.0
		cP.rb/day/psi	cP.rm ³ /day/bars	cP.rcc/hr/atm	
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			1
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			1
10	IPRSNUM1	IPRSNUM1 is a positive integer defining which pressure table number to be used for flow from the first grid block to the second grid block			1
11	IPRSNUM2	IPRSNUM2 is a positive integer defining which pressure table number to be used for flow from the second grid block to the first grid block			1
12	FACEI	FACEI is a character string that defines the face associated with flow from the first grid block to the second grid block, where FACEI can have values of: X+, X-, Y+, Y-, Z+, or Z-.			None

No.	Name	Description			Default
		Field	Metric	Laboratory	
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+, orZ-.			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).			0.0
		feet	meters	cm	
15	DISPNNC	DISPNNC s a positive real number that defines the dispersion coefficient $\frac{1}{(Area \times Porosity)}$ between the first grid block (I1, J1, K1) and the second grid block (I2, J2, K2), used with the DISPERSE option.			0.0
		ft ²	m ²	cm ²	
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).			None
		ft ²	m ²	cm ²	
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.			None
		mD	mD	mD	
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 6.54: NNC Keyword Description

If the transmissibility across a fault needs to be modified see the FAULTS and MULTFLT keywords for an alternative method to modifying fault transmissibilities. Transmissibility between reservoir regions can be modified by using MULTREGT keywords and MULTX, MULTY and MULTZ series of keywords can be used to modify transmissibility between cells.

Example

```
--
--      MANUALLY DEFINE NON-NEIGHBOR CONNECTIONS
--
--      ----- BOX -----  -- TRANSNCC --
--      I1   J1   K1   I2   J2   K2
NCC
      1     1     1     1     2     2      0.2000    / SET NNC FOR FAULT
      1     1     2     1     2     3      0.2000    / SET NNC FOR FAULT
      1     1     3     1     2     4      0.2000    / SET NNC FOR FAULT
/
```

The above example defines the transmissibility between cells (1, 1, 1) and (1, 2, 2), (1, 1, 2) and (1, 2, 3) and (1, 1, 3) and (1, 2, 4) to be 0.200.

6.3.56 NOGGF – DEACTIVATE OUTPUT OF GRID GEOMETRY FILE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

Description

This keyword deactivates the output of a standard GRID or extended GRID file, as well as the extensible EGRID file for post-processing applications.

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

6.3.57 NTG – DEFINE THE NET-TO-GROSS RATIO FOR ALL THE CELLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

NTG defines the Net-to-Gross Ratio (“NTG”) for all the cells in the model via an array. The keyword can be used for all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
I	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.			None
		dimensionless	dimensionless	dimensionless	
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 6.55: 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.

6.3.58 OLDTRAN – ACTIVATE CARTESIAN REGULAR GRID TRANSMISSIBILITIES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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.

Example

```
--
--      ACTIVATE CARTESIAN REGULAR GRID TRANSMISSIBILITIES
--
OLDTRAN
```

The above example manually activates Cartesian Regular Grid transmissibility calculations.

6.3.59 OPERATE – DEFINE MATHEMATICAL OPERATIONS ON ARRAYS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

Description

This keyword defines the 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.

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

6.3.60 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 MULTNUM region array.

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

6.3.61 OUTRAD - DEFINE THE OUTER RADIUS OF A RADIAL GRID

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

OUTRAD²⁵ 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	
I	OUTRAD	A single real positive number greater than INRAD defining the outer radius of a radial grid.			None
		feet	m	cm	
Notes: I) The keyword is terminated by “/”.					

Table 6.56: OUTRAD Keyword Description

See also the DRV, DTHETA 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.

Examples

```
--
--      OUTER RADIUS OF FIRST GRID BLOCK IN THE RADIAL DIRECTION
--
OUTRAD
  1000.0
/
```

The above example defines the outer radius to be 1000.0 feet.

²⁵ Radial grids are not currently implemented in OPM Flow version 2017-10 but is expected to be incorporated in the next release.

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

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

Description

PERMR²⁶ 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.

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

²⁶ Radial grids are not currently implemented in OPM Flow version 2017-10 but is expected to be incorporated in the next release.

6.3.63 PERMX - DEFINE THE PERMEABILITY IN THE X DIRECTION FOR ALL THE CELLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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	
I	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.			None
		mD	mD	mD	

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 6.57: 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.

6.3.64 PERMXX – DEFINE THE PERMEABILITY TENSOR IN THE XX DIRECTION FOR ALL THE CELLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

Description

This keyword defines the permeability tensor off-diagonal values for the XX direction.

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

6.3.65 PERMXY – DEFINE THE PERMEABILITY TENSOR IN THE XY DIRECTION FOR ALL THE CELLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

Description

This keyword defines the permeability tensor off-diagonal values for the XY direction.

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

6.3.66 PERMY - DEFINE THE PERMEABILITY IN THE Y DIRECTION FOR ALL THE CELLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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	
I	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.			None
		mD	mD	mD	

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 6.58: 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.

6.3.67 PERMYY – DEFINE THE PERMEABILITY TENSOR IN THE YY DIRECTION FOR ALL THE CELLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

Description

This keyword defines the permeability tensor off-diagonal values for the YY direction.

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

6.3.68 PERMYZ – DEFINE THE PERMEABILITY TENSOR IN THE YZ DIRECTION FOR ALL THE CELLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

Description

This keyword defines the permeability tensor off-diagonal values for the YZ direction.

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

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

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

PERMZ defines the permeability in the Z direction for all the cells in the model via an array. The keyword can be used for all grid types, except for the Radial Grid geometry.

No.	Name	Description			Default
		Field	Metric	Laboratory	
I	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.			None
		mD	mD	mD	

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 6.59: 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.

6.3.70 PERMZX – DEFINE THE PERMEABILITY TENSOR IN THE ZX DIRECTION FOR ALL THE CELLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

Description

This keyword defines the permeability tensor off-diagonal values for the ZX direction.

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

6.3.71 PERMZZ – DEFINE THE PERMEABILITY TENSOR IN THE ZZ DIRECTION FOR ALL THE CELLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

Description

This keyword defines the permeability tensor off-diagonal values for the ZZ direction.

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

6.3.72 PINCH – DEFINE PINCH-OUT LAYER OPTIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

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.			Defined
		ft. 0.001	m 0.001	cm 0.001	
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.			Defined
		ft. 1.0E20	m 1.0E20	cm 1.0E20	
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	<p>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:</p> <ul style="list-style-type: none">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. <p>Note if PINCHCAL has been set equal to ALL then PINCHMUL is reset to TOP, irrespective of the entered value for PINCHMUL.</p>			TOP
Notes: <ul style="list-style-type: none">1) The keyword is terminated by “/”.					

Table 6.60: 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
-- 1*            1*     1*         1*
/
```

For the second example, the MINPV keyword is used to set the minimum pore volume to 500 m³ (metric units) and then the PINCH keyword is invoked with PINCHGAP set equal to GAP, as follows:

```
--
-- MINIMUM PORE VOLUME FOR ACTIVE CELLS
--
MINPV
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 m³. These deactivated cells are inactive in the model and therefore are not included in the flow calculations; however, by default they will result in no-flow barriers but may not be thin enough for PINCH to create NNCs across them. By setting PINCHGAP equal to GAP on the PINCH keyword (the default setting), then OPM Flow generates NNCs across the cells that have been deactivated by the MINPV keyword. However, in this case there may be grid blocks in the model with a pore volume greater than MINPV but a thickness less than the pinch-out threshold. These cells will not be deactivated by the PINCH keyword.

6.3.73 PINCHNUM – DEFINE PINCH-OUT REGIONS FOR THE PINCHREG KEYWORD

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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

No.	Name	Description	Default
I	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.	I
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 6.61: 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 CRITERIA 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.

6.3.74 PINCHREG - DEFINE PINCH-OUT REGION OPTIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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.			Defined
		ft. 0.001	m 0.001	cm 0.001	
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.			Defined
		ft. 1.0E20	m 1.0E20	cm 1.0E20	
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	<p>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:</p> <ul style="list-style-type: none">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. <p>Note if PINCHCAL has been set equal to ALL then PINCHMUL is reset to TOP, irrespective of the entered value for PINCHMUL.</p>			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 6.62: PINCHREG Keyword Description

Example

```
--
--      SET PINCH-OUT CRITERIA 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.

6.3.75 PORO - DEFINE THE POROSITY VALUES FOR ALL THE CELLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

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

No.	Name	Description			Default
		Field	Metric	Laboratory	
I	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			None
		dimensionless	dimensionless	dimensionless	
Notes: 1) TThe 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) he keyword is terminated by “/”.					

Table 6.63: PORO Keyword Description

See also the NTG, PERMX, PERMY and PERMZ 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.

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

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

Description

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

This keyword is not recognized by OPM Flow and usage will result in an error.

6.3.77 RADFIN4 – DEFINE A RADIAL LOCAL GRID REFINEMENT WITH FOUR COLUMNS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

Description

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

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

6.3.78 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 format in the commercial simulator and was subsequently superseded by the second format. The second format consists of the keyword followed by a series of character strings that indicate the data to be printed. In most cases the character string is the keyword used to load the data in the OPM Flow input deck, for example PORO for the porosity array. It 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 6.64: 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 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      /
```

A	B	C	D	E	F	G	H	I	J	K	L	M	N	O	P	Q	R	S	T	U	V	W	X	Y	Z
---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---

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

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	-------------	------	-------	---------	----------	---------	----------

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.	1
5	TYPE	A character string set to either T or 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 6.65: 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 the a 46 x 112 x 22 grid with one set of irregular corner-point data.

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

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

SWATINIT defines the initial water saturation for all the cells in the model via an array. The keyword can be used for 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			None
		dimensionless	dimensionless	dimensionless	

Notes:

- 1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
- 2) The keyword is terminated by “/”.

Table 6.66: 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.

A	B	C	D	E	F	G	H	I	J	K	L	M	N	O	P	Q	R	S	T	U	V	W	X	Y	Z
---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---

6.3.81 THCONR – DEFINE ROCK AND FLUID THERMAL CONDUCTIVITY

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

The THCONR keyword defines the rock plus fluid thermal conductivity for when the temperature calculation is activated by the TEMP keyword in the RUNSPEC section.

Currently OPM Flow does not support this feature and hence the keyword is ignored; however, it is documented here for completeness.

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			None
		Btu/ft/day/°R	kJ/m/day/K	J/cm/hr/K	

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 6.67: 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.

6.3.82 TOPS - DEFINE THE DEPTH AT THE CENTER OF THE TOP FACE FOR EACH CELL

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

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	
I	TOPS	<p>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.</p> <p>See the DIMENS keyword in the RUNSPEC section for the definition of NX, NY and NZ.</p> <p>Repeat counts may be used, for example 10*5201.0.</p>			None
		feet	m	cm	
Notes:					
I) The keyword is terminated by “/”.					

Table 6.68: TOPS Keyword Description

See also the DEPTH 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
/

```

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

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

ZCORN defines the depth of each corner point of a grid block on the pillars defining the reservoir grid.

A total of $8 \times NX \times NY \times NZ$ 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 uses with Irregular Corner-Point Grids.

No.	Name	Description			Default
		Field	Metric	Laboratory	
I	ZCORN	An array of depths with 8 depths for each cell, for a total of 8 x Nx x NY x NZ entries			None
		feet	metres	cm	
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 6.69: 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

To be written in a future release of the manual.

7.2 DATA REQUIREMENTS

To be written in a future release of the manual.

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 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.

7.3.4 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.

7.3.5 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.

7.3.6 DEPTH - EDITS THE DEPTH AT THE CENTER OF EACH CELL

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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	
I	DEPTH	DEPTH is an array of real numbers defining the depth at the center of each cell in the model. Only the values in Repeat counts may be used, for example 30*5201.0.			None
		feet	m	cm	
Notes: I) The keyword is terminated by “/”.					

Table 7.1: TOPS Keyword Description

See also the TOPS keyword to define the top 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.

```
-- ----- 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
/
```

7.3.7 EDIT - DEFINE THE START OF THE EDIT SECTION OF KEYWORDS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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.

7.3.8 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.

7.3.9 EDITNNC – EDIT NON-NEIGHBOR CONNECTIONS BETWEEN CELLS MANUALLY

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

EDITNNC enables non-neighbor connections, 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

$$T_{new} = C \times T_{old}.$$

No.	Name	Description	Default
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.	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.	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.	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.	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.	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.	None
7	TRANSMUL	TRANSMUL is a positive real number that defines a constant that scales the transmissibility between the first grid block (I1, J1, K1) and the second grid block (I2, J2, K2).	1
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	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	0
10	IPRSNUM1	IPRSNUM1 is a positive integer defining which pressure table number to be used for flow from the first grid block to the second grid block	0
11	IPRSNUM2	IPRSNUM2 is a positive integer defining which pressure table number to be used for flow from the second grid block to the first grid block	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

No.	Name	Description	Default
14	DIFFNNC	DIFFNNC is a positive real number that scales the diffusivity between the first grid block (I1, J1, K1) and the second grid block (I2, J2, K2).	0.0
Notes: 1) Each record must be terminated by a "/" and the keyword is terminated by "/".			

Table 7.2: EDITNNC Keyword Description

If the transmissibility across a fault needs to be modified see the FAULTS and MULTFLT keywords for an alternative method to modifying fault transmissibilities. Transmissibility between reservoir regions can be modified by using MULTREGT keywords and MULTX, MULTY and MULTZ series of keywords can be used to modify transmissibility between cells.

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

Example

```
--
--      MANUALLY DEFINE NON-NEIGHBOR CONNECTIONS
--
--      ----- BOX -----  -- TRANSNCC --
--      I1    J1    K1      I2    J2    K2
EDITNCC
      1      1      1      1      2      2      0.2000    / SET NNC FOR FAULT
      1      1      2      1      2      3      0.2000    / SET NNC FOR FAULT
      1      1      3      1      2      4      0.2000    / SET NNC FOR FAULT
/
```

The above example multipliers the transmissibility between cells (1, 1, 1) and (1, 2, 2), (1, 1, 2) and (1, 2, 3) and (1, 1, 3) and (1, 2, 4) by 0.200.

7.3.10 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.

7.3.11 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.

7.3.12 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.

7.3.13 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.

7.3.14 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.

7.3.15 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.

7.3.16 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.

7.3.17 MULTPV – MULTIPLY CELL PORE VOLUMES BY A CONSTANT

MULTPV multiplies 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.

7.3.18 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.

7.3.19 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.

7.3.20 MULTX - MULTIPLY CELL TRANSMISSIBILITY IN THE +X DIRECTION

MULTX multiplies the transmissibility between two cell faces in the +X direction for all the cells in the model via an array, that is the keyword sets the transmissibility multiplier of block (I, J, K) between the cells (I, J, K) and (I+1, J, K).

See [MULTX - Multiply Cell Transmissibility in the +X Direction](#) in the GRID section for a full description.

7.3.21 MULTX- - MULTIPLY CELL TRANSMISSIBILITY IN THE -X DIRECTION

MULTX- multiplies 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.

7.3.22 MULTY - MULTIPLY CELL TRANSMISSIBILITY IN THE +Y DIRECTION

MULTY multiplies 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.

7.3.23 MULTY- - MULTIPLY CELL TRANSMISSIBILITY IN THE -Y DIRECTION

MULTY- multiplies 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.

7.3.24 MULTZ - MULTIPLY CELL TRANSMISSIBILITY IN THE +Z DIRECTION

MULTZ multiplies 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.

7.3.25 MULTZ- - MULTIPLY CELL TRANSMISSIBILITY IN THE -Z DIRECTION

MULTZ- multiplies the transmissibility between two cell faces in the -Z direction for all the cells in the model via an array, that is the keyword sets the transmissibility multiplier of block (I, J, K) between the cells (I, J, K-1) and (I, J, K).

See [MULTZ- - Multiply Cell Transmissibility in the -Z Direction](#) in the GRID section for a full description.

7.3.26 OPERATE – DEFINE MATHEMATICAL OPERATIONS ON ARRAYS

This keyword defines the 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.

See [OPERATE – Define Mathematical Operations on Arrays](#) in the GRID section for a full description.

7.3.27 PORV - DEFINE THE PORE VOLUMES FOR ALL THE CELLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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.

No.	Name	Description			Default
		Field	Metric	Laboratory	
I	PORV	PORV is an array of real positive numbers assigning a pore volume to each cell in the model. Repeat counts may be used, for example 20*100.0.			None
		rb	rm ³	rcc	

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 7.3: 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

7.3.28 TRANX - DEFINE THE TRANSMISSIBILITY IN THE X DIRECTION FOR ALL THE CELLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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 permeability in the X direction to each cell in the model. Repeat counts may be used, for example 20*100.0.			None
		cP.rb/day/psi	cP.rm ³ /day/bars	cP.rcc/hr/atm	
Notes: <div>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.</div> <div>2) Values not reset by this keyword remain unaltered.</div> <div>3) The keyword is terminated by “/”.</div>					

Table 7.4: 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.

7.3.29 TRANY - DEFINE THE TRANSMISSIBILITY IN THE Y DIRECTION FOR ALL THE CELLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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	
I	TRANY	TRANY is an array of real positive numbers assigning the permeability in theY direction to each cell in the model. Repeat counts may be used, for example 20*100.0.			None
		cP.rb/day/psi	cP.rm ³ /day/bars	cP.rcc/hr/atm	
Notes: <div><div>1)</div><div>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.</div></div> <div><div>2)</div><div>Values not reset by this keyword remain unaltered.</div></div> <div><div>3)</div><div>The keyword is terminated by “/”.</div></div>					

Table 7.5: 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.

7.3.30 TRANZ - DEFINE THE TRANSMISSIBILITY IN THE Z DIRECTION FOR ALL THE CELLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

TRANZ 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 permeability in the Z direction to each cell in the model. Repeat counts may be used, for example 20*100.0.			None
		cP.rb/day/psi	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 7.6: 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

To be written in a future release of the manual.

8.2 DATA REQUIREMENTS

To be written in a future release of the manual.

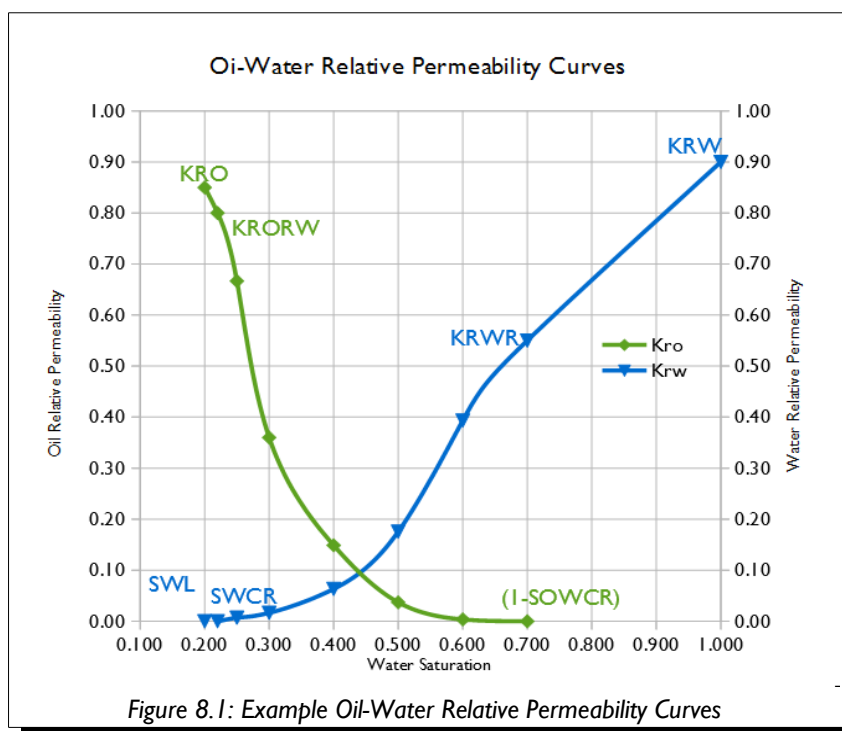
8.2.1 FLUID PROPERTY TABLES

To be written in a future release of the manual.

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 8.1 indicating the oil end-point data (KRO, KRORW and $(1 - \text{SOWCR})$) and the water end-point data (KRWR, KRW, SWL and SWCR).

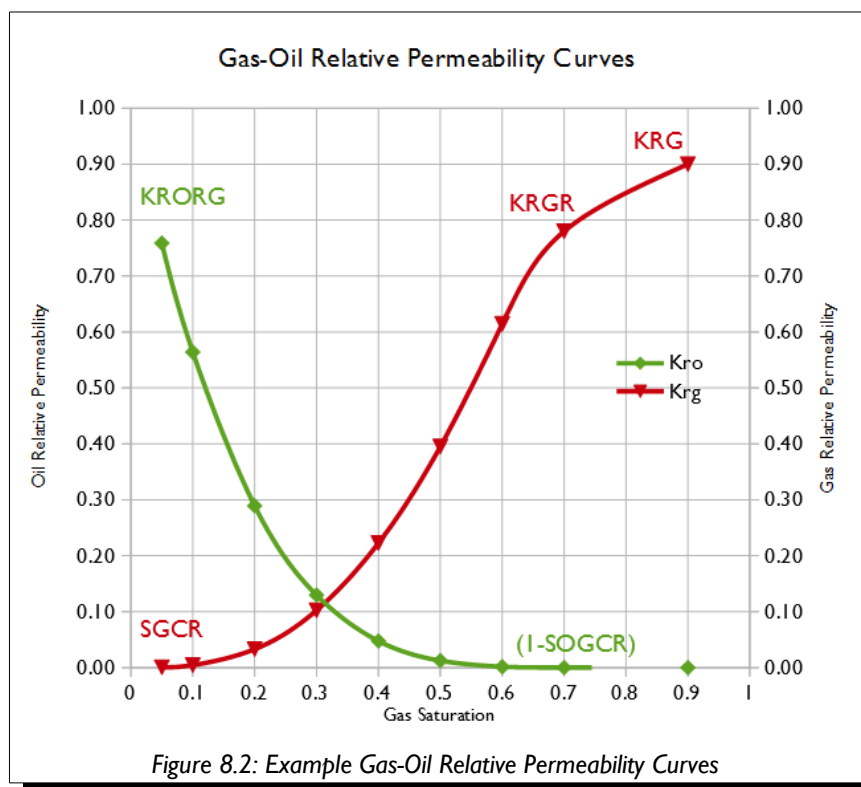


The associated oil-water end-point definitions are outlined in the following table:

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.

Table 8.1: Oil-Water Relative Permeability End-Point Data Definitions

Similarly for gas-oil systems, Figure 8.2 illustrates a typical gas-oil relative permeability set of curves indicating the oil end-point data (KRORG and $(1 - \text{SOGCR})$) and the gas end-point data (KRGR, KRG and SGCR).



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

Type	End-Point Keyword	Gas-Oil End-Point Definitions
Saturation	SGCR	Critical gas saturation, that is the largest gas saturation for which the gas relative permeability is zero.
	SOGCR	Critical oil-in-gas saturation, that is the largest oil saturation for which the oil relative permeability is zero in an oil-gas-concrete 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.

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

End-point scaling is activated in the RUNSPEC section with the ENDSKALE keyword and the data used to apply end-point scaling is entered in the PROPS section using the end-point keywords defined in Table 8.1 and Table 8.2 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 \pm , SWCRX \pm and SWCRX \pm 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: <ol style="list-style-type: none"> 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 8.3: 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
I	SALTCON	Field	Metric	Laboratory	None
		A real positive columnar vector that sets the salt concentrations for the given relative permeability saturation tables.			
		lb/stb	kg/sm ³	gm/scc	

Notes:

I) Each data set must be terminated by a “/” including the last data set.

Table 8.4: 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 ASPKDAM – DEFINE ASPHALTENE PERMEABILITY DAMAGE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

Description

This keyword defines the data required to model permeability damage, that is permeability reduction, due to the deposit of asphaltene coming out solution. This modeling option is not implemented in OPM Flow as this feature is normally incorporated in compositional simulators.

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

8.3.5 ASPPW2D – DEFINE ASPHALTENE TWO PARAMETERS PRECIPITATION DATA

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

Description

This keyword defines the data required to model asphaltene precipitation case where the precipitation is a function of pressure and temperature based on the percentage molar weight of one or more specified components. The characterization is specified by the first item of the ASPHALTE keyword, and the component range is specified by the ASPFLOC keyword. This modeling option is not implemented in OPM Flow as this feature is normally incorporated in compositional simulators.

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

8.3.6 ASPREWG -DEFINE ASPHALTENE AS PERCENTAGE WEIGHT

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

Description

This keyword defines the data required to model asphaltene precipitation case where the precipitation is a function of pressure based on the percentage molar weight of one or more specified components. The characterization is specified by the first item of the ASPHALTE keyword, and the component range is specified by the ASPFLOC keyword. This modeling option is not implemented in OPM Flow as this feature is normally incorporated in compositional simulators.

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

8.3.7 ASPWETF – DEFINE ASPHALTENE WETTABILITY FACTOR DATA

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

Description

This keyword defines the wettability factor as a function of asphaltene deposition. This modeling option is not implemented in OPM Flow as this feature is normally incorporated in compositional simulators.

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

8.3.8 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.

8.3.9 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.

8.3.10 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.

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

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

DENSITY defines the oil, water and gas surface densities for the fluids for various regions in the model. The number of DENSITY vector data sets is defined by the NTPVT parameter on the TABDIMS keyword in the RUNSPEC section and the allocation of the DENSITY data sets to different grid blocks in the model is done via the PVTNUM keyword in the 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.			Defined
		lb/ft ³ 600	kg/m ³ 600	gm/cc 600	
2	WATDEN	WATDEN is a real number defining the density of water at surface conditions.			Defined
		lb/ft ³ 999.014	kg/m ³ 999.014	gm/cc 999.014	
3	GASDEN	GASDEN is a real number defining the density of gas at surface conditions.			Defined
		lb/ft ³ 1.000	kg/m ³ 1.000	gm/cc 1.000	
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 8.5: DENSITY Keyword Description

According to the SPE SI standard²⁷, **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 uses 14.7 psia and 60 °F, while for SI units some areas use 101.325 kPa and 15 °C.

See also the GRAVITY keyword.

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
```

²⁷ 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.

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.

8.3.12 EHSTR – DEFINE HYSTERESIS MODEL AND PARAMETERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	--------------	-------------------------	--------------------------	-------------------------	--------------------------

Description

The EHSTR 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 Carlson²⁸ and Killough²⁹ 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	<div>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:</div> <table><tr><th colspan="3">Water Wet Hysteresis Models</th></tr><tr><th>HYSMOD</th><th>Non-Wetting Phases</th><th>Wetting Phase</th></tr><tr><td>-1</td><td colspan="2">Equilibration option for equilibrating the model with the SATNUM (drainage curves) and running the model with imbibition curves (IMBNUM). This option implies no hysteresis.</td></tr><tr><td>0</td><td>Carlson Hysteresis Model</td><td>SATNUM</td></tr><tr><td>1</td><td>Carlson Hysteresis Model</td><td>IMBNUM</td></tr><tr><td>2</td><td>Killough Hysteresis Model</td><td>SATNUM</td></tr><tr><td>3</td><td>Killough Hysteresis Model</td><td>IMBNUM</td></tr><tr><td>4</td><td>Killough Hysteresis Model</td><td>Killough Hysteresis Model</td></tr><tr><th colspan="3">Oil Wet to Water Wet Water Wet</th></tr><tr><td>5</td><td>Carlson Non- Wetting Modeling for Gas and Water</td><td>SATNUM</td></tr><tr><td>6</td><td>Killough Non- Wetting Modeling for Gas and Water</td><td>SATNUM</td></tr><tr><td>7</td><td>Killough Non- Wetting Modeling for Gas and Water</td><td>Killough Non- Wetting Modeling for the Wetting Oil Phase</td></tr></table> <div>Note only the default value of zero is supported by OPM Flow.</div>	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	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																																					

²⁸ 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).

²⁹ 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: <ol style="list-style-type: none"> 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. <ol style="list-style-type: none"> 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: <ol style="list-style-type: none"> 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: <ol style="list-style-type: none"> 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 a non-wetting phase. This option is ignored by OPM Flow.	None
9		Not used	

No.	Name	Description	Default
I0		Not used	
I1		Not used	
I2		Not used	
I3		Not used	
Notes: I) The keyword is terminated by "/".			

Table 8.6: 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).

8.3.13 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.

8.3.14 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 versus depth for the three phases. This functionality is not supported in OPM Flow.

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

8.3.15 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. This functionality is not supported in OPM Flow.

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

8.3.16 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.

8.3.17 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.

8.3.18 FILLEPS – ACTIVATE SATURATION END-POINT EXPORT TO THE INIT FILE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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

There is no data required for this keyword.

This keyword is not supported by OPM Flow but is documented here for completeness.

Example

```
--
--      ACTIVATES 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.

8.3.19 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.

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 ($^{\circ}API$), the relationship between relative density (γ_o) of oil and API gravity (γ_{API}) is given by: $\gamma_{API} = \frac{141.5}{\gamma_o} - 131.5$			None
		$^{\circ}API$	$^{\circ}API$	$^{\circ}API$	
2	WATGRAV	WATGRAV is a real number defining the density of water at surface conditions.			Defined
		(water = 1.0) 0.7773	(water = 1.0) 0.7773	(water = 1.0) 0.7773	
3	GRAVGAS	GRAVGAS is a real number defining the density of gas at surface conditions.			Defined
		(air = 1.0) 1.000	(air = 1.0) 1.000	(air = 1.0) 1.000	
Notes: 1) The each data set terminated by terminated by “/” at the end of the line, there is no “/” terminator for the keyword.					

Table 8.7: GRAVITY Keyword Description

According to the SPE SI standard³⁰, **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 uses 14.7 psia and 60 °F, while for SI units some areas use 101.325 kPa and 15 °C.

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.

³⁰ 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.

8.3.20 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. This functionality is not supported in OPM Flow.

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

8.3.21 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. This functionality is not supported in OPM Flow.

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

8.3.22 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 for all grid types.

The capillary pressure for a grid block is scaled by:

$$P_c = P_{c_{TABLE}} \left(\frac{IPCG}{P_{c_{TABLE-MAX}}} \right) \quad (8.1)$$

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 $S_g = 1 - S_{wco}$.

No.	Name	Description			Default
		Field	Metric	Laboratory	
I	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.			None
		psia	bars	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) The keyword is terminated by “/”.

Table 8.8: IPCG Keyword Description

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 a IPCG for 300 cells in the model as defined by the DIMENS keyword in the RUNSPEC section.

8.3.23 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 HYSSTER option on the SATOPTS keyword in the RUNSPEC section has to be activated to invoke the hysteresis option. The keyword can be used for all grid types.

The capillary pressure for a grid block is scaled by:

$$P_c = P_{c_{TABLE}} \left(\frac{IPCW}{P_{c_{TABLE-MAX}}} \right) \quad (8.2)$$

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).

No.	Name	Description			Default
		Field	Metric	Laboratory	
I	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.			None
		psia	bars	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) The keyword is terminated by “/”.

Table 8.9: IPCW Keyword Description

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 a IPCW for 300 cells in the model as defined by the DIMENS keyword in the RUNSPEC section.

A	B	C	D	E	F	G	H	I	J	K	L	M	N	O	P	Q	R	S	T	U	V	W	X	Y	Z
---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---

8.3.24 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 for all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
I	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) Note this the non-direction dependent 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 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 8.10: 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 endpoint 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-, 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.

A	B	C	D	E	F	G	H	I	J	K	L	M	N	O	P	Q	R	S	T	U	V	W	X	Y	Z
---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---

8.3.25 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 for all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
I	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) Note this the non-direction dependent 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 ISGLZ± 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 8.11: 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 endpoint 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-, 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.

A	B	C	D	E	F	G	H	I	J	K	L	M	N	O	P	Q	R	S	T	U	V	W	X	Y	Z
---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---

8.3.26 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 for all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
I	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) Note this the non-direction dependent 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± , ISGUY± and ISGU± 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 8.12: 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 endpoint 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-, 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.

8.3.27 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 for all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
I	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) Note this the non-direction dependent version of the critical gas saturation used with the end-point scaling option. If directional end-point scaling has been activated then the ISOGCRX± , ISOGCRY± and ISOGCRZ± 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 8.13: 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 endpoint 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-, 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.

8.3.28 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 ENDSALE 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 for all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
I	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: <div>1) Note this the non-direction dependent version of the critical gas saturation used with the end-point scaling option. If directional end-point scaling has been activated then the ISOWCRX± , ISOWCRY± and ISOWCRZ± series of keyword should be used.</div> <div>2) If the value for a cell has been defaulted then OPM Flow users the value from the cell's relative permeability table.</div> <div>3) The keyword is terminated by “/”.</div>					

Table 8.14: 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 endpoint 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-, 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

8.3.29 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 for all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
I	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) Note this the non-direction dependent version of the critical water saturation used with the end-point scaling option. If directional end-point scaling has been activated then the ISWCRX \pm , ISWCRY \pm and ISWCRZ \pm 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 8.15: 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 endpoint 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-, 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.

A	B	C	D	E	F	G	H	I	J	K	L	M	N	O	P	Q	R	S	T	U	V	W	X	Y	Z
---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---

8.3.30 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 for all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
I	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) Note this the non-direction dependent 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 ISWLZ± 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 8.16: 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 endpoint 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-, 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.

8.3.31 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 for all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
I	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) Note this the non-direction dependent 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± , ISWUY± and ISWUZ± 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 8.17: 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 endpoint 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-, 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.

8.3.32 KRG – END-POINT SCALING OF GRID CELL KRG(SGU) (DRAINAGE)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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 ENDSKALE 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 “/”.

Table 8.18: KRG Keyword Description

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
--
--      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
/
```

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:

$$k_{rg} = k_{rg\ TABLE} \left(\frac{KRG}{k_{rg\ TABLE-MAX}} \right) \quad (8.3)$$

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 8.19: 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 endpoint 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 endpoint 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 hysteresis 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.

8.3.33 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 ($I - SOGCR$), or critical water saturation in a gas-water run (S_{wc}), for all the cells in the model via an array. The ENDSKALE 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	
I	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 KGRGT should be less than KRGR. 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 “/”.

Table 8.20: KRGR Keyword Description

Examples

The first example defines an input box for the whole grid and for layers one to three, for layer one KRGR is set equal to 0.500, for layer two KRGR equals 0.570, and for layer three KRGR equals 0.580.

```
--
--      DEFINE INPUT BOX FOR EDITING INPUT ARRAYS (NX=100, NY=100)
--
--      ----- BOX -----
--      I1  I2    J1  J2    K1  K2
BOX      1*  1*    1*  1*    1    3
--
--      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
/
```

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 8.21: 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 endpoint 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 endpoint 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 hysteresis 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.

8.3.34 KRO – END-POINT SCALING OF GRID CELL $K_{ro}(SWL)$ (DRAINAGE)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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 “/”.

Table 8.22: KRO Keyword Description

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
/
```

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:

$$k_{ro} = k_{ro\ TABLE} \left(\frac{KRO}{k_{ro\ TABLE-MAX}} \right) \quad (8.4)$$

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 8.23: 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 endpoint 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-, 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 endpoint 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-, KROY-, KROZ and KROZ-, instead of the KRO keyword.

If hysteresis 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.

8.3.35 KRORG – END-POINT SCALING OF GRID CELL $K_{ro}(S_{gcr})$ (DRAINAGE)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

KRORG defines the scaling parameter the drainage relative permeability of oil at the critical gas saturation (S_{gcr}), 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 “/”.

Table 8.24: KRORG Keyword Description

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
/
```

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 8.25: 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 endpoint 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-, 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, KRORGX, 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 endpoint 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-, KRORGY-, KRORGZ and KRORGZ-, instead of the KRORG keyword.

If hysteresis 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.

8.3.36 KRORW – END-POINT SCALING OF GRID CELL $K_{ro}(SWCR)$ (DRAINAGE)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

KRORW defines the scaling parameter the drainage relative permeability of oil at the critical water saturation (SWCR), for all the cells in the model via an array. The ENDSALE 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 “/”.

Table 8.26: KRORW Keyword Description

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
/
```

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 8.27: 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 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 endpoint 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 endpoint 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 hysteresis 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.

8.3.37 KRW – END-POINT SCALING OF GRID CELL KRW(Sw =1.0) (DRAINAGE)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

KRW defines the scaling parameter at the maximum drainage oil relative permeability value (SWU), that is for $S_w = 1.0$, for all the cells in the model via an array. The ENDSKALE 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 “/”.

Table 8.28: KRW Keyword Description

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
--
--      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
/
```

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:

$$k_{rw} = k_{rw_{TABLE}} \left(\frac{KRW}{k_{rw_{TABLE-MAX}}} \right) \quad (8.5)$$

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 8.29: 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 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 endpoint 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 endpoint 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 hysteresis 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.

8.3.38 KRWR – END-POINT SCALING OF GRID CELL KRWR($S_w = 1.0$) (DRAINAGE)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

KRWR defines the scaling parameter at the maximum drainage oil relative permeability value (S_w), 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	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 “/”.

Table 8.30: KRWR Keyword Description

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
--
--      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
/
```

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 8.31: 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 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 endpoint 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-, 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, KRWR Y and KRWRZ instead of KRWR. There is also the facility to make the directional endpoint 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-, KRWR Y-, KRWRZ-, KRWRZ- and KRWRZ-, instead of the KRWR keyword.

If hysteresis 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.

8.3.39 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.

8.3.40 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.

8.3.41 MSFN – MISCIBLE NORMALIZED RELATIVE PERMEABILITY TABLES

<i>RUNSPEC</i>	<i>GRID</i>	<i>EDIT</i>	<i>PROPS</i>	<i>REGIONS</i>	<i>SOLUTION</i>	<i>SUMMARY</i>	<i>SCHEDULE</i>
----------------	-------------	-------------	--------------	----------------	-----------------	----------------	-----------------

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). Why 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 stating from 0 and terminating at 1.0, 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 0.0 and less than or equal to 1.0 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 0.0 and less than or equal to 1.0 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 8.32: 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.

A	B	C	D	E	F	G	H	I	J	K	L	M	N	O	P	Q	R	S	T	U	V	W	X	Y	Z
---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---

8.3.42 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.

8.3.43 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.

8.3.44 OILVISCT – OIL VISCOSITY VERSUS TEMPERATURE FUNCTIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

OILVISCT defines the oil viscosity as a function of temperature for when temperature option has been activated by the TEMP 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.

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.			None
		°F	°C	°C	
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 PRS and RS variables on the VISCREF keyword.			None
		cP	cP	cP	

Notes:

1)

The keyword is followed by NTPVT tables as declared on the TABDIMS keyword in the RUNSPEC section.

2)

Each table must contain a minimum of two rows and a maximum of NPPVT rows as declared on the TABDIMS keyword in the RUNSPEC section.

1)

Each table is terminated by “/” and there is no “/” terminator for the keyword.

Table 8.33: OILVISCT Keyword Description

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 the OILVISCT keyword for when the temperature 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.

```
--
--      OIL VISCOSITY VERSUS TEMPERATURE TABLES
--
--      WATER      WATER
--      TEMP       VISC
--      -----
OILVISCT
      3000.0      0.600      / TABLE NO. 01
      3200.0      0.650      / TABLE NO. 02
      3300.0      0.680      / TABLE NO. 03
      3400.0      0.720      / TABLE NO. 04
      3500.0      0.725      / TABLE NO. 05
```

There is no terminating "/" for this keyword.

8.3.45 OPERATE – DEFINE MATHEMATICAL OPERATIONS ON ARRAYS

This keyword defines the 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.

See [OPERATE – Define Mathematical Operations on Arrays](#) in the GRID section for a full description.

8.3.46 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 for all grid types.

The capillary pressure for a grid block is scaled by:

$$P_c = P_{c_{TABLE}} \left(\frac{PCG}{P_{c_{TABLE-MAX}}} \right) \quad (8.6)$$

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 $S_g = 1 - S_{wco}$.

No.	Name	Description			Default
		Field	Metric	Laboratory	
I	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.			None
		psia	bars	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) 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 8.34: PCG Keyword Description

See also the IPCG keyword for the equivalent imbibition functionality.

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 a PCW for 300 cells in the model as defined by the DIMENS keyword in the RUNSPEC section.

8.3.47 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 ENDSKALE keyword in the RUNSPEC section should be activated to enable end-point scaling and the use of this keyword. The keyword can be used for all grid types. The capillary pressure for a grid block is scaled by:

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

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).

No.	Name	Description			Default
		Field	Metric	Laboratory	
I	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.			None
		psia	bars	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) 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 8.35: PCW Keyword Description

See also the PCW keyword for the equivalent imbibition functionality.

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 a PCW for 300 cells in the model as defined by the DIMENS keyword in the RUNSPEC section.

A	B	C	D	E	F	G	H	I	J	K	L	M	N	O	P	Q	R	S	T	U	V	W	X	Y	Z
---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---

8.3.48 PLMIXPAR – DEFINE THE POLYMER TODD-LONGSTAFF MIXING PARAMETERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

The PLMIXPAR keyword defines the Todd-Longstaff³¹ 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.			None
		dimensionless	dimensionless	dimensionless	
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 8.36: 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.

³¹ 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.

8.3.49 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.			None
		lb/stb	kg/sm ³	gm/scc	
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.			None
		lb/lb	kg/kg	gm/gm	

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 8.37: 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.

--	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

There is no terminating “/” for this keyword.

8.3.50 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 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.			None
		lb/stb	kg/sm ³	gm/scc	
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 “/”			None
		lb/lb	kg/kg	gm/gm	

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 8.38: 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

```
--
--      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
--
--      PLYADS
--      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.

8.3.51 PLYDHFLF - DEFINE POLYMER THERMAL DEGRADATION HALF-LIFE TABLES

<i>RUNSPEC</i>	<i>GRID</i>	<i>EDIT</i>	<i>PROPS</i>	<i>REGIONS</i>	<i>SOLUTION</i>	<i>SUMMARY</i>	<i>SCHEDULE</i>
----------------	-------------	-------------	--------------	----------------	-----------------	----------------	-----------------

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.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	TEMP	A columnar vector of real monotonically increasing down the column values that defines the polymer concentration in the solution surrounding the rock.			None
		°F	°C	°C	
2	POLHFLF	A columnar vector of real values that defines the corresponding polymer half-life.			None
		days	days	hours	

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 8.39: PLYDHFLF Keyword Description

This keyword is not supported by OPM Flow but is documented here for completeness, as the thermal option is currently under development.

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 above 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.

8.3.52 PLYMAX - DEFINE POLYMER-SALT VISCOSITY MIXING CONCENTRATIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

The PLYMAX keyword defines maximum polymer and salt concentrations that are to be used in the mixing parameter calculation of the fluid component viscosities, for when the polymer option has been activated by the POLYMER keyword in the RUNSPEC section.

Note that If the BRINE option has not be activated by the BRINE keyword in the RUNSPEC section, then the salt concentrations in the second column are ignored.

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.			None
		lb/stb	kg/sm ³	gm/scc	
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.			None
		lb/stb	kg/sm ³	gm/scc	

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 8.40: 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.

8.3.53 PLYROCK - DEFINE POLYMER-ROCK PROPERTIES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	--------------	-------------------------	--------------------------	-------------------------	--------------------------

Description

The PLYROCK keyword defines rock properties for when the polymer option has been activated by the POLYMER keyword in the RUNSPEC section.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	PSPACE	A real positive value that is greater than or equal to zero and less than one, that defines available pore space for this rock type.			None
		dimensionless	dimensionless	dimensionless	
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.			None
		dimensionless	dimensionless	dimensionless	
3	DENSITY	A real value that defines the rock in-situ density, that is at reservoir conditions.			None
		lb/rtb	kg/rm ³	gm/rcc	
4	ADINDX	A positive integer of 1 or 2 that defines desorption option. <div>1) then polymer desorption may occurs by retracing the polymer adsorption isotherm when the local polymer concentration in the solution decreases.</div> <div>2) then no polymer desorption may occurs</div>			Defined
		Dimensionless 1	Dimensionless 1	Dimensionless 1	
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.			None
		lb/lb	kg/kg	gm/gm	

Notes:

1) The keyword is followed by NPLMIX tables as declared on the REGDIMS 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 8.41: 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 REGDIMS keyword in the RUNSPEC section being equal to three.

8.3.54 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.			None
		feet/day	m/day	cm/hour	
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.			None
		dimensionless	dimensionless	dimensionless	

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 8.42: 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.

8.3.55 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.

See the PLYSHEAR keyword for the alternative polymer shear thinning/thickening option that is implemented in OPM Flow.

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

8.3.56 PLYVISC – DEFINE POLYMER VISCOSITY SCALING FACTORS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

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.

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.			None
		lb/stb	kg/sm ³	gm/scc	
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.			None
		dimensionless	dimensionless	dimensionless	
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 8.43: 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
--      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. 01
--
-- / TABLE NO. 02
```


The above 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.

8.3.57 PMISC – DEFINE MISCIBILITY VERSUS PRESSURE TABLES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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 been 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	PRS	A columnar vector of real monotonically increasing down the column values that defines the oil phase pressure.			None
		psia	barsa	atma	
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 between zero and one, where zero means no miscibility and one means full miscibility.			None
		dimensionless	dimensionless	dimensionless	

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 8.44: PMISC Keyword Description

Example

```
--
--      MISCIBILITY VERSUS PRESSURE TABLES
--
PMISC
--      OIL      MISCIBLE
--      PRESS    FACTOR
--      -----
--      1000.0    0.000
--      2000.0    0.250
--      3000.0    1.000
--      4000.0    1.000
--
--      OIL      MISCIBLE
--      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. 01
--
--      / 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.

8.3.58 PROPS - DEFINE THE START OF THE PROPS SECTION OF KEYWORDS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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.

8.3.59 PVCD0 - OIL PVT PROPERTIES FOR DEAD OIL (CONSTANT COMPRESSIBILITY)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	--------------	-------------------------	--------------------------	-------------------------	--------------------------

Description

PVCD0 defines the oil PVT properties for dead oil³² 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 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.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	PRS	PRS is a real positive value defining the oil reference pressure for the other parameters for this data set.			None
		psia	barsa	atma	
2	OFVF	OFVF is a real positive value defining the oil formation volume factor (Bo) at the reference pressure.			None
		rb/stb	rm³/sm³	rcc/scc	
3	OCOMP	OCOMP is a real positive value defining the oil compressibility (Co) at the oil reference pressure and is defined as: $C_o = -\frac{1}{B_o} \left(\frac{dB_o}{dP} \right)$			None
		l/psia	l/barsa	l/atma	
4	OVISC	OVISC is a real positive value defining the oil viscosity (μo) at the oil reference pressure.			None
		CP	CP	CP	
5	OVISCOMP	OVISCOMP is a real positive value defining the oil viiscosibility (μoc) at the oil reference pressure, Uwc(Pref) and is defined as: $\mu_{oc} = -\frac{1}{\mu_o} \left(\frac{d\mu_o}{dP} \right)$			None
		l/psia	l/barsa	l/atma	
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 8.45: PVCD0 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.

³² “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.

8.3.60 PVDG - Gas PVT PROPERTIES FOR DRY GAS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

PVDG defines the gas PVT properties for dry gas³³. 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	PRS	A columnar vector of real monotonically increasing down the column values that defines the gas phase pressure.			None
		psia	barsa	atma	
2	GFVF	A columnar vector of real decreasing down the column values that defines the corresponding gas phase formation volume factor.			None
		rb/Mscf	rm ³ /sm ³	rcc/scc	
3	GVISC	A columnar vector of real increasing down the column values that defines the corresponding gas phase viscosity.			None
		cP	cP	cP	

Notes:

- 1) The keyword is followed by NTPVT tables as declared on the TABDIMS keyword in the RUNSPEC section.
- 2) Each table must contain a minimum of two rows and a maximum of NPPVT rows 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 8.46: PVDG Keyword Description

See also the RVCONST and RVCONSTT keywords to define the constant Rv for dry gas.

³³ 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.

8.3.61 PVDO – OIL PVT PROPERTIES FOR DEAD OIL

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	--------------	-------------------------	--------------------------	-------------------------	--------------------------

Description

PVDO defines the oil PVT properties for dead oil³⁴. 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 RCONST or RCONSTT 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.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	PRS	A columnar vector of real monotonically increasing down the column values that defines the oil phase pressure.			None
		psia	barsa	atma	
2	OFVF	A columnar vector of real decreasing down the column values that defines the corresponding oil phase formation volume factor.			None
		rb/stb	rm ³ /sm ³	rcc/scc	
3	OVISC	A columnar vector of real increasing down the column values that defines the corresponding oil phase viscosity.			None
		cP	cP	cP	

Notes:

- 1) The keyword is followed by NTPVT tables as declared on the TABDIMS keyword in the RUNSPEC section.
- 2) Each table must contain a minimum of two rows and a maximum of NPPVT rows 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 8.47: PVDO Keyword Description

See also the RCONST and RCONSTT keywords to define the constant Rs for dead oil and PVCDO as an alternative keyword to enter the dead oil properties.

³⁴ “Dead” oil is oil that it contains no dissolved gas or a relatively thick oil or residue that has lost its volatile components.


```
--
--      OIL PVT TABLE FOR DEAD OIL
```

	PSAT PSIA	B0 RB/STB	VISC 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.

8.3.62 PVDS - SOLVENT PVT PROPERTIES FOR THE SOLVENT MODEL

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	------------------------------	-------------------------	--------------------------	-------------------------	--------------------------

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	PRS	A columnar vector of real monotonically increasing down the column values that defines the solvent phase pressure.			None
		psia	barsa	atma	
2	GVFV	A columnar vector of real decreasing down the column values that defines the corresponding solvent phase formation volume factor.			None
		rb/Mscf	rm ³ /sm ³	rcc/scc	
3	GVISC	A columnar vector of real increasing down the column values that defines the corresponding solvent phase viscosity.			None
		cP	cP	cP	

Notes:

- 1) The keyword is followed by NTPVT tables as declared on the TABDIMS keyword in the RUNSPEC section.
- 2) Each table must contain a minimum of two rows and a maximum of NPPVT rows 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 8.48: 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
--
--
--      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 NO. 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 NO. 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.

8.3.63 PVTG - GAS PVT PROPERTIES FOR WET GAS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

PVTG defines the gas PVT properties for wet gas³⁵. This keyword should be used when the VAPOIL keyword has been 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	PRS		<p>A real monotonically increasing down the column values that defines the gas phase pressure. that defines the saturated condensate-gas ratio ("CGR") or R_v, the gas formation volume factor and the gas viscosity for the corresponding pressure for the stated saturated RVS.</p> <p>For a given PRS the variability of the gas formation volume factor and the gas viscosity with respect to the under-saturated R_v is optionally included as a sub table under RVU, FVFU and VISU columns, that is it is not necessary to repeat PRS for each sub table entry. However, each sub table must be terminated by a "/".</p> <p>The under saturated R_v entries are optional, except for perhaps the last PRS entry to define the PVT properties above the initial saturation pressure.</p>			None
			psia	barsa	atma	
2	RVS	RVU	<p>A columnar vector of real positive number for both the saturated (RVS) and under saturated (RVU) R_v sub table entries.</p> <p>The RVS entry on the main table is the saturated CGR at the pressure indicated by PRS and may be increasing or decreasing in value as PRS varies.</p> <p>Subsequent under-saturated R_vs for a sub table at the given PRS, as defined by RVU, are monotonically decreasing for entries in a given sub table.</p>			None
			stb/Mscf	sm^3/sm^3	rcc/scc	
3	FVFS	FVFU	<p>A columnar vector of real decreasing down the column values that defines the corresponding gas phase formation volume factor for a given pressure (PRS) and for a given R_v (either RVS or RVU).</p>			None
			rb/Mscf	rm^3/sm^3	rcc/scc	
4	VISS	VISU	<p>VISS a columnar vector of real increasing down the column values that defines the corresponding gas phase viscosity for a given pressure (PRS) and for a given RVS.</p> <p>VISU a columnar vector of real decreasing from VISS down the column values that defines the corresponding gas phase viscosity for a given pressure (PRS) and for a given RVU.</p>			None
			cP	cP	cP	

³⁵ 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 8.49: 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.

8.3.64 PVTO - OIL PVT PROPERTIES FOR LIVE OIL

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	--------------	-------------------------	--------------------------	-------------------------	--------------------------

Description

PVTO defines the oil PVT properties for live³⁶ and the keyword should only be used if there is both oil and gas phases in the model. This keyword should be used when the DISGAS keyword has been 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		<p>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.</p> <p>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 "/".</p> <p>The under-saturated PRSU entries are optional, except for perhaps the last RS entry to define the PVT properties above the initial saturation pressure.</p> <p>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 "/".</p>			None
			Mscf/stb	sm ³ /sm ³	scc/scc	
2	PRSS	PRSU	<p>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.</p> <p>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.</p> <p>Note that PRSU should be greater than PRSS.</p>			None
			psia	barsa	atma	
3	FVFS	FVFU	<p>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.</p> <p>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.</p>			None
			rb/stb	rm ³ /sm ³	rcc/scc	

³⁶ "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 “/”.			None
			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	

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 8.50: 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      B0      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	B0	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.

8.3.65 PVTW - DEFINE WATER FLUID PROPERTIES FOR VARIOUS REGIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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.			None
		psia	barsa	atma	
2	WFVF	WFVF is a real number defining the water formation volume factor (Bw) at the water reference pressure.			Defined
		rb/stb 1.0	rm³/sm³ 1.0	rcc/scc 1.0	
3	WCOMP	WCOMP is a real number defining the water compressibility (Cw) at the water reference pressure and is defined as: $C_w = -\frac{1}{B_w} \left(\frac{dB_w}{dP} \right)$			Defined
		l/psia 0.00004	l/barsa 0.00004	l/atma 0.00004	
4	WVISC	WVISC is a real number defining the water viscosity (μw) at the water reference pressure			Defined
		CP 0.50	CP 0.50	CP 0.50	
5	WVISCMP	WVISCMP is a real number defining the water viiscosity (μwc) at the water reference pressure, Uwc(Pref) and is defined as: $\mu_{wc} = -\frac{1}{\mu_w} \left(\frac{d\mu_w}{dP} \right)$			Defined
		l/psia 0.0	l/barsa 0.0	l/atma 0.0	
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 8.51: 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. 01
--      4940.0    1.030    2.8E-6    0.390    1*          / TABLE NO. 01
```

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.

8.3.66 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.

8.3.67 ROCK - DEFINE THE ROCK COMPRESSIBILITY FOR VARIOUS REGIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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.			Default
		psia 1.032	barsa 1.032	atma 1.032	
2	RCOMP	RCOMP is a real number defining the rock compressibility (Cf) at the rock reference pressure and is defined as: $C_f = -\frac{1}{V}\left(\frac{dV}{dP}\right)$			Defined
		l/psia 0.0	l/barsa 0.0	l/atma 0.0	

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 8.52: 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.

8.3.68 ROCKOPTS – DEFINE ROCK COMPACTION AND COMPRESSIBILITY OPTIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

The ROCKOPTS keyword defines various option with respect to rock compaction and rock compressibility.

No.	Name	Description	Default
1	ROCKOPT1	<p>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:</p> <ol style="list-style-type: none"> 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. <p>ROCKOPT1 should be set to PRESSURE if the OVERBURD is not used in the input deck.</p>	PRESSURE
2	ROCKOPT2	<p>ROCKOPT2 is a character string that sets the reference pressure option:</p> <ol style="list-style-type: none"> 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. <p>Note that STORE option should not be used with the OVERBURD keywords as the OVERBURD data will be overwritten.</p>	NOSTORE
3	ROCKOPT3	<p>ROCKOPT3 is a character string that defines which region array should be used to allocate the various ROCK and ROCKTAB tables. ROCKOPT3, should be set to ROCKNUM, SATNUM or PVTNUM.</p>	PVTNUM
4	ROCKOPT4	<p>ROCKOPT4 is a character string that sets the initial conditions for the HYSTER and BOBERG options:</p> <ol style="list-style-type: none"> 1) DEFLATION: This option defines the reservoir rock to be fully compacted and the deflation curve is used to calculate 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
<p>Notes:</p> <ol style="list-style-type: none"> 1) The keyword is terminated by “/”. 			

Table 8.53: 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.

8.3.69 ROCKTAB – Rock Compaction Tables

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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 should only be used if compaction option has been enabled.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	PRS	If the ROCKOPT1 variable has been set to PRESSURE on the ROCKOPTS keyword in the PROPS section, then PRS should be a columnar vector of real monotonically <u>increasing</u> down the column values, that define the reference pressure for which the other parameters correspond to.			None
		If ROCKOPT1 has been set to STRESS, then PRS should be a columnar vector of real monotonically <u>decreasing</u> down the column values.			
		psia	bars	atm	
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 PRS.			None
		dimensionless	dimensionless	dimensionless	
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 PRS.			None
		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 PRS.			
		dimensionless	dimensionless	dimensionless	
4	TRANSY	If the RKTRMDIR is absent from the input deck, then TRANSY is ignored.			None
		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 PRS.			
		dimensionless	dimensionless	dimensionless	

No.	Name	Description			Default
		Field	Metric	Laboratory	
5	TRANSZ	If the RKTRMDIR is absent from the input deck, then TRANSZ is ignored.			None
		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 PRS.			
		dimensionless	dimensionless	dimensionless	

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 8.54: ROCKTAB 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 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 present in the input deck.

```
--
--      ROCK COMPACTION TABLES
--
ROCKTAB
--      PRESS      PORV      TX(YZ)      TY      TZ
--      MULT      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      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.

8.3.70 RPTPROPS – DEFINE PROPS SECTION REPORTING

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

This keyword defines the data in the PROPS section that is to be printed to the output print file in human readable format. The keyword has two distinct forms, the first of which consists of the keyword followed by a series of integers on the next line indicating the data to be printed (see the first example). This is the original format in the commercial simulator and was subsequently superseded by the second format. The second format consists of the keyword followed by a series of character strings that indicate the data to be printed. In most cases the character string is the keyword used to load the data in the OPM Flow input deck, for example PVDG for the dry gas PVT tables. It 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 8.55: 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 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
      PVD0      SOF2      SGFN      SWFN          /
```

8.3.71 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 temperature option has been activated by the TEMP keyword in the RUNSPEC section. The RTEMPA keyword is alias for RTEMP

No.	Name	Description			Default
		Field	Metric	Laboratory	
I	RTEMP	Single real positive value that define the reservoir temperature for the model.			None
		°F	°C	°C	
Notes: I) The keyword is terminated by “/”.					

Table 8.56: RTEMP Keyword Description

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

Example

```
--
--      RESERVOIR
--      TEMPERATURE
--      -----
RTEMP      190.0 / RESERVOIR TEMPERATURE
```

The above example defines the reservoir temperature to be 190 °F.

8.3.72 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 option has been activated by the TEMP keyword in the RUNSPEC section. The RTEMP keyword is alias for RTEMPA

No.	Name	Description			Default
		Field	Metric	Laboratory	
I	RTEMPA	Single real positive value that define the reservoir temperature for the model.			None
		°F	°C	°C	
Notes: 2) The keyword is terminated by “/”.					

Table 8.57: RTEMPA Keyword Description

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

Example

```
--
--      RESERVOIR
--      TEMPERATURE
--      -----
RTEMPA      190.0 / RESERVOIR TEMPERATURE
```

The above example defines the reservoir temperature to be 190 °F.

8.3.73 SALNODE – SALT CONCENTRATION BASED PVTNUM ARRAY

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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.

No.	Name	Description			Default
		Field	Metric	Laboratory	
I	SALTCON	A real monotonically increasing positive columnar vector defining the salt concentration for a given PVTNUM table.			None
		lb/stb	kg/sm ³	gm/scc	
Notes: I) Each table is terminated by “/” including the last table; however, there is no “/” terminator for the keyword.					

Table 8.58: 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.

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 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.

A	B	C	D	E	F	G	H	I	J	K	L	M	N	O	P	Q	R	S	T	U	V	W	X	Y	Z
---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---

8.3.74 SCALECRS – SET END-POINT SCALING OPTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

The SCALECRS keyword sets the end-point scaling option to be either two-point or three-point scaling. 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
I	SCALEOPT	SCALEOPT is a character string that sets the endpoint 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 8.59: 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 8.60: End-Point Arrays Used in the End-Point Scaling Options

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.

8.3.75 SDENSITY – DEFINE THE MISCIBLE OR SOLVENT SURFACE GAS DENSITY

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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	SOLDEN	SOLDEN is a real positive number defining the density at surface conditions of either: <div>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.</div>			None
		lb/ft ³	kg/m ³	gm/cc	
Notes: <div>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.</div>					

Table 8.61: 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
              0.64209                      / MIS-SOL DENSITY 3
```

There is no terminating "/" for this keyword.

8.3.76 SGCR – END-POINT SCALING GRID CELL CRITICAL GAS SATURATIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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 ENDSKALE 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 for 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 non-direction dependent version of the critical gas saturation used with the end-point scaling option. If directional end-point scaling has been activated then the SGCRX± , SGCRY± and SGCRZ± 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 8.62: 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 endpoint 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.

8.3.77 SGCWMIS – MISCIBLE CRITICAL GAS VERSUS WATER SATURATION FUNCTIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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 stating from 0 and terminating at 1.0, that defines the water saturation.			None
		dimensionless	dimensionless	dimensionless	
2	SGCMIS	A columnar vector of real equal or increasing down the column values that are greater than or equal to zero and less then 1.0, that define the corresponding miscible gas critical gas saturation for the corresponding water saturation SWAT.			None
		dimensionless	dimensionless	dimensionless	
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 8.63: SGCWMIS Keyword Description

Example

```
--
--      MISCIBLE CRITICAL GAS VERSUS WATER SATURATION TABLE
--
SGCWMIS
--      SWAT      SGRMIS
--      FRAC      FRAC
--      -----
--      0.0000    0.0000
--      0.2000    0.0300
--      1.0000    0.0300      / TABLE NO. 01
--      SWAT      SGRMIS
--      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.

8.3.78 SGFN – GAS SATURATION TABLES (FORMAT TYPE 2)

<i>RUNSPEC</i>	<i>GRID</i>	<i>EDIT</i>	<i>PROPS</i>	<i>REGIONS</i>	<i>SOLUTION</i>	<i>SUMMARY</i>	<i>SCHEDULE</i>
----------------	-------------	-------------	--------------	----------------	-----------------	----------------	-----------------

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 stating from 0 and terminating at 1.0, that defines the gas saturation.			None
		dimensionless	dimensionless	dimensionless	
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 0.0 and less than or equal to 1.0 that defines the gas relative permeability.			None
		dimensionless	dimensionless	dimensionless	
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.			None
		psia	bars	atm	

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 8.64: 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*
	0.30	0.0062	1*
	0.45	0.0450	1*
	0.50	0.0707	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.

A	B	C	D	E	F	G	H	I	J	K	L	M	N	O	P	Q	R	S	T	U	V	W	X	Y	Z
---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---

8.3.79 SGL – END-POINT SCALING GRID CELL CONNATE GAS SATURATIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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 ENDSKALE 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 for 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 non-direction dependent 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 SGLZ± 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 8.65: 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 endpoint 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-, 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.

8.3.80 SGOF – GAS-OIL SATURATION TABLES VERSUS GAS (FORMAT TYPE 1)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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 stating from 0 and terminating at 1.0, that defines the gas saturation.			None
		dimensionless	dimensionless	dimensionless	
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 0.0 and less than or equal to 1.0 that defines the gas relative permeability.			None
		dimensionless	dimensionless	dimensionless	
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 0.0 and less than or equal to 1.0 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 $k_{rog}(S_g = 0)$, must be the same as the first entry in the corresponding SWOF table, that is at $k_{row}(S_o = 1 - S_{wco})$. The last value in the column should be zero.			None
		dimensionless	dimensionless	dimensionless	
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.			None
		psia	bars	atm	

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 8.66: 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
--
--
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. 01
/ TABLE No. 02
```

The example defines two SGOF tables for use when oil, gas and water are present in the run.

8.3.81 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 for 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 non-direction dependent 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 \pm , SGUY \pm and SGUZ \pm 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 8.67: 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 endpoint 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-, 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.

8.3.82 SGWFN – GAS-WATER SATURATION TABLES (FORMAT TYPE 2)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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 stating from 0 and terminating at 1.0, that defines the gas saturation.			None
		dimensionless	dimensionless	dimensionless	
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 0.0 and less than or equal to 1.0 that defines the gas relative permeability. Note that the first entry in the column must be zero.			None
		dimensionless	dimensionless	dimensionless	
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 0.0 and less than or equal to 1.0 that defines the water relative permeability with respect to gas saturation. The last value in the column should be zero.			None
		dimensionless	dimensionless	dimensionless	
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.			None
		psia	bars	atm	

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 8.68: 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.

8.3.83 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.

8.3.84 SLGOF – GAS-OIL SATURATION TABLES VERSUS GAS (FORMAT TYPE 1)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

The SLGOF keyword defines the oil and gas relative permeability and oil-gas capillary pressure versus liquid saturation tables for when oil and gas are present in the input deck. This keyword should only be used if both oil and gas are present in the run.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	SLIQ	A columnar vector of real monotonically increasing down the column values stating from 0 and terminating at 1.0, 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.			None
		dimensionless	dimensionless	dimensionless	
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 0.0 and less than or equal to 1.0 that defines the gas relative permeability..			None
		dimensionless	dimensionless	dimensionless	
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 0.0 and less than or equal to 1.0 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.			None
		dimensionless	dimensionless	dimensionless	
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.			None
		psia	bars	atm	
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 8.69: SLGOF Keyword Description

Example

```
--
--      GAS-OIL RELATIVE PERMEABILITY TABLES (SLGOF)
SLOF
--      SLIQ      KRG      KROG      PCOG
--      FRAC
--      -----
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
--
--
--      -----
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.

8.3.85 SOF2 – OIL SATURATION TABLES WITH RESPECT TO GAS OR WATER (FORMAT TYPE 2)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

The SOF2 keyword defines the oil relative permeability versus oil saturation tables for when oil and gas or oil and water are present in the input deck. The keyword is also used to define the relative permeability of the miscible hydrocarbon phase in SOLVENT runs. This keyword should only be used if 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 stating from 0 and terminating at 1.0, 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 $SOIL = S_o + S_g + S_s$.			None
		dimensionless	dimensionless	dimensionless	
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 0.0 and less than or equal to 1.0 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.			None
		dimensionless	dimensionless	dimensionless	

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 8.70: 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
--
--
--      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

/ TABLE NO. 02

The example defines two SOF2 tables for when oil and gas or oil and water are present in the input deck.

8.3.86 SOF3 – OIL SATURATION TABLES WITH RESPECT TO GAS AND WATER (FORMAT TYPE 2)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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 stating from 0 and terminating at 1.0, that defines the oil or the hydrocarbon solvent saturation. The final entry should be at the connate water saturation, that is 1- Swc.			None
		dimensionless	dimensionless	dimensionless	
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 0.0 and less than or equal to 1.0 that defines the oil relative permeability with respect to oil and water saturation. The first value in the column should be zero.			None
		dimensionless	dimensionless	dimensionless	
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 0.0 and less than or equal to 1.0 that defines the oil relative permeability with respect to oil, gas and connate water saturation. The first value in the column should be zero.			None
		dimensionless	dimensionless	dimensionless	

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 8.71: SOF3 Keyword Description

Example

```
--
--      OIL RELATIVE PERMEABILITY TABLES (SOF3)
--
SOF3
--      SOIL      KRO      KROG
--      FRAC      FRAC      FRAC
--      -----
--          0.00      0.000000      0.000000
--          0.05      1.197e-5      0.000000
--          0.10      0.000191      0.000000
--          0.15      0.000969      0.000000
--          0.20      0.003065      0.000000
--          0.25      0.007483      0.000000
--          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
--
--
--          0.00      0.000000      0.000000
--          0.05      1.197e-5      0.000000
--          0.10      0.000191      0.000000
--          0.15      0.000969      0.000000
--          0.20      0.003065      0.000000
--          0.25      0.007483      0.000000
--          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

/ TABLE NO. 2

The example defines two SOF3 tables for when oil, gas and water are present in the input deck.

8.3.87 SOGCR – END-POINT SCALING GRID CELL CRITICAL OIL SATURATION WITH RESPECT TO GAS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

SOGCR defines the critical oil saturation for all the cells in the model via an array when the end-point scaling option has been invoked via the ENDSCALE keyword in the RUNSPEC section. The critical oil saturation with respect to gas is defined as the maximum oil saturation for which the oil relative permeability is zero in a two-phase gas-oil relative permeability table.

The keyword can be used for all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
I	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 non-direction dependent 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 8.72: 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 endpoint 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.

8.3.88 SORWMIS – MISCIBLE RESIDUAL OIL VERSUS WATER SATURATION FUNCTIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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 stating from 0 and terminating at 1.0, that defines the water saturation.			None
		dimensionless	dimensionless	dimensionless	
2	SORMIS	A columnar vector of real equal or increasing down the column values that are greater than or equal to zero and less then 1.0, that define the corresponding miscible residual oil saturation for the corresponding water saturation SWAT.			None
		dimensionless	dimensionless	dimensionless	
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 8.73: 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.

8.3.89 SOWCR – END-POINT SCALING GRID CELL CRITICAL OIL SATURATION WITH RESPECT TO WATER

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

SOWCR defines the critical oil saturation for all the cells in the model via an array when the end-point scaling option has been invoked via the ENDSCALE keyword in the RUNSPEC section. The critical oil saturation with respect to water is defined as the maximum oil saturation for which the oil relative permeability is zero in a two-phase oil-water relative permeability table.

The keyword can be used for all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
I	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 non-direction dependent 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 8.74: 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 SOWCRX, SOWCRY and SOWCRZ instead of SOWCR. There is also the facility to make the directional endpoint 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 gas saturation of 0.20 to all 300 cells in the model as defined by the DIMENS keyword in the RUNSPEC section.

8.3.90 SPECHEAT – DEFINE THE SPECIFIC HEAT OF OIL, WATER AND GAS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

SPECHEAT defines the specific heat of the oil, water and gas phases for various PVT regions in the model. 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 the temperature option has been activated by the TEMP keyword in the RUNSPEC section.

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.			None
		°F	°C	°C	
2	OILSHEAT	OILSHEAT is a columnar vector of positive real numbers defining the specific heat of oil at the corresponding temperature, TEMP.			None
		Btu/lb/°R	kJ/kg/K	J/gm/K	
3	WATSHEAT	WATSHEAT is a columnar vector of positive real numbers defining the specific heat of water at the corresponding temperature, TEMP.			None
		Btu/lb/°R	kJ/kg/K	J/gm/K	
4	GASSHEAT	GASHEAT is a columnar vector of positive real numbers defining the specific heat of gas at the corresponding temperature, TEMP.			None
		Btu/lb/°R	kJ/kg/K	J/gm/K	

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 8.75: SPECHEAT Keyword Description

See also the SPECROCK keyword to define the rock specif heat.

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 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.

8.3.91 SPECROCK – DEFINE THE SPECIFIC HEAT OF THE RESERVOIR ROCK

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

Description

SPECROCK defines the specific heat of the reservoir rock for various PVT regions in the model. 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 the temperature option has been activated by the TEMP keyword in the RUNSPEC section.

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.			None
		°F	°C	°C	
2	ROCKHEAT	ROCKHEAT is a columnar vector of positive real numbers defining the specific heat of the rock at the corresponding temperature, TEMP.			None
		Btu/lb/°R	kJ/kg/K	J/gm/K	

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 8.76: SPECROCK Keyword Description

See also the SPECHEAT keyword to define the specif heat relationships for the oil, water and gas phases.

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 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
--
SPECHEAT
--          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
```

8.3.92 SSFN – SOLVENT AND GAS RELATIVE PERMEABILITY TABLES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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). Why 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 stating from 0 and terminating at 1.0, that defines the gas plus solvent saturation ration which is defined as either: $\frac{S_g}{(S_g + S_s)} \text{ or } \frac{S_s}{(S_g + S_s)}$ Where Sg is the gas saturation and Ss is the solvent saturation.			None
		dimensionless	dimensionless	dimensionless	
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 0.0 and less than or equal to 1.0 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..			None
		dimensionless	dimensionless	dimensionless	
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 0.0 and less than or equal to 1.0 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..			None
		dimensionless	dimensionless	dimensionless	

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 8.77: SSFN Keyword Description

Example

```
--
--      SOLVENT RELATIVE PERMEABILITY TABLES
--
SSFN
--      SGAS      KRGD      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 the MISCIBLE and SOLVENT options.

8.3.93 SWCR – END-POINT SCALING GRID CELL CRITICAL WATER SATURATION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

SWCR defines the critical water saturation for all the cells in the model via an array when the end-point scaling option has been invoked via the ENDSALE 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 for 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 non-direction dependent version of the critical water saturation used with the end-point scaling option. If directional end-point scaling has been activated then the SWCRX± , SWCRY± and SWCRZ± series of keyword should be used.
- 2) If the value for a cell has been defaulted then OPM Flow users the value from the cell’s relative permeability table.
- 3) The keyword is terminated by “/”.

Table 8.78: 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 endpoint 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-, 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.

8.3.94 SWFN – WATER SATURATION TABLES (FORMAT TYPE 2)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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 stating from 0 and terminating at 1.0, that defines the water saturation.			None
		dimensionless	dimensionless	dimensionless	
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 0.0 and less than or equal to 1.0 that defines the water relative permeability with respect to gas saturation. The first value in the column should be zero.			None
		dimensionless	dimensionless	dimensionless	
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.			None
		psia	bars	atm	

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 8.79: 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*
--
--
--      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

/ 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.

8.3.95 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 for 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 non-direction dependent 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 8.80: 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 endpoint 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-, 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.

8.3.96 SWOF – WATER-OIL SATURATION TABLES (FORMAT TYPE 1)

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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 stating from 0 and terminating at 1.0, that defines the water saturation. The first entry is the connate water saturation Swc and the last entry should be 1.0.			None
		dimensionless	dimensionless	dimensionless	
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 0.0 and less than or equal to 1.0 that defines the water relative permeability with respect to gas saturation. The first value in the column should be zero.			None
		dimensionless	dimensionless	dimensionless	
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 0.0 and less than or equal to 1.0 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.			None
		dimensionless	dimensionless	dimensionless	
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.			None
		psia	bars	atm	

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 8.81: SWOF Keyword Description

Example

```
--
--      WATER-OIL RELATIVE PERMEABILITY TABLES (SWOF)
--
SWOF
--      SWAT      KRW      KROW      PCOW
--      FRAC
--      -----
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.

8.3.97 SWU – END-POINT SCALING GRID CELL GAS SATURATION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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 for 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 non-direction dependent 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± , SWUY± and SWUZ± 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 8.82: 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 endpoint 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-, 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.

8.3.98 TLMIXPAR – DEFINE THE MISCIBLE TODD-LONGSTAFF MIXING PARAMETERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

The TLMIXPAR keyword defines the Todd-Longstaff³⁷ 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.			None
		dimensionless	dimensionless	dimensionless	
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 8.83: 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.

³⁷ 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.

8.3.99 TRACER – DEFINE PASSIVE TRACER VARIABLES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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 TRACER 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 a 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 8.84: 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.

8.3.100 VISCREF - DEFINE VISCOSITY-TEMPERATURE REFERENCE CONDITIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

VISCREF defines the reference conditions for the viscosity-temperature tables, OILVISCT and WATVISCT, for when the temperature option has been activated by TEMP keyword in the RUNSPEC section.

OPM Flow currently does not support API tracking and therefore item (3) of this keyword is ignored.

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			None
		psia	barsa	atma	
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			None
		Mscf/stb	sm ³ /sm ³	scc/scc	
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.			None
		°API	°API	°API	

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: VISCREF Keyword Description

See also the OILVISCT and WATVISCT keywords in the PROPS section.

Example

The following example shows the VISCREF keyword for when the temperature 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.

8.3.101 WATVISCT – WATER VISCOSITY VERSUS TEMPERATURE FUNCTIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

WATVISCT defines the water viscosity as a function of temperature for when temperature option has been activated by the TEMP keyword in the RUNSPEC. The reference pressure for this table is given by the VISCREF keyword in the PROPS section.

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.			None
		°F	°C	°C	
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 PRS variable on the VISCREF keyword.			None
		cP	cP	cP	

Notes:

1)

The keyword is followed by NTPVT tables as declared on the TABDIMS keyword in the RUNSPEC section.

2)

Each table must contain a minimum of two rows and a maximum of NPPVT rows as declared on the TABDIMS keyword in the RUNSPEC section.

1)

Each table is terminated by “/” and there is no “/” terminator for the keyword.

Table 8.86: WATVISCT Keyword Description

Example

The following example shows the WATVISCT keyword for when the temperature 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.

```
--
--      WATER VISCOSITY VERSUS TEMPERATURE TABLES
--
--      WATER      WATER
--      TEMP      VISC
--      -----
WATVISCT
      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.

9 REGIONS SECTION

9.1 INTRODUCTION

To be written in a future release of the manual.

9.2 DATA REQUIREMENTS

To be written in a future release of the manual.

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 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.

9.3.4 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.

9.3.5 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.

9.3.6 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.

9.3.7 ENDNUM – DEFINE THE END-POINT SCALING DEPTH REGION NUMBERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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 ENPVTD³⁸ and the ENKRVD³⁹ keywords in the PROPS section. In the RUNSPEC section the NTENDP variable on the ENDSALE keyword defines the maximum number of depth tables.

No.	Name	Description	Default
I	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 ENDSALE keyword in the RUNSPEC section.	I
Notes: <ol style="list-style-type: none"> 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 cell is not assigned a ENDNUM region number then the default value of one will be used. 3) The keyword is terminated by “/”. 			

Table 9.1: 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
/
```

³⁸ This keyword is ignored by OPM Flow and has no effect on the simulation

³⁹ This keyword is ignored by OPM Flow and has no effect on the simulation

9.3.8 EQLNUM – DEFINE THE EQUILIBRATION REGION NUMBERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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
I	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.	I
Notes: <ol style="list-style-type: none"> 1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement. 2) The EQLNUM and PVTNUM arrays need to be consistent, that is the all cells with the same EQLNUM can only belong to one PVTNUM region. 3) If cell is not assigned a EQLNUM region number then the default value will be used. 4) The keyword is terminated by “/”. 			

Table 9.2: 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
/

```

9.3.9 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.

9.3.10 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.

9.3.11 FIPNUM – DEFINE THE FLUID IN-PLACE REGION NUMBERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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
I	FIPNUM	FIPNUM defines an array of positive integers assigning 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.	I
Notes: <ol style="list-style-type: none"> 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 cell is not assigned a FIPNUM region then the default value will be used. 3) The keyword is terminated by “/”. 			

Table 9.3: 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.

9.3.12 FIPOWG – ACTIVATE OIL, GAS, AND WATER FIP ZONE REPORTING

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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.

9.3.13 IMBNUM – DEFINE THE IMBIBITION SATURATION TABLE REGION NUMBERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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
I	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.	I
Notes: <ol style="list-style-type: none"> 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 cell is not assigned a IMBNUM region then the default value of one will be used. 3) The keyword is terminated by “/”. 			

Table 9.4: 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 endpoint 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-, IMBNUMY-, IMBNUMY-, IMBNUMZ and IMBNUMZ-, instead of the IMBNUM keyword.

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
/
```

9.3.14 MISNUM – DEFINE THE MISCIBILITY REGION NUMBERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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
I	MISNUM	<p>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.</p> <p>A value of zero sets the fluids within a grid cell to be immiscible.</p> <p>The maximum number of MISNUM regions is set by the NTMIS variable on the MISCIBLE keyword in the RUNSPEC section.</p>	I
Notes: <ol style="list-style-type: none"> 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 cell is not assigned a MISNUM region then the default value of one will be used. 3) The keyword is terminated by “/”. 			

Table 9.5: 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
/
```


9.3.15 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.

9.3.16 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.

9.3.17 OPERATE – DEFINE MATHEMATICAL OPERATIONS ON ARRAYS

This keyword defines the 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.

See [OPERATE – Define Mathematical Operations on Arrays](#) in the GRID section for a full description.

9.3.18 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.

9.3.19 PLMIXNUM – DEFINE THE POLYMER REGION NUMBERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

The PLMIXNUM keyword defines the polymer region number 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
I	PLMIXNUM	PLMIXNUM defines an array of positive integers greater than 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.	I
Notes: <ol style="list-style-type: none"> 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 cell is not assigned a PLMIXNUM region then the default value of one will be used. 3) The keyword is terminated by “/”. 			

Table 9.6: 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
/
```

9.3.20 PVTNUM – DEFINE THE PVT REGIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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 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: <ol style="list-style-type: none"> 1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword, in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement. 2) The EQLNUM and PVTNUM arrays need to be consistent, that is the all cells with the same PVTNUM can only belong to one EQLNUM region. 3) If cell is not assigned a PVTNUM region then the default value will be used. 4) The keyword is terminated by “/”. 			

Table 9.7: 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.

9.3.21 REGIONS - DEFINE THE START OF THE REGIONS SECTION OF KEYWORDS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	----------------	--------------------------	-------------------------	--------------------------

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.

9.3.22 ROCKNUM – DEFINE ROCK COMPACTION TABLE REGION NUMBERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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
I	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.	I
Notes: <ol style="list-style-type: none"> 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 cell is not assigned a ROCKNUM region then the default value will be used. 3) The keyword is terminated by “/”. 			

Table 9.8: 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
/
```

9.3.23 RPTREGS – DEFINE REGIONS SECTION REPORTING

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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 format in the commercial simulator and was subsequently superseded by the second format. The second format consists of the keyword followed by a series of character strings that indicate the data to be printed. In most cases the character string is the keyword used to load the data in the OPM Flow input deck, for example FIPNUM for the fluid in-place array. It 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 9.9: 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 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          /
```

A	B	C	D	E	F	G	H	I	J	K	L	M	N	O	P	Q	R	S	T	U	V	W	X	Y	Z
---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---

9.3.24 SATNUM – DEFINE THE SATURATION TABLE REGION NUMBERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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
I	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.	I
Notes: <ol style="list-style-type: none"> 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 cell is not assigned a SATNUM region then the default value will be used. 3) The keyword is terminated by “/”. 			

Table 9.10: 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
/
```


9.3.25 TNUM – DEFINE PASSIVE TRACER CONCENTRATION REGIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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. The following table outlines the format of the TNUM keyword name.

No.	Name	Description	Default
1	TNUM	A four letter character equal 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 9.11: 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	TNUMDATA	TUNDATA defines an array of positive integers assigning a grid cell to a particular tracer table region. The maximum number of TNUMDATA regions is set by the NTTRVD variable on the EQLDIMS keyword in the RUNSPEC section.	1
Notes: <ol style="list-style-type: none"> 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 cell is not assigned a TNUMDATA region then the default value will be used. 3) The keyword is terminated by “/”. 			

Table 9.12: TNUM Keyword Data Description

See also the TRACER keyword in the PROPS section and the TBLK 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.

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
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.

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 AQUFETP – FETKOVICH AQUIFER DEFINITION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

Description

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

10.3.4 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.

10.3.5 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.

10.3.6 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.

10.3.7 DATUM – DEFINE THE DATUM DEPTH FOR THE MODEL

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	---------------------------------	-------------------------	--------------------------

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	
I	DATUM	DATUM is a single positive value that defines the datum depth for the model.			None
		feet	m	cm	
Notes: I) The keyword is terminated by “/”.					

Table 10.1: DATUM Keyword Description

See also the DATUMR keyword that defines the datum for each fluid in-place region.

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

10.3.8 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	
I	DATUMR	DATUMR is a vector of positive values that defines the datum depth for each fluid in-place region.			None
		feet	m	cm	
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 10.2: DATUMR Keyword Description

See also the DATUM keyword that defines the datum depth for the model.

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

Examples

```
-- DATUM
-- DEPTH
-- -----
DATUM
    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.

10.3.9 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.

10.3.10 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.

10.3.11 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.

10.3.12 EQUIL – DEFINE THE EQUILIBRATION INITIALIZATION DATA

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	---------------------------------	-------------------------	--------------------------

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 for 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.			0.0
		feet	m	cm	
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.			0.0
		psia	barsa	atma	
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).			0.0
		feet	m	cm	
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			0.0
		psia	barsa	atma	
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.			0.0
		psia	barsa	atma	
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.			0
		dimensionless	dimensionless	dimensionless	
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.			0
		dimensionless	dimensionless	dimensionless	

No.	Name	Description			Default
		Field	Metric	Laboratory	
9	EQLOPT3	<p>EQLOPT3 is an integer value that sets the initialization accuracy options for the equilibration calculation.</p> <p>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.</p> <p>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.</p> <p>Increasing the value of N increases the accuracy of the calculation, with the maximum value of N being set to 20 by OPM Flow.</p> <p>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.</p> <p>Note this option should be used with Irregular Corner-Point Grids.</p> <p>EQLOPT3 is ignored for Radial Grids.</p> <p>Only EQLOPT3 equal to zero is supported by OPM Flow.</p>			0
		dimensionless	dimensionless	dimensionless	
10		Not used.			
11		Not used			

Notes:

1) The keyword is followed by NTEQL 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 10.3: 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 NTEQL 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.

10.3.13 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.

10.3.14 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.

10.3.15 OPERATE – DEFINE MATHEMATICAL OPERATIONS ON ARRAYS

This keyword defines the 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.

See [OPERATE – Define Mathematical Operations on Arrays](#) in the GRID section for a full description.

10.3.16 PBUB – DEFINE THE INITIAL EQUILIBRATION BUBBLE-POINT PRESSURE FOR ALL GRID BLOCKS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

The PBUB keyword defines the initial equilibration bubble-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 for 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	
I	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			None
		psia	barsa	atma	

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 10.4: PBUB 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 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.

10.3.17 PBVD – EQUILIBRATION BUBBLE-POINT VERSUS DEPTH TABLES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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 EQLOPTI 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.			None
		feet	m	cm	
2	PBVALS	A columnar vector of real values that defines the oil bubble-point values at the corresponding DEPTH.			None
		psia	barsa	atma	

Notes:

- 1) The keyword is followed by NTEQL 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 10.5: PBVD Keyword Description

This keyword is ignored by OPM Flow and has no effect on the simulation but is documented here for completeness. 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 NTEQL 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
--
--      3000.0      3100.0
      8000.0      3125.0
--
--      3000.0      3200.0
      8000.0      3225.0
--
```

/ PSAT VS DEPTH EQUIL REGN 01
/ PSAT VS DEPTH EQUIL REGN 02
/ PSAT VS DEPTH EQUIL REGN 03

Here three tables are entered and each table is terminated by "/" and there is no keyword terminating "/".

10.3.18 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 for 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	
I	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			None
		psia	barsa	atma	

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 10.6: 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.

10.3.19 PDVD – EQUILIBRATION DEW-POINT VERSUS DEPTH TABLES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	-----------------	-------------------------	--------------------------

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.

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 dew-point values, PDVALS.			None
		feet	m	cm	
2	PDVALS	A columnar vector of real values that defines the gas dew-point values at the corresponding DEPTH.			None
		psia	barsa	atma	

Notes:

- 1) The keyword is followed by NTEQL 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 10.7: 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 NTEQL 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 “/”.

10.3.20 PRESSURE – DEFINE THE INITIAL EQUILIBRATION PRESSURES FOR ALL GRID BLOCKS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	---------------------------------	-------------------------	--------------------------

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. 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 for all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
I	PRESS	PRESS 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.			None
		psia	barsa	atma	

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 10.8: 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.

10.3.21 RESTART – RESTART RUN FROM AN EXISTING RESTART FILE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	---------------------------------	-------------------------	--------------------------

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 10.9: RESTART Keyword Description

The most direct way to start a restart run is to:

- 1) Copy the existing data file that created the RESTART file and give it a new name. For example if the RESTART file is from a case named *NOR-OPM-A01*.DATA, then the copied data file could be named *NOR-OPM-A01-RI*.DATA.
- 2) Edit the copied data file (*NOR-OPM-A01-RI*.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-RI*.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-RI*.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-RI*.DATA file after the RESTART point make any required changes, save the file and run the *NOR-OPM-A01-RI*.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 is advisable to place the SKIPREST keyword at the very beginning of the SCHEDULE section.

10.3.22 RPTRST – DEFINE DATA TO BE WRITTEN TO THE RESTART FILE

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. 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. It 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	<p>BASIC defines the frequency at which the restart data for restarting a run and the additional requested data is written to the RESTART file. The parameter is assigned a value, OPTION, using the form BASIC = OPTION, where OPTION is an integer variable set to:</p> <ol style="list-style-type: none"> 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. OPTION = 2 then the restart files are written at every report time step until this switch is reset and all the restarts are kept. 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. OPTION = 4 then the restart files are written at the first report step of each year. OPTION = 5 then the restart files are written at the first report step of each month. OPTION = 6 then the restart files are written at every time step. <p>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.</p>	
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
7	RSSAT	Saturated dissolved gas-oil ratio for each grid block to enable restarts.	RSSAT

A	B	C	D	E	F	G	H	I	J	K	L	M	N	O	P	Q	R	S	T	U	V	W	X	Y	Z
---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---

No.	Name	Description	Array Name
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 10.10: 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
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

No.	Restart Variable Name	Variable Description	Variable Array Name
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: 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 10.11: 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=4 /
```

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 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=4  KRG  KRO  KRW /
```


10.3.23 RPTSOL – DEFINE SOLUTION SECTION REPORTING

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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 format in the commercial simulator and was subsequently superseded by the second format. The second format consists of the keyword followed by a series of character strings that indicate the data to be printed. In most cases the character string is the keyword used to load the data in the OPM Flow input deck, for example PVDG for the dry gas PVT tables. It 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 10.12: 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 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          /
```

10.3.24 RS – DEFINE THE INITIAL EQUILIBRATION GOR (Rs) FOR ALL GRID BLOCKS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	---------------------------------	-------------------------	--------------------------

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 for all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
I	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*I.30.			None
		Mscf/stb	sm ³ /sm ³	scc/scc	

Notes:

- 1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
- 2) The keyword is terminated by “/”.

Table 10.13: 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.

10.3.25 RSVD – EQUILIBRATION DISSOLVED GAS-OIL RATIO (Rs) VERSUS DEPTH TABLES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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 EQLOPTI 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.			None
		feet	m	cm	
2	RSVALS	A columnar vector of real values that defines the dissolved gas-oil ratio values at the corresponding DEPTH.			None
		Mscf/stb	sm ³ /sm ³	scc/scc	

Notes:

- 1) The keyword is followed by NTEQL 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: 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 NTEQL 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.

10.3.26 RTEMP - DEFINE THE INITIAL RESERVOIR TEMPERATURE FOR THE MODEL

This keyword defines the reservoir temperature for when temperature option has been activated by the TEMP keyword 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.

10.3.27 RTEMPA - DEFINE THE INITIAL RESERVOIR TEMPERATURE FOR THE MODEL

This keyword defines the reservoir temperature for when temperature option has been activated by the TEMP keyword 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.

10.3.28 RTEMPVD - DEFINE THE INITIAL RESERVOIR TEMPERATURE VERSUS DEPTH TABLES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

This keyword defines the reservoir temperature versus depth tables for when the temperature option has been activated by the TEMP keyword in the RUNSPEC section. The RTEMPVD keyword is an alias for the TEMPVD keyword.

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 TEMP.			None
		feet	m	cm	
2	TEMP	A real monotonically increasing down the column values that defines the corresponding reservoir temperature for the given depth			None
		°F	°C	°C	

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 EQ:DIMS keyword in the RUNSPEC section.

1)

Each table is terminated by “/” and there is no “/” terminator for the keyword.

Table 10.15: RTEMPVD Keyword Description

See also the RTEMP 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

```
--
--      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
--
--
--      1000.0      90.000
--      2000.0      100.000
--      3000.0      130.000
--      4000.0      160.000
--
--
--      1000.0      90.000
--      2000.0      100.000
--      3000.0      130.000
--      4000.0      160.000
```

/ TABLE N0. 01

/ TABLE N0. 02

/ 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.

10.3.29 RV – DEFINE THE INITIAL EQUILIBRATION CGR (Rv) FOR ALL GRID BLOCKS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	---------------------------------	-------------------------	--------------------------

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, PDEV, 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 for all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
I	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			None
		stb/Mscf	sm ³ /sm ³	scc/scc	

Notes:

- 1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
- 2) The keyword is terminated by “/”.

Table 10.16: RV Keyword Description

See also the PBUB, PDEV, 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.

10.3.30 RVVD – EQUILIBRATION VAPORIZED OIL-GAS RATIO (Rv) VERSUS DEPTH TABLES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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.			None
		feet	m	cm	
2	RVVALS	A columnar vector of real values that defines the vaporized oil-gas ratio values, values at the corresponding DEPTH.			None
		stb/Mscf	sm ³ /sm ³	scc/scc	

Notes:

- 1) The keyword is followed by NTEQL 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 10.17: 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 NTEQL 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
--      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.

10.3.31 SGAS – DEFINE THE INITIAL EQUILIBRATION GAS SATURATION FOR ALL GRID BLOCKS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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, PDEV, 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 for all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
I	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.			None
		dimensionless	dimensionless	dimensionless	

Notes:

- 1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
- 2) The keyword is terminated by “/”.

Table 10.18: 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, PDEV, 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.

10.3.32 SOIL – DEFINE THE INITIAL EQUILIBRATION OIL SATURATION FOR ALL GRID BLOCKS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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 for all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
I	SOIL	SOIL is an array of real positive numbers that are greater than or equal to zero and less than or equal to one assigning the initial equilibration oil saturation values to each cell in the model. Repeat counts may be used, for example 20*0.600.			None
		dimensionless	dimensionless	dimensionless	

Notes:

- 1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
- 2) The keyword is terminated by “/”.

Table 10.19: 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 GAS 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.

10.3.33 SOLUTION - DEFINE THE START OF THE SOLUTION SECTION OF KEYWORDS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	-----------------	-------------------------	--------------------------

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 initialize the model, by:

- 1) defining fluid contacts and pressures, or
- 2) defining pressures and fluid saturations for all cells in the model, or
- 3) by restarting from a previously 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.

10.3.34 SSOL – DEFINE THE INITIAL EQUILIBRATION SOLVENT SATURATION FOR ALL GRID BLOCKS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	-----------------	-------------------------	--------------------------

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 for all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
I	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.			None
		dimensionless	dimensionless	dimensionless	

Notes:

- 1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
- 2) The keyword is terminated by “/”.

Table 10.20: 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 model.

10.3.35 STONE1 – ACTIVATE STONE’S FIRST THREE PHASE OIL RELATIVE PERMEABILITY MODEL

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

This keyword activates Stone’s⁴⁰ first three phase oil relative permeability model as modified by Aziz and Settari⁴¹. 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.

⁴⁰ 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.

⁴¹ Aziz, K. and Settari, A. *Petroleum Reservoir Simulation*, London, UK, Applied Science Publishers (1979), page 398.

10.3.36 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’s⁴² first three phase oil relative permeability model as modified by Aziz and Settari⁴³. The STONE1EX keyword should only be used in three phase runs containing the oil, gas and water phases and when the STONEI 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	
I	STONEPARI	A real positive value that defines the exponent to be used in the Modified Stone first three phase oil relative permeability model.			None
		dimensionless	dimensionless	dimensionless	
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 10.21: STONE1EX Keyword Description

If the STONEI and STONE2 keywords are not present in the input deck then the default three phase oil relative permeability model is employed.

Example

Given NTSFUN equals five on the TABDIMS keyword in the RUNSPEC section, then:

```
--
--      DEFINE STONE'S FIRST THREE PHASE RELATIVE PERMEABILITY MODEL PARAMETER
--
STONE1EX
    1.000          / SATURATION TABLE NO. 01
    1.000          / SATURATION TABLE NO. 02
    2.000          / SATURATION TABLE NO. 03
    1.000          / SATURATION TABLE NO. 04
    3.000          / SATURATION TABLE NO. 05
```

Defines the exponents to be used in the Modified Stone first three phase oil relative permeability model, for each of the five saturation tables.

⁴² 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.

⁴³ Aziz, K. and Settari, A. Petroleum Reservoir Simulation, London, UK, Applied Science Publishers (1979), page 398.

10.3.37 STONE2 – ACTIVATE STONE’S SECOND THREE PHASE OIL RELATIVE PERMEABILITY MODEL

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

This keyword activates Stone’s⁴⁴ second three phase oil relative permeability model as modified by Aziz and Settari⁴⁵. 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 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.

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

Example

```
--
--      ACTIVATE STONE’S SECOND THREE PHASE RELATIVE PERMEABILITY MODEL
--
STONE2
```

The above example switches on the Modified Stone three phase relative permeability model

⁴⁴ Stone, H. L. “Estimation of Three-Phase Relative Permeability and Residual Oil Data,” *Journal of Canadian Petroleum Technology* (1973) 12, No. 4, 53-61.

⁴⁵ Aziz, K. and Settari, A. *Petroleum Reservoir Simulation*, London, UK, Applied Science Publishers (1979), page 398.

10.3.38 SWAT – DEFINE THE INITIAL EQUILIBRATION WATER SATURATION FOR ALL GRID BLOCKS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	---------------------------------	-------------------------	--------------------------

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 water 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 for all grid types.

No.	Name	Description			Default
		Field	Metric	Laboratory	
I	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.			None
		dimensionless	dimensionless	dimensionless	

Notes:

- 1) The number of entries should correspond to the NX x NY x NZ parameters on the DIMENS keyword in the RUNSPEC section, unless the BOX keyword defines a sub area of the grid, in which case the total number of entries should correspond to the number of cells defined by the BOX statement.
- 2) The keyword is terminated by “/”.

Table 10.22: 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.

10.3.39 TEMPVD - DEFINE THE INITIAL RESERVOIR TEMPERATURE VERSUS DEPTH TABLES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	-----------------	-------------------------	--------------------------

Description

This keyword defines the reservoir temperature versus depth tables for when the temperature option has been activated by the TEMP keyword in the RUNSPEC section. The TEMPVD keyword is an alias for the RTEMPVD keyword.

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.			None
		feet	m	cm	
2	RTEMP	A real monotonically increasing down the column values that defines the corresponding reservoir temperature for the given depth			None
		°F	°C	°C	

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 EQ:DIMS keyword in the RUNSPEC section.
- 3) Each table is terminated by “/” and there is no “/” terminator for the keyword.

Table 10.23: RTEMPVD Keyword Description

See also the RTEMP 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.

10.3.40 THPRES - DEFINE EQUILIBRATION REGION THRESHOLD PRESSURES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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.			None
		dimensionless	dimensionless	dimensionless	
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.			None
		dimensionless	dimensionless	dimensionless	
3	THPRES	THPRES defines the threshold pressure from EQLNUM1 to EQLNUM2 and from EQLNUM2 to EQLNUM1. The default value of I* 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.			I*
		psia	barsa	atma	

Notes:
1) The keyword is followed by any number of records with each record terminated by “/”.
2) The keyword is terminated by “/”.

Table 10.24: 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.

10.3.41 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.

10.3.42 VAPPARS – OIL VAPORIZATION PARAMETERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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.			0
		dimensionless	dimensionless	dimensionless	
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.			0
		dimensionless	dimensionless	dimensionless	

Notes:

2) The keyword is terminated by “/”.

Table 10.25: 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 R_s and the vaporized oil R_s 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.

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

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

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

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.

11.3.2 DATE - ACTIVATE THE DATE OPTION FOR THE SUMMARY FILE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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.

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 DAYS	YEARS YEARS	FPR PSIA	FOEW	FOPR STB/DAY	FOPT 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 YEARS	DAY	MONTH	YEAR	FPR PSIA	FOEW	FOPR 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

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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.

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 GWSET - EXPORT WELL STATUS VECTORS BY GROUP 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 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 – ACTIVATES 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 – Activates Output of the Monitoring Data and File](#) in the RUNSPEC section for a full description.

11.3.7 NOMONITO – DEACTIVATES 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 – Deactivates Output of the Monitoring Data and File](#) in the RUNSPEC section for a full description.

11.3.8 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.

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.

11.3.9 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.

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.

11.3.10 RUNSUM – ACTIVATE RSM FILE OUTPUT OF THE SUMMARY DATA

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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.

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.

11.3.11 SEPARATE – ACTIVATE THE SEPARATE RSM FILE OUTPUT OPTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

Description

This keyword activates the writing out of the SUMMARY file data in a columnar format to the RSM file, if the RUNSUM keyword has 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.

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.

11.3.12 SUMMARY - DEFINE THE START OF THE SUMMARY SECTION OF KEYWORDS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	----------------	--------------------------

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.

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.

11.3.13 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 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 enables the size of the SUMMARY files to be reduced depending on the size of the time interval. However, the keyword will produce irregular time steps reports of the SUMMARY data.

No.	Name	Description			Default
		Field	Metric	Laboratory	
I	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 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.			None
		days	days	hours	
Notes: I) The keyword is terminated by “/”.					

Table 11.1: 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.

12.3 KEYWORD DEFINITIONS

12.3.1 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.

12.3.2 CECON – DEFINE WELL CONNECTIONS ECONOMIC LIMIT CRITERIA

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

Description

CECON set 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.

12.3.3 COMPDAT – DEFINE WELL CONNECTIONS TO THE GRID

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

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 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 I* 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 I* 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	<p>An integer greater than or equal to zero and less than NTSFUN as declared on the TABDIMS keyword in the RUNSPEC, that defines the saturation table number to be used for flow between the reservoir grid block and the well connections.</p> <p>If SATNUM is set to zero or defaulted with I* then:</p> <ul style="list-style-type: none"> 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	<p>A real value greater than or equal to zero that defines the transmissibility connection factor between the well bore and the reservoir grid block.</p> <p>If set to zero or defaulted with I* then items (9) through (13) are used to calculate CONFACT.</p>			Defined
		cP.rb/day/psia 0	cP.rm ³ /day/bars 0	cP.rcc/hr/atm 0	
9	RW	<p>A real positive value that defines the well bore <u>diameter</u> of the connections for the well.</p> <p>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.</p>			None
		feet	m	cm	
10	KH	<p>A real value that defines the effective KH (permeability x length) for the connections.</p> <p>If less than or equal to zero or defaulted by I* then KH is calculated from the connected grid blocks. KH is ignored if CONFAC has been directly entered.</p>			Calculated from connected grid blocks
		mD.ft	mD.m	mD.cm	
11	SKIN	<p>A real value that defines the connections dimensionless skin factor.</p> <p>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.</p>			0.0
		dimensionless	dimensionless	dimensionless	
12	DFACT	<p>A real value that defines the non-Darcy D factor coefficient for gas wells.</p> <p>This value should be defaulted with I* 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.</p>			I*
		day/Mscf	day/m ³	hour/sc	

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 12.1: 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).

12.3.4 COMPIMB – ASSIGN IMBIBITION SATURATION TABLES TO WELL CONNECTIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

The COMPIMB keyword assigns imbibition saturation tables to well connections. The COMPDAT keyword in the SCHEDULE section also assigns imbibition saturation tables to connections, but in this case the table number is the same as for the drainage curve. If this is not the required assignment then the COMPIMB keyword can be used to reset the imbibition saturation table number. For this to be effective the COMPIMB keyword must precede the COMPDAT keyword, otherwise it will have no effect.

The COMPIMB keyword should only be used if the hysteresis option has been activated via the HYSTER variable on the SATOPTS keyword in the RUNSPEC section.

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 I* 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 I* 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 I* 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 I* 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 grid block and the well connections. If IMBNUM is set to zero or defaulted with I* 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 I*, 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 12.2: 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
COMPIMP
OP01      1*  1*  20  56   11
OP01      1*  1*  75 100   12
OP02      1*  1*  1*  1*   11
/
```

Well OP01 has two sets of COMPIMP 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.

12.3.5 COMPLUMP – ASSIGN WELL CONNECTIONS TO COMPLETIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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 I* 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 I* 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 I* 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 I* 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 I*, 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 I2.3: 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 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 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.

12.3.6 COMPORD - DEFINE WELL CONNECTION ORDERING

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

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 12.4: 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.

12.3.7 COMPSEGS – DEFINE WELL CONNECTIONS FOR MULTI-SEGMENT WELLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

Description

The COMSEGS keyword defines how a multi-segment well is connected to the reservoir by defining or modifying existing well connections.

OPM Flow has not implemented multi-segment wells and therefore this keyword is ignored by OPM Flow and has no effect on the simulation (multi-segment wells are available in prototypes and will eventually be integrated with OPM Flow).

12.3.8 DATES – ADVANCE SIMULATION BY REPORTING DATE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

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: <ol style="list-style-type: none"> 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 12.5: DATES Keyword Description

See also the TSTEP keyword 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  JUL  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  JUL  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  JULY   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.

12.3.9 DRSDT – SOLUTION GAS (Rs) MAXIMUM RATE OF INCREASE PARAMETERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

DRSDT defines the maximum rate at which the solution gas-oil ratio (Rs) can be increased in a grid cell. This keyword 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	<p>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.</p> <p>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.</p> <p>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.</p>			None
		Mscf/stb/day	sm ³ /sm ³ /day	scc/scc/day	
2	DRSDT2	<p>DRSDT2 is a character string that defines whether the DRSDT1 is applied to either all grid blocks or just those grid blocks containing free gas:</p> <p>1) ALL: means the DRSDT1 maximum rate at which Rs is allowed to increase in a grid cell is applied to all grid blocks.</p> <p>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.</p> <p>Note if the keyword is not present in the input deck then DRSDT2 is set to the default value of ALL.</p>			ALL

Notes:

1) The keyword is terminated by “/”.

Table 12.6: 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 DRVDT keywords in the SCHEDULE section that controls how vaporized oil is treated and the rate at which vaporized oil-gas ratio increases within a grid block, respectively.

Examples

The first example prevents the solution gas-oil ratio from increasing and applies this to all grid cells.

```
--
--      MAXIMUM RATE OF INCREASE OF SOLUTION GOR IN GRID BLOCKS
--
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.

```
--
--      MAXIMUM RATE OF INCREASE OF SOLUTION GOR IN GRID BLOCKS
--
DRSDT
--      MAX RS      ALL/FREE
--      DRSDT1      DRSDT2
--      -----
--      0.0003      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.

12.3.10 DRVDT – SOLUTION OIL (Rv) MAXIMUM RATE OF INCREASE PARAMETERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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. This keyword should only be used if the OIL, GAS, and VAPOIL 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 a 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	
I	DRVDTI	<p>DRVDTI 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.</p> <p>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 DRVDTI allows Rv to increase rapidly until there is no free oil or the gas within the grid block is fully saturated.</p> <p>Note if the keyword is not present in the input deck then DRVDTI is assumed to be a very large number resulting in complete re-solution of the oil into the available undersaturated gas.</p>			None
		stb/Mscf/day	sm ³ /sm ³ /day	scc/scc/day	
Notes: I) The keyword is terminated by “/”.					

Table 12.7: 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 DRSDT keywords in the SCHEDULE section that controls how vaporized oil is treated and the rate at which dissolved gas-oil ratio increases within a grid block, respectively.

Examples

The example prevents the solution oil-gas ratio from increasing.

```
--
--      SOLUTION OIL (RV) MAXIMUM RATE OF INCREASE PARAMETERS
--
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.

12.3.11 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.

12.3.12 GCONINJE – GROUP INJECTION TARGETS AND CONSTRAINTS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

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
4	RATE	A real positive value that defines the maximum surface injection rate target or constraint for the phase declared by the TYPE variable/			None
		Liquid stb/d Gas Mscf/d	Liquid sm ³ /day Gas sm ³ /day	Liquid scc/hour Gas scc/hour	

A	B	C	D	E	F	G	H	I	J	K	L	M	N	O	P	Q	R	S	T	U	V	W	X	Y	Z
-------------------	-------------------	-------------------	-------------------	-------------------	-------------------	-------------------	-------------------	-------------------	-------------------	-------------------	-------------------	-------------------	-------------------	-------------------	-------------------	-------------------	-------------------	-------------------	-------------------	-------------------	-------------------	-------------------	-------------------	-------------------	-------------------

No.	Name	Description			Default
		Field	Metric	Laboratory	
5	RESV	A real positive value that defines the maximum reservoir volume injection rate target or constraint.			None
		rtb/d	rm ³ /day	rcc/hour	
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.			None
		dimensionless	dimensionless	dimensionless	
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.			None
		dimensionless	dimensionless	dimensionless	
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 I*.			None
		dimensionless	dimensionless	dimensionless	
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 I*.			None
11		Not used should be defaulted with I*.			
12		Not used should be defaulted with I*.			
13		Not used should be defaulted with I*.			

No.	Name	Description			Default
		Field	Metric	Laboratory	
Notes: 1) The keyword is followed by any numbers records with each record terminated by a “/” and the keyword should be terminated by a “/”.					

Table 12.8: GCONINJE Keyword Description

See also the GRUPTREE keyword to define the hierarchy of the groups below the FIELD level, the GCONPROD keyword to define a group's production targets and constraints, the WCONPROD keyword to define a production well's targets and constraints, and the WCONINJE keyword to define an injection well's targets and constraints. All the aforementioned keywords are described in the SCHEDULE section.

Example

The following example defines the injection targets and constraints for the field and two groups that are one level below the field group, since the GRUPTREE keyword has not been entered to define the group hierarchy.

```
--
--      GROUP INJECTION TARGETS AND CONSTRAINTS
--
-- GRUP  FLUID  CNTL   SURF   RESV   REINJ   VOID   GRUP  GUIDE  GUIDE  GRUP  GRUP
-- NAME  TYPE   MODE   RATE   RATE   FRAC    FRAC   CNTL  RATE  DEF   REINJ  RESV
GCONINJE
FIELD   WAT    VREP   35E3   1*     1*     1*     NO    1*    1*    1*    1*  /
GRP01   WAT    VREP   1*     1*     1*     1.0   YES   1*    1*    1*    1*  /
GRP02   WAT    VREP   1*     1*     1*     1.0   YES   1*    1*    1*    1*  /
/
```

In this example, group GRP01 and GRP02 are injecting water via voidage replacement with a voidage replacement of one and are under the control on the field group, that imposes a 35,000 m³/day total water injection limit.

12.3.13 GCONPROD – GROUP PRODUCTION TARGETS AND CONSTRAINTS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

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.			None
		stb/d	sm ³ /day	scc/hour	
4	WRAT	A real positive value that defines the maximum surface water production rate target or constraint.			None
		stb/d	sm ³ /day	scc/hour	
5	GAS	A real positive value that defines the maximum surface gas production rate target or constraint			None
		Mscf/d	sm ³ /day	scc/hour	

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	
7	ACTION	<p>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:</p> <ol style="list-style-type: none"> 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. <p>The corrective action takes places at the end of the time step in which the constraint is violated.</p>			NONE
8	GRPCNTL	<p>A defined character string that determines if this group is subject to higher level group control.</p> <ol style="list-style-type: none"> 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. <p>This feature is currently not supported by OPM Flow.</p>			None
9	GRPGUIDE	<p>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</p> <p>This feature is not supported by OPM Flow and should be defaulted with 1*.</p>			None
		dimensionless	dimensionless	dimensionless	

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 I*.			None
11		Not used should be defaulted with I*.			
12		Not used should be defaulted with I*.			
13		Not used should be defaulted with I*.			
14	RESV	A real positive value that defines the maximum reservoir volume production rate target or constraint.			
		rtb/d	rm ³ /day	rcc/hour	
15		Not used should be defaulted with I*.			
16		Not used should be defaulted with I*.			
17		Not used should be defaulted with I*.			
18		Not used should be defaulted with I*.			
19		Not used should be defaulted with I*.			
20		Not used should be defaulted with I*.			
21		Not used should be defaulted with I*.			
Notes: 1) The keyword is followed by any numbers records with each record terminated by a “/” and the keyword should be terminated by a “/”.					

Table 12.9: 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.

12.3.14 GECON – GROUP ECONOMIC CRITERIA FOR PRODUCTION GROUPS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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.

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.			0.0
		stb/d	sm ³ /day	scc/hour	
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,			0.0
		Mscf/d	sm ³ /day	scc/hour	
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.			0.0
		dimensionless	dimensionless	dimensionless	
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.			0.0
		Mscf/stb	sm ³ /sm ³	scc/scc	

No.	Name	Description			Default
		Field	Metric	Laboratory	
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.			0.0
		stb/Mscf	sm ³ /sm ³	scc/scc	
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. 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.t			NO
9		Not used			
Notes: 1) The keyword is followed by any numbers records with each record terminated by a “/” and the keyword should be terminated by a “/”.					

Table 12.10: 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.

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 economic criteria for the field with a minimum oil rate of 2,000 m³/day and a maximum water cut of 95%.

```
--  
--          GROUP ECONOMIC CRITERIA FOR PRODUCTION GROUPS  
--  
--  GRUP  OIL   GAS   WCT   GOR   WGR   WORK  END   MAX  
--  NAME  MIN   MIN   MAX   MAX   MAX   OVER  RUN   WELLS  
GECON  
FIELD    2E3   1*    0.95   1*    1*    CON   'YES'  1*      /  
/
```

If the economic limits are violated then the run will stop at the next report time step.

12.3.15 GEFAC – DEFINE GROUP EFFICIENCY

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

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).			1.0
		dimensionless	dimensionless	dimensionless	
3	NETOPTN	Not used			1*

Notes:
1) The keyword is followed by any numbers records with each record terminated by a “/” and the keyword should be terminated by a “/”.

Table 12.11: 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.

12.3.16 GRUPNET – DEFINE GROUP STANDARD NETWORK PARAMETERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

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 WELSPEDS 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 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 modelling 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 I* or set to a negative number.			I*
		psia	barsa	atma	
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: <ol style="list-style-type: none"> 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 I* or zero, then GRPNAME is not part of the network. 4) IF VFPTAB is set equal to 9999 then this implies that there is no pressure change between the LOWER and HIGHER group. If a non-zero value is entered then the vertical lift performance tables must be entered via the VFPPROD or VFPINJ keyword in the SCHEDULE section.			0

No.	Name	Description			Default
		Field	Metric	Laboratory	
4	ALQ-PIPE	<p>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.</p> <p>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.</p> <p>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.</p>			0.0
5	OPTIONI	<p>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. OPTIONI should be set to one of the following character strings:</p> <p>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.</p> <p>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.</p> <p>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.</p> <p>Only groups containing wells can use OPTIONI equal to YES or NO, a group without wells should set OPTIONI to NO.</p> <p>Numerical convergence controls and iteration limits for wells using OPTIONI set equal to YES are defined via the NETBALAN keyword in the SCHEDULE section.</p>			NO

No.	Name	Description			Default
		Field	Metric	Laboratory	
6	OPTION2	<p>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:</p> <ul style="list-style-type: none">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. <p>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.</p>			NO
7	OPTION3	<p>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:</p> <ul style="list-style-type: none">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. <p>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.</p>			NONE

Notes:

1) The keyword is followed by any numbers records with each record terminated by a “/” and the keyword should be terminated by a “/”.

Table 12.12: 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 from 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)

12.3.17 GRUPRIG – GROUP DRILLING AND WORKOVER RIG SPECIFICATIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

Description

Defines a groups drilling and workover specifications.

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

12.3.18 GRUPTREE – DEFINE GROUP TREE HIERARCHY

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

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 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 the HIGHER group name. Undefined group relationships are automatically assigned to the FIELD group.	None
Notes: 1) The keyword is followed by any numbers records with each record terminated by a "/" and the keyword should be terminated by a "/".			

Table 12.13: 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 from 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.

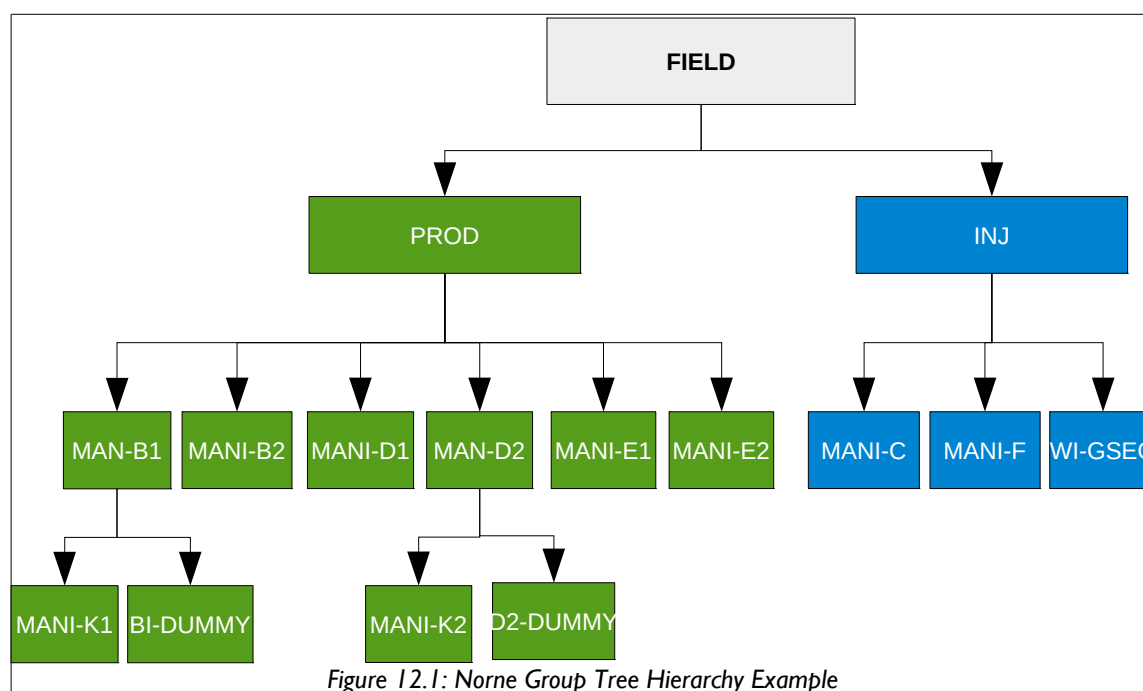


Figure 12.1: Norne Group Tree Hierarchy Example

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.

12.3.19 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. The 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.

12.3.20 MULTPV – MULTIPLY CELL PORE VOLUMES BY A CONSTANT

MULTPV multiplies 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.

12.3.21 MULTX - MULTIPLY CELL TRANSMISSIBILITY IN THE +X DIRECTION

MULTX multiplies 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.

12.3.22 MULTX- - MULTIPLY CELL TRANSMISSIBILITY IN THE -X DIRECTION

MULTX- multiplies 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.

12.3.23 MULTY - MULTIPLY CELL TRANSMISSIBILITY IN THE +Y DIRECTION

MULTY multiplies 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.

12.3.24 MULTY- - MULTIPLY CELL TRANSMISSIBILITY IN THE -Y DIRECTION

MULTY- multiplies 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.

12.3.25 MULTZ - MULTIPLY CELL TRANSMISSIBILITY IN THE +Z DIRECTION

MULTZ multiplies 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.

12.3.26 MULTZ- - MULTIPLY CELL TRANSMISSIBILITY IN THE -Z DIRECTION

MULTZ- multiplies 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-I) and (I, J, K).

See [MULTZ- - Multiply Cell Transmissibility in the -Z Direction](#) in the GRID section for a full description.

12.3.27 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 performed 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.

12.3.28 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 be solved then the time step will be chopped back again to perhaps to less than one day. Using the 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	NSTEPI	NSTEPI is a real positive value that defines the maximum length of the next time step.			None
		days	days	hours	
2	NSTEP2	NSTEP2 is a character string that should be set to either NO or YES to state if the NSTEPI should be applied to future reporting time steps. <div>1) NO: Means that NSTEPI should not be applied to subsequent reporting time steps.</div> <div>2) YES: means that STEP1 should be applied to subsequent reporting time steps.</div> The default value of NO means that NSTEPI will only be applied once.			NO

Notes:

2) The keyword is terminated by “/”.

Table 12.14: NEXTSTEP Keyword Description

See also the DATES and TSTEP keyword 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 Flow](#) 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  JUL    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, 2021), the simulator will progressively 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.

12.3.29 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.

See section [2.2 Running Flow](#) on how to set the numerical control parameters for OPM Flow.

12.3.30 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 $f_w = \frac{q_w}{q_w + q_o}$.			None
		dimensionless	dimensionless	dimensionless	
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.			None
		dimensionless	dimensionless	dimensionless	
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 12.15: 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 /

12.3.31 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.

12.3.32 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.

12.3.33 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 X in the PROPS section for a full description.

12.3.34 PLYSHEAR – ACTIVATE AND DEFINE POLYMER SHEARING PARAMETERS

The PLYSHEAR keyword activates and defines the polymer shear thinning-thickening option for when the polymer option has been activated by the POLYMER keyword in the RUNSPEC section.

See X in the PROPS section for a full description.

12.3.35 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 [PLYSHEAR – Activate and Define Polymer Shearing Parameters](#) in the PROPS section for a full description.

12.3.36 PLYVISC – DEFINE POLYMER VISCOSITY SCALING FACT

PLYVISC 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.

12.3.37 PRORDER – DEFINE A GROUP PRODUCTION RULES SEQUENCE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

Description

Defines the order of group production rules to be implemented when a group's target is not satisfied.

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

12.3.38 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.

12.3.39 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.

12.3.40 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.

12.3.41 RPTSCHED – DEFINE SCHEDULE SECTION REPORTING

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

This keyword defines the data in the SCHEDULE section that is to be printed to the output print file in human readable format. The keyword has two distinct forms, the first of which consists of the keyword followed by a series of integers on the next line indicating the data to be printed (see the first example). This is the original format in the commercial simulator and was subsequently superseded by the second format. The second format consists of the keyword followed by a series of character strings that indicate the data to be printed. In most cases the character string is the keyword used to define the data in the OPM Flow input deck, for example WELSPECS to defined the basic well definitions. 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	<p>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:</p> <ol style="list-style-type: none"> 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. <p>See the RPTRST keyword in the SOLUTION section for a more flexible way to write out restart files.</p>	
4			
....		
Notes: 1) The keyword is terminated by “/”.			

Table 12.16: 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 in 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  JUL  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.

12.3.42 SAVE – ACTIVATES 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 – Activates 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.

12.3.43 SCHEDULE - DEFINE THE START OF THE SCHEDULE SECTION OF KEYWORDS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	-----------------

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.

12.3.44 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.

12.3.45 SKIPREST – ACTIVATE SKIPPING OF RESTART SCHEDULE DATA

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

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
```

12.3.46 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 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 enables the size of the SUMMARY files to be reduced depending on the size of the time interval. However, the keyword will produce irregular time steps reports of the SUMMARY data.

See [SUMTHIN – Define SUMMARY DATA Reporting Time Steps](#) in the SUMMARY 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.

12.3.47 TSTEP – ADVANCE SIMULATION BY 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 TSTEP data sets or keywords may be entered to advance the simulator to the next report time.

No.	Name	Description			Default
		Field	Metric	Laboratory	
I	TSTEP	A vector of real positive numbers that define the length of the time intervals to subsequent report steps..			None
		days	days	hours	
Notes: I) The keyword is followed by a vector of numbers separated by a space and the keyword is terminated by a “/”.					

Table 12.17: TSTEP Keyword Description

See also the DATES keyword 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

The first example shows how to advance the simulation via the reporting time steps from the given start date of January 1, 2022 set via the START keyword in the RUNSPEC section, to the next year, without any actions or reporting taking place.

```
-- =====
--
-- SCHEDULE SECTION
--
-- =====
SCHEDULE
--
-- -----
-- SCHEDULE SECTION - 2022-01-01
-- -----
--
-- ADVANCE SIMULATION BY REPORTING TIME
--
-- JAN FEB MAR APR MAY JUN JULY 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
/
```

Note if the simulation 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 honoured.

12.3.48 TUNING - NUMERICAL TUNING CONTROL

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 TUNINGDP keyword in the SCHEDULE section that is optimized for high throughput runs.

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

See section [2.2 Running Flow](#) and the `use_TUNING` option to have OPM Flow read the first record of the TUNING keyword.

12.3.49 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.

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

See section [2.2 Running Flow](#) and the `use_TUNING` option to have OPM Flow read the first record of the TUNING keyword and how to set the numerical controls for OPM Flow..

12.3.50 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 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.

12.3.51 VFPINJ – DEFINE INJECTION VERTICAL FLOW PERFORMANCE TABLES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

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 12.18. 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 WELSPRES 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 I*.			I*
		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 I*.			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.			None
		Liquid: stb Gas: Mscf	Liquid: sm ³ Gas: sm ³	Liquid: scc Gas: scc	
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.			None
		psia	barsa	atma	
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.			None
	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	
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 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 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 12.18: VFPINJ Keyword Description

See also the WELSPFCS 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 than 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.

12.3.52 VFPPROD – DEFINE PRODUCTION VERTICAL FLOW PERFORMANCE TABLES

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 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 12.19. 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	
I-5	GFR	<p>A defined character string that defines the flowing gas fraction and should be set to one of the following character strings:</p> <p>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'</p> <p>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</p> <p>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.</p>			None
I-6	VFPTYPE	A defined character string that should be defaulted or set equal to THP.			THP
I-7	ALQ	<p>A defined character string that defines the artificial lift quantity and should be set to one of the following character strings:</p> <p>1) GRAT: for the artificial lift quantity being the gas lift gas injection rate.</p> <p>2) IGLR: for the artificial lift quantity being the gas lift gas, injection gas-liquid ratio.</p> <p>3) TGLR: for the artificial lift quantity being the gas lift gas, injection total gas-liquid ratio.</p> <p>4) COMP: for the artificial lift quantity being the compressor power, for a compressor.</p> <p>5) PUMP: for the artificial lift quantity being the pump rating for a pump.</p> <p>6) DENO: for oil surface density.</p> <p>7) DENG: for gas surface density.</p> <p>The default value of I* is " " or undefined that covers the case when the ALQ variable is not entered.</p>			I*
I-8	VFPUNITS	<p>Units used for the BHP-DATA on this keyword.</p> <p>This variable is ignored by OPM Flow and should be defaulted with I*.</p>			I*
		FIELD	METRIC	LAB	
I-9	VFPVALUE	<p>A defined character string that should be defaulted or set equal to BHP.</p> <p>This variable is ignored by OPM Flow and should be defaulted with I*.</p>			BHP
I-10	/	Record terminated by a "/"			Not Applicable
2-I	FLO-DATA	<p>A real positive monotonically increasing vector that defines the numerical values of the flowing phase declared by the FLOW variable.</p> <p>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.</p>			None
		<p>Liquid: stb</p> <p>Gas: Mscf</p>	<p>Liquid: sm³</p> <p>Gas: sm³</p>	<p>Liquid: scc</p> <p>Gas: scc</p>	

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.			None
		psia	barsa	atma	
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.			None
		WOR: dimensionless WCT: dimensionless WGR: stb/Mscf	dimensionless dimensionless dimensionless	dimensionless dimensionless dimensionless	
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.			None
		GOR: Mscf/stb GLR: Mscf/stb OGR: stb/Mscf	dimensionless dimensionless dimensionless	dimensionless dimensionless dimensionless	
5-2	/	Record terminated by a “/”			Not Applicable
6-1	ALQ-DATA	A real positive monotonically increasing vector that defines the numerical values of the artificial lift quantity declared by the ALQ variable. The number of entries must greater than two and less than or equal to MXMALQ as defined on the VFPPDIMS keyword in the RUNSPEC section.			None
		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	
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.			None
	psia	barsa	atma		
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 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 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 12.19: 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 VFPIJ 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.

12.3.53 WCONHIST – DEFINE WELL HISTORICAL PRODUCTION RATES AND PRESSURES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

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 than 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	<p>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:</p> <ol style="list-style-type: none"> 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). <p>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.</p>			None
4	ORAT	A real positive value that defines the observed surface oil production rate target or constraint.			Defined
		stb/d 0.0	sm ³ /day 0.0	scc/hour 0.0	
5	WRAT	A real positive value that defines the observed surface water production rate target or constraint.			Defined
		stb/d 0.0	sm ³ /day 0.0	scc/hour 0.0	
6	GRAT	A real positive value that defines the observed surface gas production rate target or constraint			Defined
		Mscf/d 0.0	sm ³ /day 0.0	scc/hour 0.0	
7	VFPTAB	<p>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.</p> <p>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.</p> <p>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.</p>			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.			Defined
		psia 0.0	barsa 0.0	atma 0.0	
10	BHP	A real positive value that defines the observed bottom-hole pressure.			Defined
		psia 0.0	barsa 0.0	atma 0.0	
11		Not Used			
12		Not Used			

Notes:
1) The keyword is followed by any numbers records with each record terminated by a “/” and the keyword should be terminated by a “/”.

Table 12.20: 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 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.

12.3.54 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.

12.3.55 WCONINJE – WELL INJECTION TARGETS AND CONSTRAINTS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

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	<p>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:</p> <ol style="list-style-type: none"> 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.			None
		Liquid stb/d Gas Mscf/d	Liquid sm ³ /day Gas sm ³ /day	Liquid scc/hour Gas scc/hour	
6	RESV	A real positive value that defines the maximum reservoir volume injection rate target or constraint.			None
		rtb/d	rm ³ /day	rcc/hour	
7	BHP	<p>A real positive value that defines the maximum bottom-hole pressure target or constraint.</p> <p>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.</p>			Defined
		psia 10,000	barsa 6,895	atma 6,803	
8	THP	A real positive value that defines the maximum tubing head pressure target or constraint.			None
		psia	barsa	atma	
9	VFPTAB	<p>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.</p> <p>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.</p> <p>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.</p>			0

No.	Name	Description			Default
		Field	Metric	Laboratory	
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 numbers records with each record terminated by a “/” and the keyword should be terminated by a “/”.					

Table 12.21: 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.

12.3.56 WCONINJH – WELL HISTORICAL OBSERVED INJECTION RATES AND PRESSURES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

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 than 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
		Liquid stb/d Gas Mscf/d	Liquid sm ³ /day Gas sm ³ /day	Liquid scc/hour Gas scc/hour	

No.	Name	Description			Default
		Field	Metric	Laboratory	
5	BHP	A real positive value that defines the observed bottom-hole pressure.			0.0
		psia	barsa	atma	
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.			None
		psia	barsa	atma	
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 I*.			
9		Not used and should be defaulted with I*.			
10		Not used and should be defaulted with I*.			
11		Not used and should be defaulted with I*.			
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 numbers records with each record terminated by a “/” and the keyword should be terminated by a “/”.					

Table 12.22: 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 GI01 is declared as a gas injection well under gas rate control as TARGET variable is defaulted to rate control by using I* (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 I* is used to indicate the last entry should be used.

12.3.57 WCONPROD – DEFINE WELL PRODUCTION TARGETS AND CONSTRAINTS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

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	<p>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:</p> <ol style="list-style-type: none"> 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.			None
		stb/d	sm ³ /day	scc/hour	
5	WRAT	A real positive value that defines the maximum surface water production rate target or constraint.			None
		stb/d	sm ³ /day	scc/hour	
6	GRAT	A real positive value that defines the maximum surface gas production rate target or constraint			None
		Mscf/d	sm ³ /day	scc/hour	
7	LRAT	A real positive value that defines the maximum surface liquid (oil plus water) production rate target or constraint.			None
		stb/d	sm ³ /day	scc/hour	
8	RESV	A real positive value that defines the maximum reservoir volume production rate target or constraint.			None
		rtb/d	rm ³ /day	rcc/hour	
9	BHP	<p>A real positive value that defines the minimum bottom-hole pressure target or constraint.</p> <p>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.</p>			Defined

No.	Name	Description			Default
		Field	Metric	Laboratory	
		psia 14.70	barsa 1.01325.	atma 1.0	
10	THP	<p>A real positive value that defines the minimum tubing head pressure target or constraint.</p> <p>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.</p>			Defined
		psia 0.0	barsa 0.0	atma 0.0	
11	VFPTAB	<p>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.</p> <p>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.</p> <p>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.</p>			0
12	ALQ-WELL	<p>A real positive value that defines the artificial lift quantity to be used in conjunction with the VFPPROD assigned to the well via VFPTAB variable.</p> <p>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.</p> <p>Note that the units for ALQ-WELL is dependent on the associated variable on the VFPPROD keyword.</p>			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 numbers records with each record terminated by a “/” and the keyword should be terminated by a “/”.					

Table 12.23: 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.

12.3.58 WDFACCOR – GAS FLOW DEPENDENT SKIN FACTOR

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

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^{46-47, 48} and ⁴⁹.

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

⁴⁶ Dake, L.P. *Fundamentals of Reservoir Engineering*, Amsterdam, The Netherlands, Elsevier Science BV (1978) Chapter 8.6, pages 252-257.

⁴⁷ Geertsma, J., 1974. *Estimating the Coefficient of Inertial Resistance in Fluid Flow Through Porous Media*. Soc.Pet.Eng.J., October: 445-450.

⁴⁸ Gewers, C.W.W. and Nichol, L.R., 1969. *Gas Turbulence Factor in a Microvugular Carbonate*. J.Can.Pet.Tech., April.

⁴⁹ Wong, S.W., 1970. *Effects of Liquid Saturation on Turbulence Factors for Gas Liquid Systems*. J.Can.Pet.Tech., October

12.3.59 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 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.

12.3.60 WECON – WELL ECONOMIC CRITERIA FOR PRODUCTION WELLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

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.			0.0
		stb/d	sm ³ /day	scc/hour	
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.			0.0

No.	Name	Description			Default
		Field	Metric	Laboratory	
		Mscf/d	sm ³ /day	scc/hour	
4	WCUT	<p>A real positive value that defines the maximum economic surface water cut, above which an economic action will take place.</p> <p>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).</p> <p>A value less than or equal to zero switches off this criterion.</p>			0.0
		dimensionless	dimensionless	dimensionless	
5	GOR	<p>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).</p> <p>A value less than or equal to zero switches off this criterion.</p> <p>Note that this feature is currently not supported in OPM Flow</p>			0.0
		Mscf/stb	sm ³ /sm ³	scc/scc	
6	WGR	<p>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).</p> <p>A value less than or equal to zero switches off this criterion.</p> <p>Note that this feature is currently not supported in OPM Flow.</p>			0.0
		stb/Mscf	sm ³ /sm ³	scc/scc	
7	ACTION	<p>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:</p> <ol style="list-style-type: none"> 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. <p>The corrective action takes places at the end of the time step in which the constraint is violated.</p> <p>Only ACTION equal to CON is currently supported by OPM Flow.</p>			None
8	END	<p>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:</p> <ol style="list-style-type: none"> 1) NO: no action is taken and the run continues. 2) YES: terminate the run at the next report time step. <p>Note that this feature is currently not supported in OPM Flow</p>			NO
9		Not used			

No.	Name	Description			Default
		Field	Metric	Laboratory	
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 numbers records with each record terminated by a “/” and the keyword should be terminated by a “/”.					

Table 12.24: 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.

12.3.61 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.

Note that GEFAC is implemented in OPM Flow.

12.3.62 WELCNTL – MODIFY WELL CONTROL AND TARGETS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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.

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	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 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 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 anliquid 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			None
	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	same as	same as	
		VFPPROD or VFPINJ	VFPPROD or VFPINJ	VFPPROD or VFPINJ	

Notes:
1) The keyword is followed by any numbers records with each record terminated by a “/” and the keyword should be terminated by a “/”.

Table 12.25: 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.

12.3.63 WELOPEN – DEFINE WELL AND WELL CONNECTIONS FLOWING STATUS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

The WELOPEN keyword defines the status of wells and well 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, WCONDINJE, or WCONDINJH 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.			I*
4	J	An integer less than or equal to NY that defines the connection location in the J-direction.			I*
5	K	An integer less than or equal to NZ that defines the connection location in the K-direction.			I*
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.			I*
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.			I*
Notes:					
1) The keyword is followed by any numbers records with each record terminated by a “/” and the keyword should be terminated by a “/”.					

Table 12.26: WELOPEN Keyword Description

If variables I, J, K, K1 and K2 are all set to a negative number or defaulted with I* 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.

12.3.64 WELSEGS – DEFINE MULTI-SEGMENT WELLS AND THEIR SEGMENT STRUCTURE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

Description

The WELSEGS keyword defines a well to be a multi-segment well and defines the well's segment structure.

OPM Flow has not implemented multi-segment wells and therefore this keyword is ignored by OPM Flow and has no effect on the simulation.

12.3.65 WELSPECS – DEFINE WELL SPECIFICATIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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. 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	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 I* 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	<p>A real value that defines the well drainage radius for the well used to calculate a well's productivity or injectivity index.</p> <p>A default of zero results in the pressure equivalent radius of the grid blocks containing the well connections are used.</p>			0.0
		feet	m	cm	
8	INFLOW	<p>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:</p> <ol style="list-style-type: none"> 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 Goodrich⁵⁰ pressure square inflow equation will be 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.⁵¹ <p>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.</p> <p>Only INFLOW equal to STD and NO are currently implemented in OPM Flow.</p>			STD
9	AUTO	<p>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:</p> <ol style="list-style-type: none"> 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. <p>The corrective action takes places at the end of the time step in which the constraint is violated.</p>			SHUT

⁵⁰ 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.

⁵¹ 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 I* 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 numbers 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 I 2.27: 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.

12.3.66 WELTARG – MODIFY WELL TARGETS AND CONSTRAINTS VALUES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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			None
	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	same as	same as	
		VFPPROD or VFPINJ	VFPPROD or VFPINJ	VFPPROD or VFPINJ	
Notes:					
1) The keyword is followed by any numbers records with each record terminated by a “/” and the keyword should be terminated by a “/”.					

Table 12.28: 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.

12.3.67 WGRUPCON – DEFINE WELL GUIDES FOR GROUP CONTROL

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

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.			-1.0
		dimensionless	dimensionless	dimensionless	

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. <u>This should only be used if the well has been declared an injection via the WCONINJE keyword in the SCHEDULE section.</u>			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.			1.0
		dimensionless	dimensionless	dimensionless	
Notes: 1) The keyword is followed by any numbers records with each record terminated by a “/” and the keyword should be terminated by a “/”.					

Table 12.29: 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.

12.3.68 WHISTCTL - DEFINE WELL HISTORICAL TARGET PHASE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

The WHISTCTL keyword changes the target control for wells declared as history match wells via the WCONHIST keyword in the SCHEDULE section. The target phase is set on the WCONHIST keyword and WHISTCTL overrides this value for all subsequent entries on the WCONHIST keyword.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	TARGET	<p>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.</p> <p>TARGET should be set to one of the following character strings:</p> <ul style="list-style-type: none">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. <p>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.</p>			None
2	END	<p>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:</p> <ul style="list-style-type: none">1) NO: no action is taken and the run continues.2) YES: terminate the run at the next report time step. <p>Wells set to BHP control via the WCONHIST or WHISCTL keywords are ignored. Only END equal to NO is currently supported in OPM Flow.</p>			NO

Notes:

- 1) The keyword is terminated by a “/”.

Table 12.30: WHISTCTL Keyword Description

History matching wells are handled differently than ordinary wells that use the WCONPROD keyword for controlling their production targets and constraints. However, the wells still need to be defined like ordinary production wells using the WELSPECS keyword in the SCHEDULE section. History matching wells are converted to ordinary wells by restating a well's control mode using either the WCONPROD or WELTARG keywords in the SCHEDULE section.

See also the WCONHIST and WCONINJH keywords that are used to define the historical production and injection data. All the aforementioned keywords are described in the SCHEDULE section.

Example

The example below shows the observed gas rates for the OP01 oil producer for the first quarter of 2000.

```
-----
-- 01 JAN 2000 START OF SCHEDULE SECTION
-----
--
--      DEFINE WELL HISTORICAL TARGET PHASE
--
--      CNTL      BHP
--      MODE      STOP
WHISTCTL
--      RESV      NO
--
--      WELL HISTORICAL PRODUCTION CONTROLS
--
-- WELL  OPEN/  CNTL  OIL   WAT   GAS   VFP   VFP   THP   BHP
-- NAME  SHUT   MODE  RATE  RATE  RATE  TABLE  ALFQ  PRES  PRES
WCONHIST
OP01    OPEN   ORAT   15.5E3  100.0  1550   10      1*    900.0  1*    /
/
DATES
01 FEB 2000 /
/
--
--      WELL HISTORICAL PRODUCTION CONTROLS
--
-- WELL  OPEN/  CNTL  OIL   WAT   GAS   VFP   VFP   THP   BHP
-- NAME  SHUT   MODE  RATE  RATE  RATE  TABLE  ALFQ  PRES  PRES
WCONHIST
OP01    OPEN   ORAT   15.2E3  150.0  1520   1*      1*    875.0  3250.0 /
/
DATES
01 MAR 2000 /
/
--
--      WELL HISTORICAL PRODUCTION CONTROLS
--
-- WELL  OPEN/  CNTL  OIL   WAT   GAS   VFP   VFP   THP   BHP
-- NAME  SHUT   MODE  RATE  RATE  RATE  TABLE  ALFQ  PRES  PRES
WCONHIST
OP01    OPEN   ORAT   15.0E3  200.0  1500   1*      1*    850.0  1*    /
/
```

From January 1, 2000 the WCONHIST keyword defines well OP01, which is open and is on oil rate control, to produce 15,500 stb/d oil, with the observed rates of 100 stb/d of water and 15.5 MMscf/d of gas. However the WHISTCTL 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.

12.3.69 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.

12.3.70 WORKLIM – DEFINE WORKOVER TIME FOR A WELL

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

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.

12.3.71 WPAVE – DEFINE WELL BLOCK AVERAGE PRESSURE CALCULATION PARAMETERS

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 be compared with field observed data.

No.	Name	Description			Default
		Field	Metric	Laboratory	
1	WPAVE1	<p>A real dimensionless value that defines the weighting factor between the inner block and the surrounding blocks used to calculate the connection factor weighted average pressures.</p> <p>If WPAVE1 is greater than or equal to zero and less than or equal to one, then the average pressures are calculated 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.</p> <p>If WPAVE1 is equal to zero, then average pressure is calculated based on the pore volumes of the blocks.</p>			0.5
2	WPAVE2	<p>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.</p> <p>If WPAVE2 is equal to one, then the average pressures are calculated based only using the connection factor calculated pressures.</p> <p>If WPAVE2 is equal to zero, then average pressures are calculated based on only using the pore volumes calculated pressures.</p>			1.0
3	WPAVE3	<p>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:</p> <ol style="list-style-type: none"> 1) WELL: the hydrostatic head is calculated using the density of the fluid in the wellbore at the well connections. 2) RES: the 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
4	WPAVE4	<p>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:</p> <ol style="list-style-type: none"> 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

No.	Name	Description			Default
		Field	Metric	Laboratory	
Notes: 1) The keyword should be terminated by a “/”.					

Table 12.31: WPAVE Keyword Description

See also the WELSPEC 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.

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 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.

12.3.72 WPAVEDEP – DEFINE WELL REFERENCE DEPTH FOR PRESSURE CALCULATIONS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

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 I* 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 numbers records with each record terminated by a “/” and the keyword should be terminated by a “/”.

Table 12.32: WPAVEDEP Keyword Description

See also the WELPSPECS 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.

12.3.73 WPIMULT – DEFINE WELL CONNECTION MULTIPLIERS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

The WPIMULT keyword defines a well connection multiplier factor that scales the existing well connection values. The resulting effect is scale the well' 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.			I*
4	J	An integer less than or equal to NY that defines the connection location in the J-direction.			I*
5	K	An integer less than or equal to NZ that defines the connection location in the K-direction.			I*
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.			I*
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.			I*
Notes: 1) The keyword is followed by any numbers records with each record terminated by a “/” and the keyword should be terminated by a “/”.					

Table 12.33: WPIMULT Keyword Description

If variables I, J, K, K1 and K2 are all defaulted with zero or I* 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    OPEN   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' /
/
--
--          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*  1*  1*    10
OP03        1.100    1* 1*  1*  35    50
/
```

In this example the WPIMULT scales the well productivity of well OP01 by 1.25, scales the well connection factor in layer 10 only by 0.75 for well OP02, and for OP03, scales the top most 15 well connections by 1.100.

12.3.74 WPITAB - ASSIGN WELL PRODUCTIVITY INDEX VERSUS WATER CUT TABLES

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Description

The WPITAB keyword assigns the well productivity index multiplier versus water cut tables, that are used to scaled a well's connection factors based on the connection's current producing water cut, to a well. The tables are defined via the PIMULTAB keyword in the SCHEDULE section.

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 numbers records with each record terminated by a “/” and the keyword should be terminated by a “/”.

Table 12.34: 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 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.

12.3.75 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.

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.			None
		lb/stb	kg/sm ³	gm/scc	
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.			None
		lb/stb	kg/sm ³	gm/scc	
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 numbers records with each record terminated by a “/” and the keyword should be terminated by a “/”.

Table 12.35: 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.

12.3.76 WRFT – ACTIVATE WELL RFT REPORTING TO THE RFT FILE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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 Schulmberger 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	
I	WELNAME	<p>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.</p> <p>If the WELNAME is left blank then the data is written out for all wells at the time they are first opened to flow.</p> <p>If the WELNAME is given, then the RFT data for the well at the time step the keyword is invoked is written out.</p>			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 12.36: 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.

12.3.77 WRFTPLT – ACTIVATE WELL RFT AND PLT REPORTING TO THE RFT FILE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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 activates 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 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 <u>reporting</u> 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 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 <u>reporting</u> 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 12.37: 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.

12.3.78 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 has not implemented multi-segment wells and therefore this keyword is ignored by OPM Flow and has no effect on the simulation.

12.3.79 WSOLVENT - DEFINE GAS INJECTION WELL SOLVENT FRACTION

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

Description

WSOLVENT defines a gas injection well's solvent fraction in the injection stream that is to be used when the solvent option 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 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
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.			None
Notes: 1) The keyword is followed by any numbers records with each record terminated by a “/” and the keyword should be terminated by a “/”.					

Table 12.38: 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.

12.3.80 WTEMP – DEFINE AN INJECTION WELL'S FLUID TEMPERATURE

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

Description

The WELTEMP keyword defines the temperature of the injection fluid being injected by an injection well. This keyword should only be used if the temperature option has been invoked by the TEMP keyword in the RUNSPEC section.

OPM Flow has not implemented temperature modeling and therefore this keyword is ignored by OPM Flow and has no effect on the simulation.

12.3.81 WTEST – WELL TESTING CRITERIA FOR RE-OPENING CLOSED WELLS

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

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.			None
		days	days	hours	
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.			“ ”
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 prorated the rate at which the well is put back on production. If START is large compared to the time step size, then the well is brought on gradually, if it is less then the well is opened faster. The default value of 0.0 means the well is opened immediately.			0.0
		days	days	hours	

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 12.39: 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.

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 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.

12.3.82 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.

12.3.83 ZIPPY2 – ACTIVATE AUTOMATIC TIME STEP CONTROL

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

Description

The ZIPPY2 keyword activates the commercial simulator's alternative automatic time step selection algorithm that assumes no prior knowledge of the problem, as opposed to the standard time step algorithm that is controlled via the TUNNING keyword in the SCHEDULE section, combined with posterior knowledge gained from previous time steps.

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

See section [2.2 Running Flow](#) on how to control time stepping for OPM Flow.

13 KEYWORD INDEX - ALPHABETIC LISTING

A

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Alphabetic Listing of Keywords Starting with the Letter A	Status
<i>ACTDIMS – ACTION Keyword Dimensions</i>	
<i>ACTNUM – Set the Status of a Grid Block To Active or Inactive</i>	
<i>ADD – Add a Constant to a Specified Array</i>	
<i>ADDREG – Add a Constant to an Array based on a Region Number</i>	
<i>ADSALNOD – Salt Concentration Based on SATNUM Array</i>	
<i>AIMS - Intelligent Time Stepping Activation</i>	
<i>ALL – Export Standard Summary Variable Vectors to File</i>	
<i>API – Activate API Tracking</i>	
<i>AQUANCON - Analytic Aquifer Assignment</i>	
<i>AQUCON – Numerical Aquifers Connection to the Grid</i>	
<i>AQUDIMS – Aquifer Dimensions</i>	
<i>AQUFETP – Fetkovich Aquifer Definition</i>	
<i>AQUNUM – Numerical Aquifer Assignment</i>	
<i>ASPKDAM – Define Asphaltene Permeability Damage</i>	
<i>ASPPW2D – Define Asphaltene Two Parameters Precipitation Data</i>	
<i>ASPREWG -Define Asphaltene as Percentage Weight</i>	
<i>ASPWETF – Define Asphaltene Wettability Factor Data</i>	

B

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

Alphabetic Listing of Keywords Starting with the Letter B	Status
BOX - Define a Range of Grid Blocks to Enter Property Data	

C

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

Alphabetic Listing of Keywords Starting with the Letter C	Status
CECON – Define Well Connections Economic Limit Criteria	
CIRCLE – Completion of Radial Grid Circle Activation	
COALNUM – Define the Coal Region Numbers	
COMPDAT – Define Well Connections to the Grid	
COMPIMB – Assign Imbibition Saturation Tables to Well Connections	
COMPLUMP – Assign Well Connections to Completions	
COMPORD - Define Well Connection Ordering	
COMPSEGS – Define Well Connections for Multi-Segment Wells	
COORD – Define a Set of Coordinates Lines for a Reservoir Grid	
COORDSYS – Define Coordinate Grid Options	
COPY – Copy Array Data to Another Array	
COPYREG – Copy an Array to Another Array based on a Region Number	
CPR – Activate Constrained Pressure Residual (“CPR”) Linear Solver	

D

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Alphabetic Listing of Keywords Starting with the Letter D	Status
<i>DATE - Activate the DATE Option for the SUMMARY File</i>	
<i>DATES – Advance Simulation by Reporting Date</i>	
<i>DATUM – Define the Datum Depth for the Model</i>	
<i>DATUMR – Define Datum Depths for the FIPNUM Regions</i>	
<i>DEBUG – Define the Debug Data to be Printed to File</i>	
<i>DENSITY – Define the Surface Oil, Water Gas Densities for the Fluids</i>	
<i>DEPTH - Edits the Depth at the Center of Each Cell</i>	
<i>DIMENS – Define the Dimension of the Model</i>	
<i>DISGAS – Define Dissolved Gas to be Present in the Model</i>	
<i>DRSDT – Solution Gas (Rs) Maximum Rate of Increase Parameters</i>	
<i>DRV - Define the Size of Grid Blocks in the R Direction via a Vector</i>	
<i>DRVDT – Solution Oil (Rv) Maximum Rate of Increase Parameters</i>	
<i>DTHETAV - Sets the Size of Grid Blocks in THETA Direction via a Vector</i>	
<i>DUMPFLUX – Activate Writing Out of a Flux File</i>	
<i>DX - Define the Size of Grid Blocks in the X Direction for All Cells</i>	
<i>DXV - Define the Size of Grid Blocks in the X Direction via a Vector</i>	
<i>DY - Define the Size of Grid Blocks in the Y Direction for All Cells</i>	
<i>DYV - Define the Size of Grid Blocks in the Y Direction via a Vector</i>	
<i>DZ - Define the Size of Grid Blocks in the Z Direction for All Cells</i>	
<i>DZV - Define the Size of Grid Blocks in the Z Direction via a Vector</i>	

E

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Alphabetic Listing of Keywords Starting with the Letter E	Status
<i>ECHO – Activate Echoing of User Input Files to the Print File</i>	
<i>EDIT - Define the Start of the EDIT Section of Keywords</i>	
<i>EDITNNC – Edit Non-Neighbor Connections Between Cells Manually</i>	
<i>EHYSTR – Define Hysteresis Model and Parameters</i>	
<i>END – Define the End of the Input File</i>	
<i>ENDBOX – Define the End of the BOX Defined Grid</i>	
<i>ENDINC – Define the End of an Include File</i>	
<i>ENDNUM – Define the End-Point Scaling Depth Region Numbers</i>	
<i>ENDSCALE – Activate Relative Permeability End-Point Scaling Option</i>	
<i>ENDSKIP – Deactivates Skipping of Keywords and Input Data</i>	
<i>ENKRVD – Define Relative Permeability End-Points versus Depth Functions</i>	
<i>ENKRVDX ENKRVD – Define Relative Permeability End-Points versus Depth Functions</i>	
<i>ENKRVDX- ENKRVD – Define Relative Permeability End-Points versus Depth Functions</i>	
<i>ENKRVDY ENKRVD – Define Relative Permeability End-Points versus Depth Functions</i>	
<i>ENKRVDY- ENKRVD – Define Relative Permeability End-Points versus Depth Functions</i>	
<i>ENKRVDZ ENKRVD – Define Relative Permeability End-Points versus Depth Functions</i>	
<i>ENKRVDZ- ENKRVD – Define Relative Permeability End-Points versus Depth Functions</i>	
<i>ENPTVD – Define Relative Permeability Saturation End-Points versus Depth</i>	
<i>ENPTVDX ENPTVD – Define Relative Permeability Saturation End-Points versus Depth</i>	
<i>ENPTVDX- ENPTVD – Define Relative Permeability Saturation End-Points versus Depth</i>	
<i>ENPTVDY ENPTVD – Define Relative Permeability Saturation End-Points versus Depth</i>	
<i>ENPTVDY- ENPTVD – Define Relative Permeability Saturation End-Points versus Depth</i>	
<i>ENPTVDZ ENPTVD – Define Relative Permeability Saturation End-Points versus Depth</i>	
<i>ENPTVDZ- ENPTVD – Define Relative Permeability Saturation End-Points versus Depth</i>	
<i>EQLDIMS – Define the Equilibration Data Dimensions</i>	
<i>EQLNUM – Define the Equilibration Region Numbers</i>	
<i>EQLOPTS – Activates the Equilibration Options</i>	
<i>EQUALREG – Sets an Array to a Constant by Region Number</i>	
<i>EQUALS – Sets a Specified Array to a Constant</i>	
<i>EQUIL – Define the Equilibration Initialization Data</i>	

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

Alphabetic Listing of Keywords Starting with the Letter E	Status
EXCEL - Activate the EXCEL Option for the SUMMARY File	
EXTRAPMS – Activate Extrapolation Warning Messages	

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	
FIELD – Activates the Oil Field System of Units for the Model	
FILLEPS – Activate Saturation End-Point Export to the INIT File	
FIPNUM – Define the Fluid In-Place Region Numbers	
FIPOWG – Activate Oil, Gas, and Water FIP Zone Reporting	
FLUXNUM – Define the Flux Regions	
FMTIN – Activates The Format Input File Option	
FMTOUT – Activates The Format Output File Option	
FWSET - Export Well Status Vectors for the Field to File	
FULLIMP – Activates Fully Implicit Solution Option	

G

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

Alphabetic Listing of Keywords Starting with the Letter G	Status
GAS – Define the Gas Phase to be Present in the Model	
GCONINJE – Group Injection Targets and Constraints	
GCONPROD – Group Production Targets and Constraints	
GDORIENT - Define Grid Orientation Parameters	
GECON – Group Economic Criteria for Production Groups	
GEFAC – Define Group Efficiency	
GWSET - Export Well Status Vectors by Group to File	
GRAVITY – Define the Surface Oil, Water Gas Gravities for the Fluids	
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	
GRUPNET – Define Group Standard Network Parameters	
GRUPRIG – Group Drilling and Workover Rig Specifications	
GRUPTREE – Define Group Tree Hierarchy	

H

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

Alphabetic Listing of Keywords Starting with the Letter H	Status
There Are Keywords Beginning with the Letter H	

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

Alphabetic Listing of Keywords Starting with the Letter I	Status
<i>IMBNUM – Define the Imbibition Saturation Table Region Numbers</i>	
<i>IMBNUMX IMBNUM – Define the Imbibition Saturation Table Region Numbers</i>	
<i>IMBNUMY IMBNUM – Define the Imbibition Saturation Table Region Numbers</i>	
<i>IMBNUMZ IMBNUM – Define the Imbibition Saturation Table Region Numbers</i>	
<i>IMBNUMX- IMBNUM – Define the Imbibition Saturation Table Region Numbers</i>	
<i>IMBNUMY- IMBNUM – Define the Imbibition Saturation Table Region Numbers</i>	
<i>IMBNUMZ- IMBNUM – Define the Imbibition Saturation Table Region Numbers</i>	
<i>IMKRVD – Imbibition Relative Permeability End-Points versus Depth Functions</i>	
<i>IMKRVDX IMKRVD – Imbibition Relative Permeability End-Points versus Depth Functions</i>	
<i>IMKRVDX- IMKRVD – Imbibition Relative Permeability End-Points versus Depth Functions</i>	
<i>IMKRVDY IMKRVD – Imbibition Relative Permeability End-Points versus Depth Functions</i>	
<i>IMKRVDY- IMKRVD – Imbibition Relative Permeability End-Points versus Depth Functions</i>	
<i>IMKRVDX IMKRVD – Imbibition Relative Permeability End-Points versus Depth Functions</i>	
<i>IMKRVDZ- IMKRVD – Imbibition Relative Permeability End-Points versus Depth Functions</i>	
<i>IMPES – Activates Implicit Pressure Explicit Saturation Solution Option</i>	
<i>IMPTVD – Imbibition Relative Permeability Saturation End-Points versus Depth</i>	
<i>IMPTVDX IMPTVD – Imbibition Relative Permeability Saturation End-Points versus Depth</i>	
<i>IMPTVDX- IMPTVD – Imbibition Relative Permeability Saturation End-Points versus Depth</i>	
<i>IMPTVDY IMPTVD – Imbibition Relative Permeability Saturation End-Points versus Depth</i>	
<i>IMPTVDY- IMPTVD – Imbibition Relative Permeability Saturation End-Points versus Depth</i>	
<i>IMPTVDZ IMPTVD – Imbibition Relative Permeability Saturation End-Points versus Depth</i>	
<i>IMPTVDZ- IMPTVD – Imbibition Relative Permeability Saturation End-Points versus Depth</i>	
<i>INCLUDE – Load Another Data File at the Current Position</i>	
<i>INIT – Activates the INIT File Option</i>	
<i>IPCG – End-Point Scaling of Grid Cell Gas Capillary Pressure (Imbibition)</i>	
<i>IPCW – End-Point Scaling of Grid Cell Water Capillary Pressure (Imbibition)</i>	
<i>INRAD – Define the Inner Radius of a Radial Grid</i>	
<i>ISGCR – End-Point Scaling of Grid Cell Critical Gas Saturation (Imbibition)</i>	
<i>ISGL – End-Point Scaling of Grid Cell Connate Gas Saturation (Imbibition)</i>	
<i>ISGU – End-Point Scaling of Grid Cell Maximum Gas Saturation (Imbibition)</i>	

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

Alphabetic Listing of Keywords Starting with the Letter I	Status
<i>ISOGCR – End-Point Scaling of Grid Cell Critical Oil Saturation to Gas (Imbibition)</i>	
<i>ISOLNUM – Define the Independent Reservoir Regions</i>	
<i>ISOWCR – End-Point Scaling of Grid Cell Critical Oil Saturation to Water (Imbib.)</i>	
<i>ISWCR – End-Point Scaling of Grid Cell Critical Water Saturation (Imbibition)</i>	
<i>ISWL – End-Point Scaling of Grid Cell Connate Water Saturation (Imbibition)</i>	
<i>ISWU – End-Point Scaling of Grid Cell Maximum Water Saturation (Imbibition)</i>	

J

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

Alphabetic Listing of Keywords Starting with the Letter J	Status
JFUNC - Activates the Leverett J-function Option	

K

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Alphabetic Listing of Keywords Starting with the Letter K	Status
<i>KRG – End-Point Scaling of Grid Cell Krg(Sgu) (Drainage)</i>	
<i>KRGX KRG – End-Point Scaling of Grid Cell Krg(Sgu) (Drainage)</i>	
<i>KRGX- KRG – End-Point Scaling of Grid Cell Krg(Sgu) (Drainage)</i>	
<i>KRGY KRG – End-Point Scaling of Grid Cell Krg(Sgu) (Drainage)</i>	
<i>KRGY- KRG – End-Point Scaling of Grid Cell Krg(Sgu) (Drainage)</i>	
<i>KRGZ KRG – End-Point Scaling of Grid Cell Krg(Sgu) (Drainage)</i>	
<i>KRGZ- KRG – End-Point Scaling of Grid Cell Krg(Sgu) (Drainage)</i>	
<i>KRGR – End-Point Scaling of Grid Cell Krgr(1-Sogcr) (Drainage)</i>	
<i>KRGRX KRGR – End-Point Scaling of Grid Cell Krgr(1-Sogcr) (Drainage)</i>	
<i>KRGRX- KRGR – End-Point Scaling of Grid Cell Krgr(1-Sogcr) (Drainage)</i>	
<i>KRGY KRGR – End-Point Scaling of Grid Cell Krgr(1-Sogcr) (Drainage)</i>	
<i>KRGY- KRGR – End-Point Scaling of Grid Cell Krgr(1-Sogcr) (Drainage)</i>	
<i>KRGRZ KRGR – End-Point Scaling of Grid Cell Krgr(1-Sogcr) (Drainage)</i>	
<i>KRGRZ- KRGR – End-Point Scaling of Grid Cell Krgr(1-Sogcr) (Drainage)</i>	
<i>KRO – End-Point Scaling of Grid Cell Kro(Swl) (Drainage)</i>	
<i>KROX KRO – End-Point Scaling of Grid Cell Kro(Swl) (Drainage)</i>	
<i>KROX- KRO – End-Point Scaling of Grid Cell Kro(Swl) (Drainage)</i>	
<i>KROY KRO – End-Point Scaling of Grid Cell Kro(Swl) (Drainage)</i>	
<i>KROY- KRO – End-Point Scaling of Grid Cell Kro(Swl) (Drainage)</i>	
<i>KROZ KRO – End-Point Scaling of Grid Cell Kro(Swl) (Drainage)</i>	
<i>KROZ- KRO – End-Point Scaling of Grid Cell Kro(Swl) (Drainage)</i>	
<i>KRORG – End-Point Scaling of Grid Cell Kro(Sgcr) (Drainage)</i>	
<i>KRORGX KRORG – End-Point Scaling of Grid Cell Kro(Sgcr) (Drainage)</i>	
<i>KRORGX- KRORG – End-Point Scaling of Grid Cell Kro(Sgcr) (Drainage)</i>	
<i>KRORGY KRORG – End-Point Scaling of Grid Cell Kro(Sgcr) (Drainage)</i>	
<i>KRORGY- KRORG – End-Point Scaling of Grid Cell Kro(Sgcr) (Drainage)</i>	
<i>KRORGZ KRORG – End-Point Scaling of Grid Cell Kro(Sgcr) (Drainage)</i>	
<i>KRORGZ- KRORG – End-Point Scaling of Grid Cell Kro(Sgcr) (Drainage)</i>	
<i>KRORW – End-Point Scaling of Grid Cell Kro(Swcr) (Drainage)</i>	
<i>KRORWX KRORW – End-Point Scaling of Grid Cell Kro(Swcr) (Drainage)</i>	

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Alphabetic Listing of Keywords Starting with the Letter K	Status
KRORWX- KRORW – End-Point Scaling of Grid Cell Kro(Swcr) (Drainage)	
KRORWY KRORW – End-Point Scaling of Grid Cell Kro(Swcr) (Drainage)	
KRORWY- KRORW – End-Point Scaling of Grid Cell Kro(Swcr) (Drainage)	
KRORWZ KRORW – End-Point Scaling of Grid Cell Kro(Swcr) (Drainage)	
KRORWZ- KRORW – End-Point Scaling of Grid Cell Kro(Swcr) (Drainage)	
KRW – End-Point Scaling of Grid Cell Krw(Sw =1.0) (Drainage)	
KRWX KRW – End-Point Scaling of Grid Cell Krw(Sw =1.0) (Drainage)	
KRWX- KRW – End-Point Scaling of Grid Cell Krw(Sw =1.0) (Drainage)	
KRWY KRW – End-Point Scaling of Grid Cell Krw(Sw =1.0) (Drainage)	
KRWY- KRW – End-Point Scaling of Grid Cell Krw(Sw =1.0) (Drainage)	
KRWZ KRW – End-Point Scaling of Grid Cell Krw(Sw =1.0) (Drainage)	
KRWZ- KRW – End-Point Scaling of Grid Cell Krw(Sw =1.0) (Drainage)	
KRWR – End-Point Scaling of Grid Cell KRWR(Sw =1.0) (Drainage)	
KRWRX KRWR – End-Point Scaling of Grid Cell KRWR(Sw =1.0) (Drainage)	
KRWRX- KRWR – End-Point Scaling of Grid Cell KRWR(Sw =1.0) (Drainage)	
KRWRY KRWR – End-Point Scaling of Grid Cell KRWR(Sw =1.0) (Drainage)	
KRWRY- KRWR – End-Point Scaling of Grid Cell KRWR(Sw =1.0) (Drainage)	
KRWRZ KRWR – End-Point Scaling of Grid Cell KRWR(Sw =1.0) (Drainage)	
KRWRZ- KRWR – End-Point Scaling of Grid Cell KRWR(Sw =1.0) (Drainage)	

L

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

Alphabetic Listing of Keywords Starting with the Letter L	Status
LAB - Activates the Laboratory System of Units for the Model	
LGR – Define Local Grid Refinement Parameters	

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	
MAXVALUE – Sets a Maximum Value for an Array Element	
MEMORY – Define Allocated Memory	
MESSAGES – Define Message Print Limits and Stop Limits	
METRIC – Activates the Metric System of Units for the Model	
MINPV – Set a Minimum Grid Block Pore Volume Threshold	
MINVALUE – Set a Minimum Value for an Array Element	
MISCIBLE – Define Miscibility Todd-Longstaff Parameters	
MISNUM – Define the Miscibility Region Numbers	
MONITOR – Activates Output of the Monitoring Data and File	
MSFN – Miscible Normalized Relative Permeability Tables	
MSGFILE – Active or Deactivate Message File Output	
MULTFLT – Multiply the Transmissibility of a Defined Fault by a Constant	
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	
MULTPV – Multiply Cell Pore Volumes by a Constant	
MULTREGP– Multiply Pore Volumes Based On Region Number	
MULTREGT– Multiply Transmissibilities Between Regions	
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
NETBALAN – Network Balancing Parameters	
NEWTRAN – Activate Irregular Corner-Point Grid Transmissibilities	
NEXTSTEP – Maximum Next Time Step Size	
NNC – Define Non-Neighbor Connections Between Cells Manually	
NOCASC – Activate Linear Solver Tracer Algorithm	
NOECHO – Deactivate Echoing of User Input Files to the Print File	
NOGGF – Deactivate Output of Grid Geometry File	
NOINSPEC – Deactivate Output of the INIT Index File	
NOMONITO – Deactivates Output of the Monitoring Data and File	
NONNC – Deactivates Non-Neighbor Connections	
NORSSPEC – Deactivate Output of the RESTART Index File	
NOSIM – Activates the No Simulation Mode for Data File Checking	
NOWARN – Deactivate Warning Messages	
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	

O

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

Alphabetic Listing of Keywords Starting with the Letter O	Status
OIL – Define the Oil Phase to be Present in the Model	
OILVISCT – Oil Viscosity versus Temperature Functions	
OLDTRAN – Activate Cartesian Regular Grid Transmissibilities	
OPERATE – Define Mathematical Operations on Arrays	
OPERNUM – Define Regions for Mathematical Operations on Arrays	
OPTIONS – Activate Various Program Options	
OUTRAD - Define the Outer Radius of a Radial Grid	

P

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Alphabetic Listing of Keywords Starting with the Letter P	Status
PARALLEL – Define Run Configuration	
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)	
PCW – End-Point Scaling of Grid Cell Water Capillary Pressure (Drainage)	
PDEW – Define the Initial Equilibration Dew-Point Pressure for All Grid Blocks	
PDVD – Equilibration Dew-Point versus Depth Tables	
PERMR – Define the Permeability for Each Cell in the R Direction	
PERMX - Define the Permeability in the X Direction for All the Cells	
PERMXX – Define the Permeability Tensor in the XX Direction for All the Cells	
PERMXY – Define the Permeability Tensor in the XY Direction for All the Cells	
PERMY - Define the Permeability in the Y Direction for All the Cells	
PERMYX – Define the Permeability Tensor in the YX Direction for All the Cells	
PERMYZ – Define the Permeability Tensor in the YZ Direction for All the Cells	
PERMZ - Define the Permeability in the Z Direction for All the Cells	
PERMZX – Define the Permeability Tensor in the ZX Direction for All the Cells	
PERMZZ – Define the Permeability Tensor in the ZZ Direction for All the Cells	
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	
PINCHREG - Define Pinch-Out Region Options	
PLMIXPAR – Define the Polymer Todd-Longstaff Mixing Parameters	
PLYADS - Define Polymer Rock Adsorption Tables	
PLYADSS - Define Polymer Rock Adsorption with Salt Dependence Tables	
PLYDHFLF - Define Polymer Thermal Degradation Half-Life Tables	
PLYMAX - Define Polymer-Salt Viscosity Mixing Concentrations	
PLYROCK - Define Polymer-Rock Properties	
PLYSHEAR – Activate and Define Polymer Shearing Parameters	

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

Alphabetic Listing of Keywords Starting with the Letter P	Status
<i>PLYSHLOG - Activate and Define the Polymer Shearing Logarithmic Parameters</i>	
<i>PLYVISC – Define Polymer Viscosity Scaling Factors</i>	
<i>PMISC – Define Miscibility versus Pressure Tables</i>	
<i>POLYMER – Define the POLYMER Phase to be Present in the Model</i>	
<i>PORO - Define the Porosity Values for All the Cells</i>	
<i>PORV - Define the Pore Volumes for All the Cells</i>	
<i>PRESSURE – Define the Initial Equilibration Pressures for All Grid Blocks</i>	
<i>PROPS - Define the Start of the PROPS Section of Keywords</i>	
<i>PRORDER – Define a Group Production Rules Sequence</i>	
<i>PVCDO - Oil PVT Properties for Dead Oil (Constant Compressibility)</i>	
<i>PVDG - Gas PVT Properties for Dry Gas</i>	
<i>PVDO – Oil PVT Properties for Dead Oil</i>	
<i>PVDS - Solvent PVT Properties for the Solvent Model</i>	
<i>PVTG - Gas PVT Properties for Wet Gas</i>	
<i>PVTNUM – Define the PVT Regions</i>	
<i>PVTO - Oil PVT Properties for Live Oil</i>	
<i>PVTW - Define Water Fluid Properties for Various Regions</i>	

Q

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

Alphabetic Listing of Keywords Starting with the Letter Q	Status
There Are Keywords Beginning with the Letter Q	

R

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Alphabetic Listing of Keywords Starting with the Letter R	Status
<i>RADFIN – Define a Radial Local Grid Refinement with One Column</i>	
<i>RADFIN4 – Define a Radial Local Grid Refinement with Four Columns</i>	
<i>RADIAL – Radial Grid Activation Option</i>	
<i>REGDIMS – Define the Maximum Number of Regions for a Region Array</i>	
<i>REGIONS - Define the Start of the REGIONS Section of Keywords</i>	
<i>RESTART – Restart Run From an Existing Restart File</i>	
<i>RKTRMDIR - Activate ROCKTAB Keyword Directional Transmissibility Multipliers</i>	
<i>ROCK - Define the Rock Compressibility for Various Regions</i>	
<i>ROCKCOMP – Activate Rock Compaction</i>	
<i>ROCKNUM – Define Rock Compaction Table Region Numbers</i>	
<i>ROCKOPTS – Define Rock Compaction and Compressibility Options</i>	
<i>ROCKTAB – Rock Compaction Tables</i>	
<i>RPTGRID – Define GRID Section Reporting</i>	
<i>RPTONLY - Activate the Report Time Steps Only Option for the SUMMARY File</i>	
<i>RPTONLYO - Deactivate the Report Time Steps Only Option for the SUMMARY File</i>	
<i>RPTPROPS – Define PROPS Section Reporting</i>	
<i>RPTREGS – Define REGIONS Section Reporting</i>	
<i>RPTSOL – Define SOLUTION Section Reporting</i>	
<i>RPTRST – Define Data to be Written to the RESTART File</i>	
<i>RPTRUNSP – Activates RUNSPEC Reporting</i>	
<i>RPTSCHED – Define SCHEDULE Section Reporting</i>	
<i>RS – Define the Initial Equilibration GOR (Rs) for All Grid Blocks</i>	
<i>RSVD – Equilibration Dissolved Gas-Oil Ratio (Rs) versus Depth Tables</i>	
<i>RTEMP -Define the Initial Reservoir Temperature for the Model</i>	
<i>RTEMPA -Define the Initial Reservoir Temperature for the Model</i>	
<i>RTEMPVD - Define the Initial Reservoir Temperature versus Depth Tables</i>	
<i>RUNSPEC -Define the Start of the RUNSPEC Section of Keywords</i>	
<i>RUNSUM – Activate RSM File Output of the SUMMARY Data</i>	
<i>RV – Define the Initial Equilibration CGR (Rv) for All Grid Blocks</i>	
<i>RVVD – Equilibration Vaporized Oil-Gas Ratio (Rv) versus Depth Tables</i>	

S

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Alphabetic Listing of Keywords Starting with the Letter S	Status
<i>SALNODE – Salt Concentration Based PVTNUM Array</i>	
<i>SATNUM – Define the Saturation Table Region Numbers</i>	
<i>SATOPTS – Activate Relative Permeability Assignment Options</i>	
<i>SAVE – Activates Output of a SAVE File for Fast Restarts</i>	
<i>SCALECRS – Set End-Point Scaling Option</i>	
<i>SCHEDULE - Define the Start of the SCHEDULE Section of Keywords</i>	
<i>SDENSITY – Define the Miscible or Solvent Surface Gas Density</i>	
<i>SEPARATE – Activate the Separate RSM File Output Option</i>	
<i>SGAS – Define the Initial Equilibration Gas Saturation for All Grid Blocks</i>	
<i>SGCR – End-Point Scaling Grid Cell Critical Gas Saturations</i>	
<i>SGCRX SGCR – End-Point Scaling Grid Cell Critical Gas Saturations</i>	
<i>SGCRX- SGCR – End-Point Scaling Grid Cell Critical Gas Saturations</i>	
<i>SGCRY SGCR – End-Point Scaling Grid Cell Critical Gas Saturations</i>	
<i>SGCRY- SGCR – End-Point Scaling Grid Cell Critical Gas Saturations</i>	
<i>SGCRZ SGCR – End-Point Scaling Grid Cell Critical Gas Saturations</i>	
<i>SGCRZ- SGCR – End-Point Scaling Grid Cell Critical Gas Saturations</i>	
<i>SGCWMIS – Miscible Critical Gas versus Water Saturation Functions</i>	
<i>SGFN – Gas Saturation Tables (Format Type 2)</i>	
<i>SGL – End-Point Scaling Grid Cell Connate Gas Saturations</i>	
<i>SGLX SGL – End-Point Scaling Grid Cell Connate Gas Saturations</i>	
<i>SGLX- SGL – End-Point Scaling Grid Cell Connate Gas Saturations</i>	
<i>SGLY SGL – End-Point Scaling Grid Cell Connate Gas Saturations</i>	
<i>SGLY- SGL – End-Point Scaling Grid Cell Connate Gas Saturations</i>	
<i>SGLZ SGL – End-Point Scaling Grid Cell Connate Gas Saturations</i>	
<i>SGLZ- SGL – End-Point Scaling Grid Cell Connate Gas Saturations</i>	
<i>SGOF – Gas-Oil Saturation Tables versus Gas (Format Type 1)</i>	
<i>SGU – End-Point Scaling Grid Cell Gas Saturation</i>	
<i>SGUX SGU – End-Point Scaling Grid Cell Gas Saturation</i>	
<i>SGUX- SGU – End-Point Scaling Grid Cell Gas Saturation</i>	
<i>SGUY SGU – End-Point Scaling Grid Cell Gas Saturation</i>	

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Alphabetic Listing of Keywords Starting with the Letter S	Status
SGUY- <i>SGU – End-Point Scaling Grid Cell Gas Saturation</i>	
SGUZ <i>SGU – End-Point Scaling Grid Cell Gas Saturation</i>	
SGUZ- <i>SGU – End-Point Scaling Grid Cell Gas Saturation</i>	
SGWFN – <i>Gas-Water Saturation Tables (Format Type 2)</i>	
SHRATE - <i>Activate and Define the Polymer Shearing Logarithmic Parameters</i>	
SKIP – <i>Activate Skipping of All Keywords and Input Date</i>	
SKIP100 – <i>Activate Skipping of “Black-Oil” Keywords and Input Date</i>	
SKIP300 – <i>Activate Skipping of “Compositional” Keywords and Input Date</i>	
SKIPREST – <i>Activate Skipping of Restart Schedule Data</i>	
SLGOF – <i>Gas-Oil Saturation Tables versus Gas (Format Type 1)</i>	
SMRYDIMS – <i>Define Maximum Number of Summary Vectors to be Written</i>	
SOF2 – <i>Oil Saturation Tables with Respect to Gas or Water (Format Type 2)</i>	
SOF3 – <i>Oil Saturation Tables with Respect to Gas and Water (Format Type 2)</i>	
SOGCR – <i>End-Point Scaling Grid Cell Critical Oil Saturation with Respect to Gas</i>	
SOGCRX <i>SOGCR – End-Point Scaling Grid Cell Critical Oil Saturation with Respect to Gas</i>	
SOGCRX- <i>SOGCR – End-Point Scaling Grid Cell Critical Oil Saturation with Respect to Gas</i>	
SOGCRY <i>SOGCR – End-Point Scaling Grid Cell Critical Oil Saturation with Respect to Gas</i>	
SOGCRY- <i>SOGCR – End-Point Scaling Grid Cell Critical Oil Saturation with Respect to Gas</i>	
SOGCRZ <i>SOGCR – End-Point Scaling Grid Cell Critical Oil Saturation with Respect to Gas</i>	
SOGCRZ- <i>SOGCR – End-Point Scaling Grid Cell Critical Oil Saturation with Respect to Gas</i>	
SOIL – <i>Define the Initial Equilibration Oil Saturation for All Grid Blocks</i>	
SOLUTION - <i>Define the Start of the SOLUTION Section of Keywords</i>	
SOLVENT – <i>Define the SOLVENT Phase to be Present in the Model</i>	
SORWMIS – <i>Miscible Residual Oil versus Water Saturation Functions</i>	
SOWCR – <i>End-Point Scaling Grid Cell Critical Oil Saturation with Respect to Water</i>	
SOWCRX <i>SOWCR – End-Point Scaling Grid Cell Critical Oil Saturation with Respect to Water</i>	
SOWCRX- <i>SOWCR – End-Point Scaling Grid Cell Critical Oil Saturation with Respect to Water</i>	
SOWCRY <i>SOWCR – End-Point Scaling Grid Cell Critical Oil Saturation with Respect to Water</i>	
SOWCRY- <i>SOWCR – End-Point Scaling Grid Cell Critical Oil Saturation with Respect to Water</i>	
SOWCRZ <i>SOWCR – End-Point Scaling Grid Cell Critical Oil Saturation with Respect to Water</i>	
SOWCRZ- <i>SOWCR – End-Point Scaling Grid Cell Critical Oil Saturation with Respect to Water</i>	
SPECGRID- <i>Define the Dimensions of a Corner-Point Grid</i>	

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Alphabetic Listing of Keywords Starting with the Letter S	Status
SPECHEAT – Define the Specific Heat of Oil, Water and Gas	
SPECROCK – Define the Specific Heat of the Reservoir Rock	
SSFN – Solvent and Gas Relative Permeability Tables	
SSOL – Define the Initial Equilibration Solvent Saturation for All Grid Blocks	
START – Simulation Start Date	
STONE1 – Activate Stone's First Three Phase Oil Relative Permeability Model	
STONE1EX – Define Stone's First Three Phase Oil Relative Permeability Parameter	
STONE2 – Activate Stone's Second Three Phase Oil Relative Permeability Model	
SUMMARY - Define the Start of the SUMMARY Section of Keywords	
SUMTHIN – Define SUMMARY DATA Reporting Time Steps	
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 SWCR – End-Point Scaling Grid Cell Critical Water Saturation	
SWCRX- SWCR – End-Point Scaling Grid Cell Critical Water Saturation	
SWCRY SWCR – End-Point Scaling Grid Cell Critical Water Saturation	
SWCRY- SWCR – End-Point Scaling Grid Cell Critical Water Saturation	
SWCRZ SWCR – End-Point Scaling Grid Cell Critical Water Saturation	
SWCRZ- SWCR – End-Point Scaling Grid Cell Critical Water Saturation	
SWFN – Water Saturation Tables (Format Type 2)	
SWL – End-Point Scaling Grid Cell Connate Water Saturation	
SWLX SWL – End-Point Scaling Grid Cell Connate Water Saturation	
SWLX- SWL – End-Point Scaling Grid Cell Connate Water Saturation	
SWLY SWL – End-Point Scaling Grid Cell Connate Water Saturation	
SWLY- SWL – End-Point Scaling Grid Cell Connate Water Saturation	
SWLZ SWL – End-Point Scaling Grid Cell Connate Water Saturation	
SWLZ- SWL – End-Point Scaling Grid Cell Connate Water Saturation	
SWOF – Water-Oil Saturation Tables (Format Type 1)	
SWU – End-Point Scaling Grid Cell Gas Saturation	
SWUX SWU – End-Point Scaling Grid Cell Gas Saturation	
SWUX- SWU – End-Point Scaling Grid Cell Gas Saturation	
SWUY SWU – End-Point Scaling Grid Cell Gas Saturation	

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

Alphabetic Listing of Keywords Starting with the Letter S	Status
SWUY- <i>SWU – End-Point Scaling Grid Cell Gas Saturation</i>	
SWUZ <i>SWU – End-Point Scaling Grid Cell Gas Saturation</i>	
SWUZ- <i>SWU – End-Point Scaling Grid Cell Gas Saturation</i>	

T

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Alphabetic Listing of Keywords Starting with the Letter T	Status
<i>TABDIMS – Define the Number of Tables and the Table Dimensions</i>	
<i>TEMP – Activate the Temperature Modeling Option</i>	
<i>TEMPVD - Define the Initial Reservoir Temperature versus Depth Tables</i>	
<i>THCONR – Define Rock and Fluid Thermal Conductivity</i>	
<i>THPRES - Define Equilibration Region Threshold Pressures</i>	
<i>TITLE – Define the Title for the Input Deck</i>	
<i>TLMIXPAR – Define the Miscible Todd-Longstaff Mixing Parameters</i>	
<i>TNUM – Define Passive Tracer Concentration Regions</i>	
<i>TOPS - Define the Depth at the Center of the Top Face for Each Cell</i>	
<i>TRACER – Define Passive Tracer Variables</i>	
<i>TRACERS – Activate Tracer Options and Set Tracer Array Dimensions</i>	
<i>TRANX - Define the Transmissibility in the X Direction for All the Cells</i>	
<i>TRANY - Define the Transmissibility in the Y Direction for All the Cells</i>	
<i>TRANZ - Define the Transmissibility in the Z Direction for All the Cells</i>	
<i>TSTEP – Advance Simulation by Reporting Time</i>	
<i>TUNING - Numerical Tuning Control</i>	
<i>TUNINGDP – Numerical Tuning Control for High Throughput Cases</i>	
<i>TVDP – Define the Initial Equilibration Tracer Saturation versus Depth Functions</i>	

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 Feature	
UDQDIMS – Define the Dimensions of the User Defined UDQ Feature	
UNIFIN – Activates The Unified Input File Option	
UNIFOUT – Activates The Unified Output File Option	

V

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

Alphabetic Listing of Keywords Starting with the Letter V	Status
VAPOIL – Define Vaporize Oil in Wet Gas to be Present in the Model	
VAPPARS – Oil Vaporization Parameters	
VFPIDIMS – Injection Vertical Flow Performance Table Dimensions	
VFPINJ – Define Injection Vertical Flow Performance Tables	
VFPIDIMS – Injection Vertical Flow Performance Table Dimensions	
VFPPROD – Define Production Vertical Flow Performance Tables	
VISCREF - Define Viscosity-Temperature Reference Conditions	

W

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
---------	------	------	-------	---------	----------	---------	----------

Alphabetic Listing of Keywords Starting with the Letter W	Status
<i>WARN – Activate Warning Messages</i>	
<i>WATER – Define the Water Phase to be Present in the Model</i>	
<i>WATVISCT – Water Viscosity versus Temperature Functions</i>	
<i>WCONHIST – Define Well Historical Production Rates and Pressures</i>	
<i>WCONINJ – Well Injection Targets and Constraints</i>	
<i>WCONINJE – Well Injection Targets and Constraints</i>	
<i>WCONINJH – Well Historical Observed Injection Rates and Pressures</i>	
<i>WCONPROD – Define Well Production Targets and Constraints</i>	
<i>WDFACCOR – Gas Flow Dependent Skin Factor</i>	
<i>WDRILTIM – Define Drilling Parameters for Automatic Drilling of New Wells</i>	
<i>WECON – Well Economic Criteria for Production Wells</i>	
<i>WEFAC – Define Well Efficiency</i>	
<i>WELCNTL – Modify Well Control and Targets</i>	
<i>WELLDIMS – Define the Wells and Group Dimensions</i>	
<i>WEOPEN – Define Well and Well Connections Flowing Status</i>	
<i>WELSEGS – Define Multi-Segment Wells and Their Segment Structure</i>	
<i>WELSPECS – Define Well Specifications</i>	
<i>WELTARG – Modify Well Targets and Constraints Values</i>	
<i>WGRUPCON – Define Well Guides for Group Control</i>	
<i>WHISTCTL - Define Well Historical Target Phase</i>	
<i>WLIFT – Define Well Re-Tubing, THP and Lift Switching Workover Operations</i>	
<i>WORKLIM – Define Workover Time for a Well</i>	
<i>WPAVE – Define Well Block Average Pressure Calculation Parameters</i>	
<i>WPAVEDEP – Define Well Reference Depth for Pressure Calculations</i>	
<i>WPIMULT – Define Well Connection Multipliers</i>	
<i>WPITAB - Assign Well Productivity Index versus Water Cut Tables</i>	
<i>WPOLYMER - Define Water Injection Well Polymer and Salt Concentrations</i>	
<i>WRFT – Activate Well RFT Reporting to the RFT File</i>	
<i>WRFTPLT – Activate Well RFT and PLT Reporting to the RFT File</i>	
<i>WSEGDIMS – Define Multi-Segment Well Dimensions</i>	

RUNSPEC	GRID	EDIT	PROPS	REGIONS	SOLUTION	SUMMARY	SCHEDULE
-------------------------	----------------------	----------------------	-----------------------	-------------------------	--------------------------	-------------------------	--------------------------

Alphabetic Listing of Keywords Starting with the Letter W	Status
WSEGITER – Define Multi-Segment Wells Iteration Parameters	
WSOLVENT - Define Gas Injection Well Solvent Fraction	
WTEMP – Define An Injection Well's Fluid Temperature	
WTEST – Well Testing Criteria for Re-Opening Closed Wells	
WTRACER – Define An Injection Well's Tracer Concentration	

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	
ZIPPY2 – Activate Automatic Time Step Control	

14 OPM FLOW RELEASE HISTORY

14.1 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, Andreas Lauser)
- opm-parser (Joakim Hove)
- opm-output (Joakim Hove)
- opm-grid (Atgeirr Rasmussen, Robert Klöfkorn, Bård Skaflestad)
- opm-material (Andreas Lauser, Robert Klöfkorn, Tor Harald Sandve)
- opm-core (Atgeirr Rasmussen, Robert Klöfkorn, Bård Skaflestad)
- ewoms (Andreas Lauser, Robert Klöfkorn, Tor Harald Sandve)
- opm-simulators (Atgeirr Rasmussen, Robert Klöfkorn, Tor Harald Sandve, Andreas Lauser)
- opm-upscaling (Arne Morten Kvarving, Atgeirr Rasmussen, Bård Skaflestad)

Andreas Lauser

End of Document