ECLIPSE

ECLIPSE* reservoir simulation software

Version 2010.1

Technical Description

Schlumberger

Proprietary notice

Copyright © 1982 - 2010 Schlumberger. All rights reserved. Reproduction or alteration without prior written permission is prohibited, except as allowed under applicable law.

Use of this product is governed by the License Agreement. Schlumberger makes no warranties, express, implied, or statutory, with respect to the product described herein and disclaims without limitations any warranties of merchantability or fitness for a particular purpose.

Trademarks & service marks

"Schlumberger," the Schlumberger logotype, and other words or symbols used to identify the products and services described herein are either trademarks, trade names, or service marks of Schlumberger and its licensors, or are the property of their respective owners. These marks may not be copied, imitated, or used, in whole or in part, without the express prior written permission of their owners. In addition, covers, page headers, custom graphics, icons, and other design elements may be service marks, trademarks, and/or trade dress of Schlumberger and may not be copied, imitated, or used, in whole or in part, without the express prior written permission of Schlumberger.

Contents

List of Figures 14
List of Tables 17

Chapter 1 - Introduction 23
Overview 23
ECLIPSE features 25
Ancillary programs 36
Additional programs 38

Chapter 2 - Adjoint Gradient Screening 49
Introduction 49
Keywords 50

Chapter 3 - The Alkaline Model 57
Introduction 57
The alkaline model 1 58
Summary of keywords 61
Example problem 63

Chapter 4 - API Tracking 69
Introduction 69

Chapter 5 - Aquifer Modeling Facilities 73
Introduction 73
Numerical aquifer 74
Carter-Tracy aquifer 76
Fetkovich aquifer 79
Constant flux aquifer 81
Constant head aquifers 82
Multi-component analytic aquifers 83
List of keywords 84

Chapter 6 - The Asphaltene Option 89
Introduction 89
Asphaltene Characterization and Precipitation 90
Asphaltene Flocculation-Dissociation 92
Asphaltene Deposition 94
Asphaltene Damage 96
List of keywords 99
Example 102

Chapter 7 - Brine Tracking 103
Introduction 103
The Brine Model 104
Summary of keywords 113
Examples 118

Chapter 8 - CO_2 Injection: Geological Storage, EOR and ECBM 123
Introduction 123

Chapter 9 - CO_2 Storage 135
Introduction 135

Chapter 10 - CO_2/H_2S Solution in the Aqueous Phase 143
Introduction 143
Water component properties 145

Chapter 11 - Chemical Reactions 147
Introduction 147
Chemical reaction keywords 148
Formulation 150
Output 156
Examples 157

Chapter 12 - The Coal Bed Methane model 161
Introduction 161
Modeling compositional effects with ECLIPSE 100 167
Using the ECLIPSE 100 coal bed methane model 169
Summary of keywords ECLIPSE 100 173
Examples 176
Summary of keywords ECLIPSE 300 188
Using the ECLIPSE 300 coal bed methane model 191

Chapter 13 - Compatibility Between ECLIPSE 100 Special Options 195
Introduction 195

Chapter 14 - Convergence Reports 201
Introduction 201

Chapter 15 - Diffusion 211
Introduction 211
Keywords 212
Diffusion coefficients: ECLIPSE 100 215
Diffusion coefficients: ECLIPSE 300 220
The cross phase diffusion option (ECLIPSE 300) 224
The cross phase diffusion option (ECLIPSE 100) 226
Diffusivity calculations 227
Using the molecular diffusion option 231

Chapter 16 - Dual Porosity 233
Introduction 233
Transmissibility calculations 234
Block to block transmissibility calculations 236
Recovery mechanisms 238
Facilities specific to dual porosity runs 246
Restrictions on dual porosity runs 248
Keyword summary 249
Transfer functions 251
Modifying the recovery versus time response (ECLIPSE 100) 252
Scaling the fracture relative permeability (ECLIPSE 100) 253
Solution of the linear equations 254
Calculation of fractional volumes for gravity drainage 256
Variation of the sigma factor with drainage process 258
Rock volume and cell activation in dual porosity thermal calculations 259

Integrated capillary pressure option....260
Discretized matrix model (ECLIPSE 100)....263
Single Medium Conductive Fractures (ECLIPSE 300)....266

Chapter 17 - Environmental Tracers 269

Introduction....269
Using the environmental tracer option....274
Summary of keywords....276
Example problem....280

Chapter 18 - Equations of State.... 287

Two-parameter equations of state....287
Flash calculations....290
Viscosity evaluation....291
Three parameter equations of state....293
Use of the equation of state for hydrocarbon mixtures....294
Calculation of phase states....296

Chapter 19 - Equations of State Regions.... 297

Introduction....297
Reservoir equation of state....298
Surface equation of state....300

Chapter 20 - File Handling in ECLIPSE 303

Introduction....303
File internal format....305
GRAF files....307

Chapter 21 - Flux Boundary Conditions 309

Introduction....309
Using flux boundary conditions....310
Summary of keywords....317

Chapter 22 - The Foam Model.... 319

Introduction....319
The simulation model....320
Using the foam model....326
Summary of keywords....328
Example data set....332

Chapter 23 - Formulation of the Equations 341

Introduction....341
Description of equations....342
Variable sets....343
The IMPES and AIM methods....347
Instability in IMPES....349
Convergence criteria for residual equations....350
Flows....352
Reservoir densities....356
Newton iteration of the non-linear residual....357

Chapter 24 - Gas Field Operations Model.... 359

Introduction....359
Description of the facilities....361

Applying separate gas supply contracts to individual groups 370
Using the gas field operations model 373
Summary of keywords 380

Chapter 25 - Gas Lift Optimization 385

Introduction 385
Using the gas lift optimization facility 392
Summary of keywords 396

Chapter 26 - Gas Plants and NGLs 397

Gas plant tables 397
The recovery plant option 399

Chapter 27 - GASSOL Solution in the Aqueous Phase 401

Introduction 401
Aqueous phase properties 403

Chapter 28 - The GASWAT Option 405

Introduction 405
Equation of state modifications 407
Calorific values 411
Salinity 412
Relative permeability tables 413
Initialization 414
Summary items 415

Chapter 29 - Gas Calorific Value Control 417

Introduction 417
Adjusting guide rates to control mean calorific value 419
Controlling the energy production rate 423
Using the facility 425
Summary of keywords 430

Chapter 30 - Geomechanics 433

Introduction 433
Geomechanics external to ECLIPSE 434
Geomechanics internally within ECLIPSE Compositional 437

Chapter 31 - Gi Pseudo-compositional Model 449

Introduction 449
Using the Gi pseudo-compositional model 453
Summary of keywords 454
Example problem 456

Chapter 32 - The Gradient Option 463

Introduction 463
Data file overview 468
Output from the Gradient option 473
Gradient option technical description 482
Instantaneous gradients 485
Summary of keywords 487
Example problem (ECLIPSE 100 only) 494

Chapter 33 - Group Production Rules 505
Introduction 505
Drilling new wells 506
Opening new connections (re-perforating) 507
Reducing the THP limit 509
Changing VFP table numbers (retubing) 510
Changing well ALQ values (applying lift) 511
Activating compressors 513
The effect of closing wells during drilling and workovers 514
Using production rules with the network option 515

Chapter 34 - Hysteresis 517
Introduction 517
Using the hysteresis option 518
Relative permeability hysteresis in the non-wetting phase 519
Relative permeability hysteresis in the wetting phase 523
Modeling oil wet systems 525
Capillary pressure hysteresis 526

Chapter 35 - Hysteresis in WAG Floods 529
Introduction 529
Using the WAG hysteresis option 530
Non-wetting phase model (Gas) 531
Modification to the residual oil 534
Wetting phase model (Water) 535

Chapter 36 - Independent Reservoir Regions 537
Introduction 537
Using the option.. 538
Merging two or more corner point grids 539
Examples 540

Chapter 37 - Initializing the Study 543
Introduction 543
Data requirements 544
The equilibration algorithm 547
Calculating the initial conditions 548
Non-equilibrium initialization (ECLIPSE 300) 553
Phase Pressures in ECLIPSE 300 555

Chapter 38 - Licenses 557
License Overview 557
Special Cases 561

Chapter 39 - Local Grid Refinement and Coarsening 563
Introduction 563
Local grid refinement 564
Geometry and grid data in LGRs 584
Local grid coarsening 590
Automatic refinement (ECLIPSE 100) 594
Summary of keywords 597
Example LGR problem 601

Chapter 40 - Miscible Flood Modeling 609
Introduction 609
The Todd-Longstaff mixing parameter model 610
Effect of water saturation 611
Relative permeability model 612
Viscosity model 613
Component density model 614
The control of miscibility with pressure 615
Control of numerical diffusion in miscible flow simulation 617
Using the ECLIPSE miscible flood option 619

Chapter 41 - Mixed Water Relative Permeability Modeling 621
Mathematical Model 621

Chapter 42 - Multi-phase flash 623
Introduction 623
Multi-phase keywords 624
Using multi-phase 625

Chapter 43 - Multi porosity 627
Introduction 627
Using the Multi porosity option 628
Examples 633
Limitations 634

Chapter 44 - Multi-segment Wells 635
Introduction 635
The multi-segment well model 637
Pressure drops from VFP tables 645
The drift flux slip model 648
Down-hole separators (ECLIPSE 100) 657
Thermal multi-segment well model 662
Using the multi-segment well model 665
Converting friction wells to multi-segment wells 672
Summary of keywords 674

Chapter 45 - The Network Option 681
Introduction 681
Using the standard network option 686
The extended network model 693
The network model in ECLIPSE 300 702
Summary of keywords 704

Chapter 46 - Nine-point Schemes 707
Introduction 707

Chapter 47 - Non-Darcy Flow 711
Introduction 711
Forchheimer models 714
Velocity-dependent relative permeabilities 716

Chapter 48 - Non-Newtonian fluid flows 723
Introduction 723

Chapter 49 - Optimization 727
Introduction 727
Non-linear Gradient-Based Optimization in ECLIPSE 728
Adjoint Gradient Calculations in ECLIPSE 731
Optimizer Search Direction 734
Line Search 735
Reservoir Optimization in ECLIPSE 736
Guidelines on using the reservoir optimization feature 739
Summary of keywords (ECLIPSE 300 only) 745
Example problem (ECLIPSE 300 only) 747

Chapter 50 - Parallel option 749
Introduction 749
Activating a parallel run 750
Partitioning 751
Running LGRs in parallel (ECLIPSE 100) 756
Performance bottlenecks 757
Memory scaling 759
Summary of keywords 760

Chapter 51 - Petro-elastic Model 763
Introduction 763
Assumptions 764
The Petro-elastic Model 765
Implementation of the Petro-elastic model 773

Chapter 52 - The Polymer Flood Model 777
Introduction 777
Summary of keywords 785
Example problems 789

Chapter 53 - Potential Calculations 799
Phase potentials 799
Well potentials 802

Chapter 54 - Recovery Mechanisms 805
Introduction 805
Solution gas drive 807
Water drive 808
Summary output 809
Details of the recovery calculation 810

Chapter 55 - Reservoir Coupling 811
Introduction 811
Using the reservoir coupling facility 819
Sharing licenses with the slaves 827
Reservoir coupling file structure 828
Summary of keywords 833

Chapter 56 - Restarts 835
Introduction 835
Restarts in ECLIPSE 100 836
Restarts in ECLIPSE 300 841

Chapter 57 - Rivers 845
Introduction 845
River keywords 846
Using the River option..... 850
The River Model..... 851
Examples 855

Chapter 58 - Rock Compaction..... 859
Introduction 859
Tabulated compaction curves..... 860
Water induced compaction (ECLIPSE 100)..... 864
Palmer-Mansoori rock model..... 865
Compositional coal swelling/shrinkage model (ECLIPSE 300)..... 866

Chapter 59 - Saturation Functions 869
Introduction 869
Water saturation properties..... 873
Gas saturation properties..... 875
Oil saturation properties..... 877
Three phase oil relative permeability models..... 879
Table end points 898
Consistency requirements 899
Near critical oil and gas relative permeabilities..... 901

Chapter 60 - Saturation Table Scaling 905
Introduction 905
Scaling of capillary pressure functions..... 907
Scaling of relative permeability functions..... 908
Miscellaneous points..... 911
Special applications 913
Consistency requirements 918
Example of end point scaling..... 919

Chapter 61 - Shale Gas..... 921
Introduction 921

Chapter 62 - Solid Phase..... 925
Introduction 925
Solid keywords..... 926
Solid properties..... 929

Chapter 63 - Solution of the Linear Equations..... 937
Introduction 937
Material balance 938
Nested factorization preconditioning..... 939
Initialization for nested factorization preconditioner..... 941
Solution procedure for nested factorization 942
Orthomin 943
Convergence criteria..... 944
Treatment of wells..... 946
Nested factorization for LGRs..... 947
The ECLIPSE 300 WARP linear solver 949

Chapter 64 - The Solvent Model 955
Introduction 955
The Todd and Longstaff mixing parameter model 957
The relative permeability model 959
The PVT model 963
The pressure dependence of miscibility 966
Using the solvent model 967
Control of numerical diffusion in miscible flow simulation 970
Summary of keywords 972
Example problem 975

Chapter 65 - Surface Tension Effects 983
Evaluation of surface tensions 983
Use of surface tensions 984
Use of the MISCNUM option 987

Chapter 66 - The Surfactant Model 989
Introduction 989
The simulation model 990
Using the surfactant model 995
Summary of keywords 997
Example problem 1000

Chapter 67 - Temperature option 1009
Introduction 1009

Chapter 68 - Tensor Permeability 1015
Introduction 1015
Discretization 1017
Orthogonality error 1019
Restrictions 1021

Chapter 69 - Thermal Option 1023
Introduction 1023
Thermal keywords 1025
Formulation 1037
Thermal properties 1045
Thermal features 1072
Converting a blackoil data set to thermal 1083
Examples 1088
Selected bibliography 1090

Chapter 70 - Timing Reports 1093
Introduction 1093

Chapter 71 - Total Compressibility Checks 1099
Introduction 1099

Chapter 72 - Transmissibility Calculations 1103
Introduction 1103
Cartesian transmissibility calculations 1105
Radial transmissibility calculations 1111
Variations on the transmissibility calculation 1114

Chapter 73 - Tracer Tracking 1117
Introduction 1117
Control of numerical diffusion 1119
Tracking original fluids 1123
Delumping and lumping 1129
Diffusion and dispersion 1130
Treating passive tracers as non-linear 1131

Chapter 74 - Units 1133
The unit conventions 1133
Conversion factors 1136

Chapter 75 - Vertical Equilibrium 1139
Introduction 1139
Using the VE option 1141
The cell saturation distribution 1142
VE relative permeabilities and capillary pressures 1146
Residual flow model 1147
Miscellaneous points 1149
Compressed vertical equilibrium option 1151

Chapter 76 - Water components 1155
Introduction 1155
Water component keywords 1156
Water component properties 1158

Chapter 77 - The Wellbore Friction Option 1163
Introduction 1163
Using the wellbore friction option 1166
Graphical presentation of friction effects in horizontal wells 1172
Summary of keywords 1178
Example problem 1179

Chapter 78 - Well Inflow Performance 1187
Introduction 1187
The connection transmissibility factor 1189
The flow-dependent skin factor 1192
The phase mobilities 1194
Special inflow equations 1195
Injecting connections 1201
The well bore head term 1203
The productivity index 1204

Chapter 79 - Well Modeling Facilities 1207
Well completions. 1207
Well controls and limits 1210
Group and field control facilities 1216
Vertical flow performance 1230
Well name and well list template matching 1232

Appendix A - RUNSPEC section: pre-95A fixed record structure 1235
Compatibility with earlier releases 1235

Appendix B - RPTPROPS: pre-2002A integer controls 1247

Appendix C - RPTGRID - definition with integer controls 1251

Appendix D - RPTRST keyword: pre-2002A integer controls 1257

Appendix E - RPTSCHED: pre-2002A integer controls 1263

Appendix F - RPTSOL: pre-2002A integer controls 1275

Appendix G - ECLIPSE development history 1283

Introduction 1283
New facilities in 2009.2 and 2009.1 1284
New developments in 2008.1 and 2008.2 1333
New developments in 2007.1 1384
New Developments in 2006.1 and 2006.2 1429
Developments in 2005A and 2005A_1 1448
Developments in 2004A 1465
Developments in 2003A 1486
Developments in 2002A 1501
Developments in 2001A 1522
Developments in 2000A 1546
ECLIPSE 100: developments in 99A 1575
ECLIPSE 100: developments in 98A 1581
ECLIPSE 100: developments in 97A 1589
ECLIPSE 100: developments in 96A 1599
ECLIPSE 100: developments in 95A 1608
ECLIPSE 100: developments in 94A 1617
ECLIPSE 100: developments in 93A 1624
ECLIPSE 100: developments in 92A 1631
ECLIPSE 100: developments in 91A 1636
ECLIPSE 100: developments in 90A 1643
ECLIPSE 100: developments in 88/09 1649
ECLIPSE 100: developments in 87/11 1651
ECLIPSE 100: developments in 87/1 1653
ECLIPSE 100: developments in 86/1 1658
ECLIPSE 100: developments in 85/7 1662
ECLIPSE 100: developments in 84/11 1667
ECLIPSE 100: developments in 84/9 1668
ECLIPSE 200: developments in 99A 1671
ECLIPSE 200: developments in 98A 1673
ECLIPSE 300: developments in 99A 1711
ECLIPSE 300: developments in 98A 1723
ECLIPSE 300: developments in 97A 1727
ECLIPSE 300: developments in 96A 1732
ECLIPSE 500: developments in 99A 1737

Appendix H - Bibliography 1741

Appendix I - Index 1751

List of Figures

Figure 16.1 ..... A simple, dual porosity, dual permeability system. 235
Figure 16.2 ..... Physical view of the matrix and fracture cells for the block to block connection. 236
Figure 16.3 ..... Schematic view of the contacts between the upper and lower cells of the two porosity systems. 237
Figure 16.4 ..... A typical block of matrix material containing oil and water 239
Figure 16.5 ..... The discretized matrix material block within a fracture cell. 243
Figure 16.6 ..... A representative matrix block in a fracture potential gradient 244
Figure 16.7 ..... Estimating the potential differences to the surrounding cells 245
Figure 16.8 ..... A simple reservoir grid used to simulate a gas/oil transfer function 251
Figure 16.9 ..... Initial fractional volume of water below the water contact 256
Figure 16.10 ..... Initial fractional volume of gas above the gas contact 257
Figure 16.11 ..... Integrated capillary pressure 260
Figure 16.12 ..... Nested matrix sub-cells in linear geometry 263
Figure 16.13 ..... Nested matrix sub-cells in cylindrical and spherical geometry 264
Figure 21.1 ..... A sequence of restarts that together make a full field run 314
Figure 23.1 ..... The most common instability using IMPES 349
Figure 24.1 ..... Sales Gas Production Rate and Delivery Capacity 379
Figure 25.1 ..... Optimum Lift Gas Injection Rate 386
Figure 25.2 ..... Incremental gradient calculation 390
Figure 32.1 ..... Order for writing records 474
Figure 34.1 ..... A typical pair of relative permeability curves for a non-wetting phase 519
Figure 34.2 ..... A typical pair of relative permeability curves for a wetting phase 523
Figure 34.3 ..... The water capillary pressure case 526
Figure 34.4 ..... The gas capillary pressure case 528
Figure 35.1 ..... A typical drainage process followed by an imbibition process 531
Figure 35.2 ..... Schematic diagram showing a typical secondary drainage curve 533
Figure 35.3 ..... Two-phase and three-phase relative permeability curves 535
Figure 35.4 ..... Subsequent drainage relative permeability 536
Figure 38.1 ..... Example shows 6 MR jobs running through a system that can handle 3 jobs at a time 562
Figure 39.1 ..... Single column radial grid refinement 564
Figure 39.2 ..... Single column radial grid refinement with NTHETA=1 565
Figure 39.3 ..... Radial refinement in a four column box 566
Figure 39.4 ..... Radial refinement in a four cell box with NTHETA=4 566
Figure 39.5 ..... Cartesian grid refinement 568
Figure 39.6 ..... A composite refinement 572
Figure 39.7 ..... The numbering convention used by RADFIN for connecting radial LGRs to host cells 586
Figure 39.8 ..... The numbering conventions used by RADFIN4 for connecting radial LGRs to host cells 586
Figure 39.9 ..... Well positioning in automatically refined grids 596
Figure 40.1 ..... The two-point upstream projection technique 617
Figure 43.1 ..... Multi porosity example having three pore systems. 629
Figure 43.2 ..... Structure of the matrix discretization and nomenclature 630
Figure 43.3 ..... Triple porosity connectivity 631
Figure 44.1 ..... A multi-lateral, multi-segment well 637
Figure 44.2 ..... Well segments 638
Figure 44.3 ..... Allocating connection flows to segments 638
Figure 44.4 ..... Hydrostatic head components 639
Figure 44.5 ..... Simple looped multi-segmented well 642
Figure 44.6 ..... Schematic of a multi-segmented well with an unpacked annulus 644
Figure 44.7 ..... Segment diagram for a well with a down-hole water separator 658
Figure 44.8 ..... Heat conduction between a well and the formation 663
Figure 44.9 ..... Heat conduction between the tubing and the casing 664
Figure 44.10 ..... Segment node positions with one segment per grid block connection 665
Figure 44.11 ..... Thermal contact lengths 668
Figure 45.1 ..... Common wellhead manifold and pipeline 681
Figure 45.2 ..... Well THP versus Group Flow Rate for Wellstream and Pipeline 682

Figure 45.3 ..... An example network 682
Figure 45.4 ..... The network structure 693
Figure 45.5 ..... The group control hierarchy 694
Figure 46.1 ..... Splitting each five-point transmissibility into two half-transmissibilities 708
Figure 46.2 ..... Deriving the nine-point transmissibilities using diagonal coupling between cells 708
Figure 49.1 ..... Schematic representation of gradient based optimization 728
Figure 49.2 ..... An Example of constrained optimization. 729
Figure 49.3 ..... Line search 735
Figure 53.1 ..... Calculating the initial contact corrected potential 800
Figure 55.1 ..... Three coupled reservoirs; one is the master 812
Figure 55.2 ..... Three reservoirs coupled by a “dummy reservoir” master 813
Figure 57.1 ..... Figure showing a simple river network illustrating the river reach structure 845
Figure 58.1 ..... Rock Compaction: Hysteresis option 860
Figure 58.2 ..... Rock Compaction: Boberg Hysteresis option 860
Figure 58.3 ..... Initial multipliers 861
Figure 59.1 ..... The default 3-phase oil relative permeability model assumed by ECLIPSE 880
Figure 59.2 ..... Calculating the minimum residual oil saturation from the water saturation 881
Figure 59.3 ..... Calculating the minimum residual oil saturation from the gas saturation 881
Figure 59.4 ..... Calculating the minimum residual oil saturation for drainage and imbibition with hysteresis 882
Figure 59.5 ..... Ternary diagram showing the linear interpolation used to calculate S_{omn} and S_{omx} 885
Figure 59.6 ..... Initial saturations for each zone 898
Figure 59.7 ..... Equilibration process based on the depth of the cell 898
Figure 59.8 ..... Ternary diagram showing mobile fluid end-points 900
Figure 59.9 ..... The effect of gas injection 901
Figure 60.1 ..... Two-point scaling 910
Figure 60.2 ..... Three-point scaling 910
Figure 60.3 ..... Water saturation versus depth for a typical grid cell that intersects the oil-water transition zone 913
Figure 60.4 ..... Modeling a depth variation of critical saturation 915
Figure 63.1 ..... The structure of the Jacobian matrix for a simple 3*2*3 problem. 939
Figure 63.2 ..... Two color (red-blue) split suitable for a 4-processors parallel run 951
Figure 63.3 ..... The sub-grids from Figure 63.2 with amalgamation of LGRs and base grid 951
Figure 63.4 ..... The sub-grids from Figure 63.2 with no amalgamation of LGRs and base grid 952
Figure 64.1 ..... The two point upstream projection technique 970
Figure 65.1 ..... Two graphs to illustrate the scaling of a rock curve 984
Figure 65.2 ..... Two graphs to illustrate the scaling of a straight line 985
Figure 66.1 ..... Calculation of the relative permeability 992
Figure 68.1 ..... Stencil for MPFA flow in 2D between cells (i, j) and (i + 1, j) 1017
Figure 68.2 ..... K-orthogonality error indicator 1019
Figure 69.1 ..... Component enthalpy defined by specific heats 1054
Figure 69.2 ..... Component enthalpy defined by gas specific heat and heat of vaporization 1054
Figure 69.3 ..... Component enthalpy defined by oil specific heat and heat of vaporization 1055
Figure 69.4 ..... Specific heat data leading to Hoil > Hgas 1056
Figure 69.5 ..... Heat of vaporization data leading to decreasing gas enthalpy 1056
Figure 69.6 ..... Component enthalpy defined by equal specific heats 1057
Figure 72.1 ..... Y- and Z- transmissibility expressions 1109
Figure 73.1 ..... Total oil production rate and contributions from above and below GOC 1128
Figure 75.1 ..... A cell with approximate hydrostatic equilibrium 1139
Figure 75.2 ..... Oil-water vertical equilibrium 1142
Figure 75.3 ..... Oil-gas vertical equilibrium 1143
Figure 75.4 ..... Gas-water vertical equilibrium 1144
Figure 75.5 ..... Three-phase vertical equilibrium 1144
Figure 75.6 ..... Three phase equilibrium with immobilized oil 1145
Figure 75.7 ..... Relative permeability behavior for a residual flow model 1147
Figure 75.8 ..... V.E Curves (SATNUM) 1148
Figure 75.9 ..... Residual flow (RESIDNUM) 1148
Figure 77.1 ..... A trunk and branch multi-lateral well 1167
Figure 77.2 ..... A bird’s foot multi-lateral well 1167
Figure 77.3 ..... Defining multi-lateral well geometry 1168
Figure 77.4 ..... A multi-lateral well with sub-branches 1169

Figure 77.5 ..... Friction effects in a horizontal well 1173
Figure 79.1 ..... A five-level hierarchy 1216

List of Tables

Table 3.1 Alkaline model 1 keywords 60
Table 3.2 RUNSPEC section 61
Table 3.3 PROPS section 61
Table 3.4 Alkaline SUMMARY output controls 61
Table 3.5 SCHEDULE section 62
Table 5.1 Aquifer RUNSPEC keywords 84
Table 5.2 Aquifer GRID keywords 84
Table 5.3 Aquifer PROPS keywords 85
Table 5.4 Aquifer SOLUTION keywords 85
Table 5.5 Analytic aquifer quantities 86
Table 5.6 Numerical aquifer quantities 87
Table 5.7 Aquifer SCHEDULE keywords 87
Table 6.1 RUNSPEC keywords 99
Table 6.2 PROPS keywords 99
Table 6.3 RPTRST/RPTSCHED keywords mnemonics 100
Table 6.4 ASPHALTENE option SUMMARY section keywords - block values 101
Table 7.1 List of available salts in the Multi-Component Brine model. 105
Table 7.2 Brine RUNSPEC keywords 113
Table 7.3 Brine GRID keywords 113
Table 7.4 Brine PROPS keywords 113
Table 7.5 Brine REGIONS keywords 114
Table 7.6 Brine SOLUTION keywords 115
Table 7.7 Brine summary quantities 115
Table 7.8 Multi-Component Brine summary quantities 116
Table 7.9 SCHEDULE keywords 117
Table 8.1 Brine sample analysis 131
Table 9.1 Components that can be used with the CO2STORE option. 136
Table 9.2 Aquas Species 138
Table 9.3 CO2 specific summary quantities 139
Table 11.1 RUNSPEC keywords 148
Table 11.2 PROPS keywords 148
Table 11.3 Chemical reaction SUMMARY keywords 148
Table 11.4 SCHEDULE keywords 149
Table 11.5 RPTRST, and RPTSCHED output controls 149
Table 11.6 Example names for components 157
Table 11.7 Example stoichiometry 157
Table 11.8 Example reaction rate constants, activation energy and reaction enthalpy values 157
Table 12.1 Properties data 169
Table 12.2 Equilibration data 170
Table 12.3 Summary data 170
Table 12.4 Solvent Properties data 170
Table 12.5 Adsorption keywords 171
Table 12.6 Equilibration data 171
Table 12.7 Initial coal solvent concentration manually 171
Table 12.8 Summary data 172
Table 12.9 Field group well keywords 172
Table 12.10 RUNSPEC keywords 173
Table 12.11 GRID keywords 173
Table 12.12 PROPS keywords 173
Table 12.13 SOLUTION keywords 174
Table 12.14 Coal bed methane SUMMARY output controls 174
Table 12.15 SCHEDULE Keywords 175
Table 12.16 RUNSPEC Keywords ECLIPSE 300 188
Table 12.17 GRID Keywords ECLIPSE 300 188

Table 12.18 PROPS Keywords for ECLIPSE 300 189
Table 12.19 SOLUTION Keywords for ECLIPSE 300 189
Table 12.20 SUMMARY data....189
Table 12.21 SCHEDULE Keywords for ECLIPSE 300 190
Table 12.22 Properties....191
Table 12.23 Optional keywords 192
Table 12.24 Coal bed methane example files....193
Table 12.25 Coal bed methane example files....193
Table 14.1 Solution states 203
Table 14.2 Mnemonic values 203
Table 14.3 Possible mnemonics 206
Table 15.1 RUNSPEC keywords 212
Table 15.2 GRID section keywords 212
Table 15.3 EDIT section keywords 213
Table 15.4 PROPS keywords 213
Table 15.5 SCHEDULE keywords 214
Table 16.1 Dual porosity keywords....249
Table 17.1 Keywords used to specify environmental tracer data....274
Table 17.2 RUNSPEC keywords 276
Table 17.3 PROPS keywords 276
Table 17.4 REGIONS keywords 277
Table 17.5 SOLUTION keywords 277
Table 17.6 Environmental tracer SUMMARY output controls....277
Table 17.7 SCHEDULE keywords 279
Table 18.1 The coefficients, m1 and m2, variation with the equation of state 288
Table 18.2 The dependence of and on the equation of state 289
Table 19.1 Reservoir EOS keywords....298
Table 19.2 Surface EOS keywords....300
Table 20.1 Files produced by ECLIPSE for a simulation run....303
Table 21.1 RUNSPEC keywords 317
Table 21.2 GRID keywords....317
Table 21.3 Options keywords 318
Table 22.1 Gas mobility keywords 326
Table 22.2 Gas mobility reduction keywords 326
Table 22.3 Output controls....327
Table 22.4 RUNSPEC keywords 328
Table 22.5 GRID keywords....328
Table 22.6 PROPS keywords 328
Table 22.7 Foam SUMMARY output keywords 329
Table 22.8 RPTSOL output controls 330
Table 22.9 SCHEDULE section 330
Table 22.10 RPTSCHED output controls....331
Table 24.1 Useful gas field operations SUMMARY output controls....373
Table 24.2 RUNSPEC section 380
Table 24.3 Gas field operations SUMMARY output controls 380
Table 24.4 Simulator performance....381
Table 24.5 RUNSPEC section 382
Table 25.1 Gas lift optimization SUMMARY output controls....396
Table 25.2 SCHEDULE section 396
Table 26.1 Keywords to obtain reports of recovered NGLs and gas 399
Table 29.1 Gas Calorific Value Control SUMMARY output 428
Table 29.2 Gas calorific value control SUMMARY output controls....430
Table 29.3 SCHEDULE section 430
Table 30.1 RUNSPEC section 436
Table 30.2 SOLUTION section 436
Table 30.3 RUNSPEC section 444
Table 30.4 GRID section....444
Table 30.5 PROPS section 444
Table 30.6 REGIONS section 445

Table 30.7 SOLUTION section 445
Table 30.8 SOLUTION Mnemonics 446
Table 30.9 SCHEDULE section 447
Table 30.10 SCHEDULE Mnemonics 448
Table 31.1 RUNSPEC section 454
Table 31.2 PROPS section 454
Table 31.3 SOLUTION section 454
Table 31.4 SUMMARY section 455
Table 31.5 SCHEDULE section 455
Table 32.1 RUNSPEC section 487
Table 32.2 GRID section 487
Table 32.3 PROPS section 487
Table 32.4 REGIONS section 488
Table 32.5 HMxxxxxx keyword 488
Table 32.6 SOLUTION section 489
Table 32.7 SCHEDULE section 489
Table 32.8 GRID section 490
Table 32.9 EDIT section 490
Table 32.10 PROPS section 491
Table 32.11 HMPROPS options 491
Table 32.12 SOLUTION section 492
Table 32.13 SUMMARY section 492
Table 32.14 Miscellaneous keywords 493
Table 38.1 Feature names, keywords that invoke the functionality and the license key name 559
Table 39.1 GRID Section keywords 587
Table 39.2 RUNSPEC section 597
Table 39.3 GRID section 597
Table 39.4 EDIT section LGR keywords 598
Table 39.5 PROPS section LGR keywords 598
Table 39.6 REGIONS section LGR keywords 599
Table 39.7 SOLUTION section LGR keywords 599
Table 39.8 SCHEDULE section LGR keywords 599
Table 39.9 The Global LGR keywords 600
Table 42.1 RUNSPEC keywords 624
Table 42.2 PROPS keywords 624
Table 42.3 Solid properties that can be output to the summary file 624
Table 42.4 Solid properties that can be output with RPTRST, RPTSCHED and RPTSOL 624
Table 43.1 Multi porosity example files 633
Table 44.1 The relationship between the dimensionless pipe diameter and the critical Kutateladze number 651
Table 44.2 RUNSPEC section 674
Table 44.3 PROPS section 674
Table 44.4 Multi-segment wells SUMMARY output controls 675
Table 44.5 Multi-segment wells SUMMARY output controls for secondary wells 677
Table 44.6 Multi-segment wells SUMMARY output controls for groups including secondary well contributions 678
Table 44.7 Multi-segment wells SUMMARY output controls for field including secondary well contributions 679
Table 44.8 SCHEDULE section 679
Table 45.1 SCHEDULE section keywords 689
Table 45.2 RUNSPEC section 704
Table 45.3 Network SUMMARY output controls (ECLIPSE 100) 704
Table 45.4 Network SUMMARY output controls (ECLIPSE 300) 705
Table 45.5 SCHEDULE section 705
Table 47.1 Unit conversion factors C1 and C2 712
Table 47.2 Unit-dependent constants for the Forchheimer parameter-dependent skin-factor 715
Table 48.1 Units of conversion, Herschel-Bulkley fluids 725
Table 48.2 Usage 726
Table 49.1 RUNSPEC section 745
Table 49.2 SCHEDULE section 745
Table 49.3 OPTIMIZE section 745
Table 49.4 Miscellaneous keywords 746

Table 50.1 The division of cells between processors using the PSPLITX keyword. 754
Table 50.2 RUNSPEC section 760
Table 50.3 GRID section 760
Table 50.4 SOLUTION section 761
Table 51.1 RUNSPEC section 774
Table 51.2 REGIONS section 774
Table 51.3 PROPS section 774
Table 51.4 SOLUTION/SUMMARY section 775
Table 51.5 RPTRST options 775
Table 51.6 GRADRESV options 775
Table 52.1 RUNSPEC section 785
Table 52.2 GRID section 785
Table 52.3 PROPS section 785
Table 52.4 REGIONS section 786
Table 52.5 SOLUTION section 786
Table 52.6 Polymer flood SUMMARY output controls 787
Table 52.7 SCHEDULE section 788
Table 54.1 Mnemonics for reporting recovery mechanism SUMMARY data 809
Table 55.1 Reservoir coupling SUMMARY output controls 833
Table 55.2 SCHEDULE section 833
Table 57.1 RUNSPEC keywords 846
Table 57.2 PROPS keywords 846
Table 57.3 SOLUTION keywords 846
Table 57.4 SUMMARY keywords 847
Table 57.5 Flow rate and total water keywords 847
Table 57.6 SCHEDULE keywords 848
Table 57.7 RFT, PLT and river data output using WRFTPLT keyword 848
Table 57.8 RPTSCHED keyword output 849
Table 59.1 Family (i) 870
Table 59.2 Family (ii) 870
Table 59.3 End-Point Scaling Constraints 893
Table 59.4 Saturation functions 895
Table 60.1 Saturation table end points 905
Table 61.1 Shale gas example files 924
Table 62.1 RUNSPEC section 926
Table 62.2 GRID section 926
Table 62.3 PROPS section 926
Table 62.4 SOLUTION section 927
Table 62.5 Solid properties that can be output to the summary file 927
Table 62.6 Solid production data 927
Table 62.7 Solid properties that can be output with RPTRST, RPTSCHED and RPTSOL 928
Table 64.1 RUNSPEC keywords for solvent and Miscible models 967
Table 64.2 PVT data 967
Table 64.3 Relative permeability data 968
Table 64.4 Transition keywords 968
Table 64.5 RUNSPEC section 972
Table 64.6 PROPS section 972
Table 64.7 REGIONS section 973
Table 64.8 SOLUTION section 973
Table 64.9 Solvent SUMMARY output controls 973
Table 64.10 SCHEDULE section 974
Table 66.1 SURFACTANT keywords 995
Table 66.2 RUNSPEC section 997
Table 66.3 GRID section 997
Table 66.4 PROPS section 997
Table 66.5 REGIONS section 998
Table 66.6 SOLUTION section 998
Table 66.7 Surfactant SUMMARY output controls 998
Table 66.8 SCHEDULE section 999

Table 67.1 Default values used in calculating K-values 1012
Table 67.2 Controlling summary output for temperature option 1013
Table 67.3 Controlling summary output for temperature option 1013
Table 69.1 RUNSPEC keywords 1025
Table 69.2 GRID keywords 1026
Table 69.3 EDIT keywords 1026
Table 69.4 Rock-fluid properties 1027
Table 69.5 Hydrocarbon properties 1027
Table 69.6 Water properties 1028
Table 69.7 REGIONS keywords 1029
Table 69.8 SOLUTION keywords 1029
Table 69.9 Field, Group and Well energy rates 1029
Table 69.10 Miscellaneous quantities 1030
Table 69.11 Multi-segment wells 1030
Table 69.12 Inter-region flow quantities 1031
Table 69.13 Block quantities 1031
Table 69.14 Simulator performance quantities 1032
Table 69.15 Analytic aquifer quantities 1032
Table 69.16 SCHEDULE keywords 1032
Table 69.17 RPTRST, RPTSOL and RPTSCHED output controls 1033
Table 69.18 Other keywords 1034
Table 69.19 K-value coefficients (Field units) 1048
Table 69.20 K-value coefficients (Metric units) 1049
Table 69.21 Calculated specific heats in metric units at Ts = 20 deg C 1057
Table 69.22 Calculated specific heats in field units at Ts = 20 deg C (68 deg F) 1058
Table 69.23 Summary of convergence criteria and time stepping criteria 1082
Table 69.24 Thermal example files 1088
Table 74.1 A table of the units used for four conventions 1133
Table 74.2 Constants used in the four unit conventions 1134
Table 74.3 Some useful conversion factors 1136
Table 76.1 Water RUNSPEC keywords 1156
Table 76.2 Water RUNSPEC keywords 1156
Table 76.3 Water SOLUTION keywords 1156
Table 76.4 Water SCHEDULE keywords 1157
Table 76.5 Water component values 1159
Table 76.6 Multi-component water example files 1160
Table 77.1 RUNSPEC section 1178
Table 77.2 Wellbore friction SUMMARY output controls 1178
Table 77.3 SCHEDULE section 1178
Table A.1 Minimum data requirements 1235
Table B.1 RPTPROPS output controls 1248
Table C.1 RPTGRID output controls 1252
Table D.1 RPTRST output controls 1258
Table E.1 RPTSCHED output controls 1264
Table F.1 RPTSOL output controls 1276
Table G.1 RUNSPEC keywords 1305
Table G.2 GRID keywords 1305
Table G.3 EDIT keywords 1305
Table G.4 PROPS keywords 1306
Table G.5 REGIONS keywords 1306
Table G.6 SOLUTION keywords 1307
Table G.7 SUMMARY keywords 1307
Table G.8 SCHEDULE Keywords 1310
Table G.9 RUNSPEC keywords 1311
Table G.10 GRID keywords 1311
Table G.11 EDIT keywords 1311
Table G.12 PROPS keywords 1312
Table G.13 REGIONS keywords 1312
Table G.14 SOLUTION section keywords 1313

Table G.15 SUMMARY keywords 1313
Table G.16 SCHEDULE keyword 1315
Table G.17 Altered RUNSPEC keywords 1316
Table G.18 Altered GRID keywords 1317
Table G.19 Altered PROPS keywords 1317
Table G.20 Altered SOLUTION keywords 1317
Table G.21 Altered SCHEDULE keywords 1318
Table G.22 Altered OPTIONS 1320
Table G.23 Altered RUNSPEC keywords in ECLIPSE 300 1322
Table G.24 Altered GRID keywords in ECLIPSE 300 1323
Table G.25 Altered PROPS keywords in ECLIPSE 300 1323
Table G.26 Altered SOLUTION keywords in ECLIPSE 300 1324
Table G.27 Altered SUMMARY mnemonics 1325
Table G.28 Altered SCHEDULE keywords in ECLIPSE 300 1325
Table G.29 Altered OPTIMIZE keywords in ECLIPSE 300 1327
Table G.30 Altered OPTIONS3 switches 1328
Table G.31 RUNSPEC keywords 1353
Table G.32 GRID keywords 1353
Table G.33 PROPS keywords 1354
Table G.34 SOLUTION keywords 1354
Table G.35 SUMMARY keywords 1355
Table G.36 SCHEDULE Keywords 1356
Table G.37 RUNSPEC keywords 1357
Table G.38 GRID keywords 1357
Table G.39 PROPS keywords 1357
Table G.40 SOLUTION section keywords 1359
Table G.41 SUMMARY keywords 1359
Table G.42 SCHEDULE keyword 1363
Table G.43 Altered RUNSPEC keywords 1364
Table G.44 Altered GRID keywords 1364
Table G.45 Altered EDIT keywords 1365
Table G.46 Altered PROPS keywords 1366
Table G.47 Altered REGIONS keywords 1366
Table G.48 Altered SOLUTION keywords 1367
Table G.49 Altered SUMMARY keywords 1368
Table G.50 Altered SCHEDULE keywords 1368
Table G.51 Altered OPTIONS 1370
Table G.52 Altered DEBUG switches 1371
Table G.53 Altered RUNSPEC keywords in ECLIPSE 300 1371
Table G.54 Altered GRID keywords in ECLIPSE 300 1372
Table G.55 Altered EDIT keywords in ECLIPSE 300 1373
Table G.56 Altered PROPS keywords in ECLIPSE 300 1373
Table G.57 Altered REGIONS keywords in ECLIPSE 300 1374
Table G.58 Altered SOLUTION keywords in ECLIPSE 300 1375
Table G.59 Altered SUMMARY mnemonics 1376
Table G.60 Altered SCHEDULE keywords in ECLIPSE 300 1377
Table G.61 Altered OPTIONS3 switches 1380
Table G.62 Altered OPTIMIZE keywords 1383

Introduction

Chapter 1

Overview

The ECLIPSE simulator suite

The ECLIPSE simulator suite consists of two separate simulators: ECLIPSE 100 specializing in black oil modeling, and ECLIPSE 300 specializing in compositional modeling.

ECLIPSE 100 is a fully-implicit, three phase, three dimensional, general purpose black oil simulator with gas condensate options. ECLIPSE 300 is a compositional simulator with cubic equation of state, pressure dependent K-value and black oil fluid treatments. ECLIPSE 300 can be run in fully implicit, IMPES and adaptive implicit (AIM) modes.

Both programs are written in FORTRAN and operate on any computer with an ANSI-standard FORTRAN90 compiler and with sufficient memory.

For large simulations the simulators can be run in parallel mode. The Parallel option is based on a distributed memory architecture implemented using MPI (message passing interface).

About this manual

This manual contains the technical descriptions of the principal features in both simulators. (The data file structure and the data requirements for each keyword are described separately in the "ECLIPSE Reference Manual".) Some features are available in both ECLIPSE 100 and ECLIPSE 300, while others are available in only one of the simulators. A ‘flag table’ at the start of each chapter indicates which of the simulators contain the corresponding feature. The flag table also indicates whether the feature is a Special Option that must be purchased separately (see "ECLIPSE Options" on page 38).

In chapters that are relevant to both simulators, some paragraphs or sections may apply to only one of the simulators. These are distinguished by **margin notes** such as that containing the words *ECLIPSE 100* as shown below.

ECLIPSE 100

Example of margin note.

If you are reading this manual online (for example, the PDF file), you may click on

- A cross reference (for example, "Local Grid Refinement and Coarsening" on page 563). This takes you to the relevant chapter or section of the manual (or of another manual).
- A hyperlink (for example, the `WCONPROD` keyword). This takes you to the relevant keyword description in the Reference Manual.

ECLIPSE features

Free format input

Input data for ECLIPSE is prepared in free format using a keyword system. Any standard editor may be used to prepare the input file. Alternatively Petrel or ECLIPSE Office may be used to prepare data interactively through panels, and submit runs. Its help facility contains most of the ECLIPSE Reference Manual.

Phase and Fluid options

ECLIPSE 100

ECLIPSE 100 can be used to simulate 1, 2 or 3 phase systems. Two phase options (oil/water, oil/gas, gas/water) are solved as two component systems saving both computer storage and computer time. In addition to gas dissolving in oil (variable bubble point pressure or gas/oil ratio), ECLIPSE 100 may also be used to model oil vaporizing in gas (variable dew point pressure or oil/gas ratio).

ECLIPSE 300

ECLIPSE 300 can simulate using a cubic equation of state or pressure dependent K-values, as well as the black oil model. Four equations of state are available, implemented through Martin's generalized equation [Ref. 10]. The Redlich-Kwong, Soave-Redlich-Kwong, Peng-Robinson and Zudkevitch-Joffe cases of this equation may be used. A volume shift parameter may also be used with any equation. Different sets of equation of state parameters may be used for the reservoir and surface facilities. K-values may be supplied in table form or generated internally. See "Equations of State" on page 287 for further information.

ECLIPSE 300

In `GASWAT` mode the solution of hydrocarbon components in the water phase is permitted. See "The GASWAT Option" on page 405 for further information.

Geometry options

Both corner-point and conventional block-center geometry options are available in ECLIPSE. Radial and Cartesian block-center options are available in 1, 2 or 3 dimensions. A 3D radial option completes the circle allowing flow to take place across the 0/360 degree interface (keyword `COORDSYS`).

The ECLIPSE corner-point geometry option is unique and allows extremely complex geometries to be constructed to give a faithful representation of the reservoir geology. PETREL can be used to prepare corner-point data for ECLIPSE. PETREL may also be used to display the grid in a variety of ways. For example, in a large 3D study, you may request a number of XZ cross-sections to be displayed simultaneously. Transparent areal overlays of the grid are useful for precise checking against geological maps.

Corner-point geometry is especially useful for highly faulted reservoirs. The grid may be distorted areally to fit along fault lines and displaced vertically to model complex scissor faults. The resulting non-neighbor grid transmissibilities are computed automatically by ECLIPSE. Flow across displacement faults is handled efficiently by the ECLIPSE solution procedures.

See "Transmissibility Calculations" on page 1103 for further information.

Fully implicit technology (Black Oil)

ECLIPSE uses the fully-implicit method to provide stability over long time steps. Care is taken to ensure that the non-linear fully-implicit equations are solved precisely by reducing all residuals to very fine tolerances. Material balance errors (residual sums) are extremely small. Newton's method is used to solve the non-linear equations. The Jacobian matrix is fully expanded in all variables to ensure quadratic (fast) convergence. Various special methods are used to accelerate convergence in highly non-linear problems. The linear equations arising at each Newton iteration are solved simultaneously by Nested Factorization accelerated by Orthomin.

Traditionally, fully-implicit techniques are reserved for small coning studies, where many cell pore volumes of fluid may pass through small cells near the well bore during a single time-step. IMPES and semi-implicit techniques cannot be used on such problems unless the time steps are reduced to impractically small values. When the fully-implicit method is used the resultant linear equations are strongly coupled and can only be solved efficiently by simultaneous methods and not by simple sequential methods. Small problems of this type can be solved efficiently by direct methods such as D4 Gaussian elimination. Iterative methods, required for large problems, such as SIP and LSOR, typically fail to converge. Thus most simulators cannot apply fully implicit methods to large problems. In ECLIPSE, these restrictions are removed by Nested Factorization which solves large problems efficiently and reliably.

See "Formulation of the Equations" on page 341 for further information.

Adaptive Implicit and IMPES (Compositional)

ECLIPSE 300

In a compositional model where the number of components and hence equations to be solved is greater than say 5 or 6, the cost of using a fully implicit technique can become prohibitive in both memory and CPU time. In ECLIPSE 300 this problem is tackled by using a adaptive implicit scheme, making cells implicit only where necessary.

In ECLIPSE 300 adaptive implicit, fully implicit or IMPES solution techniques may be selected. For larger compositional studies, it is likely that AIM will be the normal choice. The IMPSAT method may be used if large capillary pressures are present. Reservoir molar densities of each component are used as solution variables, along with the pressures. These yield a system that can be converted into a pressure equation and a set of conservation equations.

See "Formulation of the Equations" on page 341 for further information.

Nested factorization

The linear equations arising at each Newton iteration are solved using Nested Factorization accelerated by Orthomin. This is the fastest iterative technique yet devised for the solution of large sets of sparse linear equations. The technique conserves material precisely at each iteration. Material balance errors can therefore arise only as a result of non-linearities. Off band matrix elements arising from non-neighbor connections are included in the factorization procedure and are therefore handled with extreme efficiency. Computing costs increase only marginally faster than the number of active cells in the reservoir ($N^{5/4}$) in contrast with direct methods (N^3). Nested Factorization is therefore particularly well suited to the solution of large problems. Two and three phase problems are solved simultaneously. Further details of the method may be found in "Solution of the Linear Equations" on page 937.

At each iteration, a new 'search direction' (approximate solution) is computed by Nested Factorization. The Orthomin procedure, which minimizes the sum of the squares of the residuals (errors), requires each new search direction to be orthogonalized to the previous search directions. Thus previous search directions must be stored, using up valuable computing memory. Fortunately it is usually sufficient to store only the last few search directions. The number of search directions is called `NSTACK` and you may set this in the `RUNSPEC` data section.

The solver is highly vectorized due to the use of active cell addressing. For example, all the Orthomin calculations on the stack are completely vectorized. Matrix multiplications are vectorized using the vector run technique. However, the tridiagonal inversion at the heart of Nested Factorization, is inherently recursive and cannot be vectorized.

Non-neighbor connections

In conventional grid systems each cell has only one neighboring cell in each direction (two in each dimension). Thus in 2D grids, each cell can have up to four neighbors, and in 3D grids each cell can have up to six neighbors. Flow only takes place between neighboring cells.

In ECLIPSE, it is possible to connect selected pairs of non-neighboring cells, allowing fluid to flow directly from one cell of the pair to the other. Typical applications of non-neighbor connections are:

1. Completing the circle in 3D radial studies.
2. Displacement faults allowing flow between different layers across the fault.
3. Local grid refinement allowing fine definition near wells, including the ability to insert radial grids within a cartesian system.

ECLIPSE generates the non-neighbor connections arising from the first two applications automatically, and calculate their transmissibilities. If you are using the Local Grid Refinement option (see "Local grid refinement" on page 39), ECLIPSE also handles the non-neighbor connections arising from the third application automatically.

Non-neighbor connections give rise to off-band elements in the Jacobian matrix and these are included in the Nested Factorization procedure used to solve the linear equations.

Run-time dimensioning

All of the internal arrays in ECLIPSE are dimensioned at run time to minimize the use of computer memory. For example, each element of the Jacobian is a 2×2 matrix in 2 phase runs and a 3×3 matrix in three phase runs. A two dimensional problem gives rise to a Jacobian matrix with 5 bands, while a 3 dimensional problem gives rise to a Jacobian matrix with 7 bands. Clearly it would be inefficient to dimension the internal arrays to cover the worst possible case. Most of the data required for ECLIPSE to compute the size of its internal arrays is provided in the `RUNSPEC` data section. Some additional information (relating to automatically generated non-neighbor connections, inactive cells, Vertical Equilibrium options etc.) is deduced from data provided in the `GRID` section. Space is saved by not storing unnecessary data for inactive cells.

Vertical equilibrium

ECLIPSE 100

ECLIPSE 100 contains a unique Vertical Equilibrium option for modeling segregated flow in three dimensions. The option may be used with either block-center or corner-point geometry. With corner-point geometry, it takes account of the shape and orientation of each distorted grid block. A mixing parameter enables you to specify the degree of fluid segregation. At one extreme, total fluid segregation is assumed, while at the other extreme the fluids are assumed to be totally dispersed throughout each grid block. The VE facility takes account of hysteresis effects arising, for example, when oil invades the water zone and retreats, leaving a residual critical oil saturation.

It is also possible, using the Compressed Vertical Equilibrium option, to make the assumption that the entire reservoir section is in vertical equilibrium. This enables ECLIPSE 100 to collapse each column of grid cells into a single cell for calculational purposes. The variation in properties with depth are accounted for by using tables of properties for the compressed single cells.

See "Vertical Equilibrium" on page 1139 for further information.

Dual porosity

The ECLIPSE Dual Porosity/Permeability option is suitable for simulating highly fractured reservoirs. Each grid block is represented by a matrix cell and a fracture cell. In the Dual Porosity option, flow through the reservoir takes place through the fracture cells only. Flow also takes place between each matrix cell and its corresponding fracture cell. Flow does not take place between neighboring matrix cells. In the Dual Porosity/ Permeability option flow may also take place between neighboring matrix cells.

The Dual Porosity option can also model the effects of gravity imbibition/drainage, molecular diffusion between the matrix and fracture cells.

ECLIPSE 100

In addition ECLIPSE 100 can also model the viscous displacement of fluid to and from the matrix.

See "Dual Porosity" on page 233 for further information.

PVT and rock data

ECLIPSE honors pressure and saturation function data precisely as it is specified by the user. It does not follow the practice, common in the industry, of smoothing the data by interpolating to a fixed number of equally spaced saturation intervals.

Different table numbers can be used for different parts of the reservoir. Thus, for example, separate saturation tables can be entered for each type of rock.

Directional relative permeabilities

Relative permeability curves may be specified separately for flow in the horizontal and vertical directions. You have the option of specifying 3 (X,Y,Z) or 6 (+X,-X,+Y,-Y,+Z,-Z) relative permeability curves for each saturation function region, rather than the usual single curve. This additional degree of freedom enables you to improve the physics of a simulation study in a relatively simple manner. For example, most reservoir cells have small vertical dimensions and large areal dimensions, and horizontal flow is more realistically modeled if the horizontal relative permeability is small below the Buckley-Leverett saturation. Directional relative permeabilities are also required for the ECLIPSE pseudo option that may be used on a three dimensional coarse grid.

Saturation table scaling

The connate, critical and maximum saturation end points of the relative permeability and capillary pressure curves may be input separately for each grid cell within the reservoir or, alternatively, as a series of depth tables for regions within the grid. The scaling option allows you to specify relative permeability and capillary pressure data which are functions of a normalized saturation. In addition, it allows the modeling of reservoirs where depth variations occur in the initial critical or connate fluid saturations. You may specify that the saturation table scaling is to be isotropic, distinguish between coordinate lines (X,Y,Z) or distinguish between coordinate directions (+X,-X,+Y,-Y,+Z,-Z).

The saturation table scaling facility may be used to equilibrate a reservoir model with the correct initial quantities of mobile fluid-in-place by means of a special correction applied at the fluid contacts.

See "Saturation Table Scaling" on page 905 for further information.

Hysteresis

Both relative permeability and capillary pressure hysteresis effects can be modeled.

ECLIPSE 100

In ECLIPSE 100 there is a choice of two models for relative permeability hysteresis in the non-wetting phase: Carlson's model and Killough's model. There is also an option of using Killough's wetting phase hysteresis model.

ECLIPSE 300

In ECLIPSE 300 there is a choice of Killough's model and Jargon's model in the non-wetting phase.

ECLIPSE 100

An alternative three phase hysteresis model is available, which is applicable in cases where all three phases are present together, for example in a water alternating gas (WAG) flood (see "Hysteresis in WAG Floods" on page 529).

See "Hysteresis" on page 517 for further information.

Rock compaction

The Rock Compaction option models the collapse of pore channels as the fluid pressure decreases. The process may be reversible, irreversible or hysteretic as required. The facility is particularly useful in dual porosity runs.

ECLIPSE 100

In addition in ECLIPSE 100 the Water Induced Compaction facility enables the mechanical effect of water on some rock types (for example chalk) to be taken into account.

See "Rock Compaction" on page 859 for further information.

Tracer tracking

The Tracer Tracking option is a general facility to determine the movement of ‘marked’ fluid elements during a simulation run. It may be used, for example, to differentiate the movement of water injected by different wells and initial aquifer water, or to predict the variation in salinity or concentrations of other chemical species.

ECLIPSE 100

Tracers may also be defined to exist in a hydrocarbon phase, and used to model the exchange of the fluid elements between the free and solution states whenever mass transfer takes place in the associated hydrocarbon phase. The Tracer option is therefore useful in predicting the movement of the initial free and/or solution gas. Initial sulphur content may be defined as a function of depth and tracked with the movement of oil.

ECLIPSE 100

An option to control the effects of numerical diffusion on tracer interfaces is available.

See "Tracer Tracking" on page 1117 for further information

API tracking

ECLIPSE 100

Unlike the ‘passive’ tracer tracking options described above, API Tracking is ‘active’. It takes full account of the mixing of oils with totally different PVT properties. This contrasts with the conventional practice of assuming that the oil magically assumes the formation volume factor, viscosity, density and bubble point of the cell into which it flows. Initial API may be defined as a function of depth in each equilibration region or independently for each cell. A fourth equation is solved at each time step to update APIs.

See "API Tracking" on page 69 for further information.

Brine tracking

The brine tracking facility is an ‘active’ tracer in the water phase. The water properties are tabulated as a function of the salt concentration. Hence if injected water has a different salinity from the original reservoir water, the different water density and viscosity can be accounted for.

See "Brine Tracking" on page 103 for further information.

Temperature model

ECLIPSE 100

The Temperature model is designed to model the cooling of the reservoir when cold water is injected into a relatively hot reservoir. An energy balance equation is solved after the main flow equations at each time step, to determine the temperature in each grid block. Both the oil and water viscosities are modified as a function of temperature for the subsequent time step.

See "Temperature option" on page 1009 for further information.

Miscibility

ECLIPSE 100

ECLIPSE 100 has a three-component Miscible Flood model. This is intended for the simulation of miscible gas injection processes. The model assumes that the reservoir fluids consist of three components: reservoir oil (stock tank oil + solution gas), injection gas (solvent) and water. The reservoir oil and solvent gas components are assumed to be miscible in all proportions. Physical dispersion of the miscible components is treated using the Todd- Longstaff technique by means of a mixing parameter which is specified on a region basis. The solvent gas gravity can differ from that of the solution gas, and the model also allows the oil/solvent screening effects at high water saturations to be studied. A special numerical diffusion control facility is available with the miscible option, and is particularly useful in studies using large values of the mixing parameter. See "Miscible Flood Modeling" on page 609 for further information.

ECLIPSE 300

In a compositional simulation miscibility is predicted naturally as the surface tension between the two hydrocarbon phases drops to zero. However the simulator still needs to model the changing relative permeability of the miscible fluid. ECLIPSE 300 contains a comprehensive miscibility model. See "Surface Tension Effects" on page 983 for further information.

Individual well controls

ECLIPSE has a comprehensive set of options for controlling individual wells. Producers can operate at a specified value of the oil rate, water rate, liquid rate, gas rate, linearly combined rate, reservoir fluid voidage rate, bottom hole pressure and tubing head pressure. The engineer supplies a target value for one of these quantities, and limiting values for any of the remaining quantities. The well operates at its specified target as long as none of the limits are violated. If a limit is going to be violated, the well automatically changes its mode of control to keep operating within its allowed limits. Efficiency factors can be applied to take account of regular downtime. The flow rates and pressures are calculated for the well under normal flowing conditions, but the cumulative flows are reduced according to the efficiency factor. Injection wells have a similar set of controls. Target or limiting values can be supplied for the injection rate at surface conditions, the injection rate at reservoir conditions, the bottom hole pressure and the tubing head pressure.

To aid the process of history matching, production wells can be placed under an additional type of control. Their observed oil, water and gas rates are input, and the well can be made to produce at the equivalent liquid or reservoir fluid volume rate. Thus the rate of pressure decline should be approximately correct even when the water cut and gas-oil ratio are not fully matched. Both the observed and calculated flow rates can be written to the `SUMMARY` file for graphical comparison.

Production wells can also be subject to an additional class of ‘economic’ constraints. A producer is automatically shut-in or re-completed if its oil, gas or liquid rate falls below an economic limit. If its water cut, gas-oil ratio or water-gas ratio exceeds the specified upper limit, the well is worked over or shut-in. The rate of doing workovers can be limited by rig availability. Water cut, gas-oil ratio and water-gas ratio limits, and minimum economic rate limits, can also be applied to individual connections, which is closed if their limits are violated. In addition there is an option to limit the water cut, gas-oil ratio or water-gas ratio from a well by successively reducing its production rate each time the limit is violated. In ECLIPSE 100 wells subject to a minimum tubing head pressure limit can have their productivity boosted by automatically switching on artificial lift when their oil or liquid flow rates fall below a specified limit, subject

to constraints on the group or field capacity for artificial lift. Individual wells can be ‘flagged’ to make the simulation run stop if they are shut-in for any reason, or another well can be opened automatically. The run can also be made to stop if all the producers in the field, or in one or more nominated groups, are shut-in. Wells that are shut-in can be tested periodically to see if they can flow economically again.

See "Well controls and limits" on page 1210 for further information.

Group and field production controls

The simulator contains facilities to control the collective behavior of groups of wells, or the field as a whole. The overall production rate of oil, water, gas, liquid or reservoir fluid (that is voidage) from one or more groups can be made to meet a specified target by one of two methods: prioritization or guide rate control.

With prioritization, wells are ranked in order of priority, and are turned on in decreasing order of priority until a group rate limit is exceeded. The well that exceeds the group rate limit has its rate cut back to meet the limit. The remaining lower-priority wells in the group are kept closed until they are needed. Well priorities can either be set by you, or calculated at specified intervals from a general formula with user-defined coefficients.

With guide rate control, the group’s target rate is apportioned between all its producers in proportion to either their production potentials or specified ‘guide rates’, but ensuring that no well violates its individual flow rate or pressure limits. When the group no longer has enough potential to meet its production target, the production rate will decline. However, the simulator can postpone the decline by automatically opening new wells from a drilling queue whenever extra potential is required, subject to constraints on the drilling rate and the maximum number of open wells allowed in each group. In ECLIPSE 100 additional actions for maintaining the production rate (reperforating, retubing, switching on lift and changing to a lower THP limit) can be performed through ‘group production rules’.

Upper limits can also be applied to groups’ production rates of oil, water, gas, liquid and reservoir fluid. There is a choice of actions that are taken whenever one of these limits is violated, ranging from a work-over of the worst-offending well to the application of group control to hold the group’s production rate at the limiting value. In addition, a set of ‘economic constraints’ equivalent to those described above for individual wells can be applied to the overall production behavior of groups.

The same set of controls and limits can also be applied to the overall field production. Under guide rate control, a target production rate for the field can either be apportioned between the groups in proportion to specified group guide rates, or apportioned directly between the wells in proportion to their production potentials or guide rates. The field production rate can be controlled simultaneously with the production from one or more of its constituent groups. The identity of the controlled phase (oil, water, gas, liquid or reservoir fluid) need not be the same in each case. For example, the field can produce oil at a required rate while one of the groups is producing at its maximum gas handling capacity. Thus the field produces at its target rate while each group honors its own set of constraints, and while each well honors its individual pressure and rate constraints.

See "Group and field control facilities" on page 1216 for further information.

Multi-level grouping hierarchy

Groups can be arranged into a multi-level tree structure of any required depth, by declaring some groups to be ‘sons’ of higher level groups. All group controls and limits can be applied at any level (including the field, which can be regarded as a group at level zero at the top of the tree).

Group injection controls

Groups, and the field, can be given targets and limits to control their injection rates. The engineer can set target or limiting values for any of the following quantities:

- Surface injection rate of a particular phase.
- Total reservoir volume injection rate of all phases.
- Re-injection fraction of a particular phase.
- Total voidage replacement fraction by all injected phases.

Thus for example a group can be made to re-inject a set proportion of its produced gas, and to inject water to make up an overall reservoir volume injection target or voidage replacement fraction.

A group injection target is apportioned between the wells in proportion to either their injection potentials or specified guide rates, subject to the wells’ individual flow and pressure limits. A target for a high level group (or the field) can optionally be apportioned among its subordinate groups in proportion to their group guide rates, which can be set directly or calculated to equal their production voidage rates or voidage replacement needs. If a group (or the field) has insufficient injection potential to meet its target, the drilling queue is scanned for a suitable injector to open.

ECLIPSE 300

In compositional cases an injected gas stream may be specified explicitly, or be defined by the production from a well or group. It may also be a mixture of several such sources. The injection rate may be limited by the availability of gas from the specified source, although advance gas and make-up gas streams can be defined to augment the supply.

See "Group injection control options" on page 1221 for further information.

Sales gas production control

The Sales Gas Production Control facility provides a means of controlling the export of associated gas from an oil field, which may at the same time have controls on the oil production rate. The sales gas production rate of a group or the field is defined as its total gas production rate, minus its total gas injection rate, minus the gas consumption rate of the group and any subordinate groups, plus the rate of gas import (if any) to the group or its subordinates. Individual gas consumption and import rates can be set for each group.

The sales gas production rate is controlled by re-injecting the surplus gas not required for sale. Sales gas control can be applied independently of any other production controls on the group or field, provided that there is enough injection capacity to inject the surplus gas. The group or field is automatically placed under gas re-injection control, and its target re-injection fraction is determined dynamically at each time step to inject the surplus gas.

See "Sales gas production control" on page 1226 for further information.

Crossflow and co-mingling in wells

When a well is completed in more than one grid block, the flow rate from each grid block is proportional to the product of three quantities:

1 The transmissibility between the grid block and the well bore.

2 The mobility of the phase in the grid block at the perforations.

3 The pressure drawdown between the grid block and the well bore.

ECLIPSE takes full account of all three quantities when apportioning a well's target flow rate between its various grid block connections. This is especially important in cases where a well is completed in two or more layers of a reservoir with poor vertical communication, where the drawdown at each layer can be substantially different.

In certain situations the drawdown in one layer may have the opposite sign to the drawdown in the other completed layers. The simulator will then allow crossflow to take place between the reservoir layers through the well bore. A material balance constraint is observed within the well bore at formation level, so that the mixture of phases flowing out of the well bore corresponds to the average phase mixture entering the well bore. ECLIPSE can handle crossflow in any type of well operating under any mode of control, including the case when the well is plugged off above the formation while the completions remain open to the reservoir.

The crossflow facility can be turned off if the engineer does not wish this to occur.

See "Well Inflow Performance" on page 1187 for further information.

Highly deviated and horizontal wells

There is no restriction on the location of the completions within each well. A highly deviated well can be completed in a number of grid blocks that do not all belong to the same vertical column. In fact a well can be completed in several grid blocks within the same layer, enabling the simulator to model wells completed in horizontally-adjacent grid blocks. This feature is also useful in 3D radial coning studies where the well connects with all the innermost, wedge-shaped, blocks in any given layer. Multi-segment Wells (see "Multi-segment wells" on page 43) or the ECLIPSE 100 Wellbore Friction option (see "Wellbore friction" on page 40) may be used to investigate the effects of friction pressure losses along the well's perforated length, which could be important for horizontal wells.

Special facilities for gas wells

ECLIPSE has two facilities for improving the accuracy of modeling gas inflow. Firstly, the non-Darcy component of the pressure drawdown resulting from turbulence near the well bore is modeled by a flow-dependent contribution to the skin factor. The simulator calculates the turbulence skin factor and the gas inflow rate in each completion by a method that ensures that they are fully consistent with each other. The calculation is performed fully implicitly, to maintain stability over long time steps. These precautions are especially important when the pressure drop due to turbulence is a significant proportion of the total drawdown.

ECLIPSE 100

The second facility is the option to use a special inflow equation. ECLIPSE 100 offers the choice of using the Russell Goodrich equation or the gas pseudo-pressure equation in place of the conventional inflow performance relationship for free gas flow. Both equations make allowance for the variation in the viscosity and the density of the gas over the pressure drop between the grid block and the well bore. For the Russell Goodrich equation, these quantities are determined at the average pressure between the grid block and the well bore, instead of at the pressure of the grid block. For the pseudo-pressure equation, these quantities are replaced by an average value integrated between the grid block and well bore pressures.

In gas condensate simulations the Generalized Pseudo-Pressure method of Whitson and Fevang can be employed. Here, the total hydrocarbon mobility is integrated between the grid block pressure and the well bore pressure, taking account of changes in relative permeability caused by condensate dropout.

See "Special inflow equations" on page 1195 for further information.

Fine grid equilibration

The initial pressures and saturations in the reservoir grid blocks can be calculated by the 'equilibration' facility. Each phase in the reservoir is given a hydrostatic pressure gradient, dependent on its local density. The saturations are determined from the local pressure differences between the phases, using the capillary pressure tables.

A common practice is to set the fluid saturations within each grid block equal to the local saturations at the center of the block. However, this procedure can produce significant errors in the Fluids-in-Place estimation when a large grid block contains a fluid contact or part of a transition zone. ECLIPSE has an option to improve the accuracy of the Fluids-in-Place calculation in this situation. The fluid saturations are determined at a number of levels within the block, and these are averaged to obtain the overall fluid saturations within the block. The method can also take account of the shape of sloping grid blocks, by weighting the fluid saturations at each level within the block with the area of the horizontal cross-section of the block at that level.

See "Initializing the Study" on page 543 for further information.

Aquifers

Aquifers may be represented by a choice of two analytic models, a constant flux model, two constant head/pressure models and a numerical model. The analytic models are the Fetkovich aquifer and the Carter-Tracy aquifer. The constant flux model allows you to specify the flow per unit area at any time during the simulation. The constant head/pressure model allows you to specify the head or pressure at any time during the simulation. The numerical model consists of a sequence of aquifer cells connected together in one dimension. The innermost cell of the numerical aquifer sequence may be connected to any number of reservoir cells. The depth, dimensions, porosity, permeability etc. of each aquifer cell may be specified separately, giving the user complete flexibility in defining each aquifer.

See "Aquifer Modeling Facilities" on page 73 for further information.

Ancillary programs

Ancillary programs available with ECLIPSE are described below.

VFPi

VFP*i* is an interactive program for preparing, examining and editing VFP tables. It calculates pressure drops in wells and pipelines, and generates vertical flow performance tables for input into ECLIPSE. The program contains extensive graphical facilities for examining tubing data and fluid properties, and plotting the results of individual pressure traverses and calculated vertical flow performance curves. A well IPR can be superimposed on the curves to provide an estimate of the flow rate at a given set of conditions. VFP tables can be read in and examined in cross section or 3D. There are facilities for transforming the tables in a number of ways, and visually editing individual points within a table. Additional features of VFP*i* include an enthalpy equation to calculate the temperature variation of the fluid along the tubing, and a facility to match calculated pressures with measured data by tuning a family of 'fitting factors'. VFP*i* can also perform pressure drop calculations for a compositional fluid, and produce compositional vertical flow performance tables for use with ECLIPSE 300.

VFP*i* offers a choice of seven multi-phase flow correlations to calculate the pressure traverse:

1. Aziz, Govier and Fogarasi,
2. Orkiszewski,
3. Hagedorn and Brown,
4. Beggs and Brill,
5. Mukherjee and Brill,
6. Gray,
7. Petalas and Aziz.

Different correlations may be used for different sections of tubing, enabling VFP*i* to handle horizontal and undulating pipelines as well as vertical well bores and risers. The calculations can include the effects of gas lift, a down-hole pump, a gas compressor, and a choke with both critical and sub-critical two-phase flow.

GRAF

GRAF provides an easy to use, menu driven graphics capability.

Typical uses of GRAF are:

- Line graphics showing, for example, field pressure against time.
- Grid graphics showing the reservoir grid including faults.
- Dynamic solution displays showing animated color time displays of saturations or pressures etc.

Line graphics are generated from `SUMMARY` files created by ECLIPSE at each report time. Each Summary file contains data relating to field, group, well, connection, region and cell quantities recorded at intermediate time-steps. Thus the Summary files (like the `RESTART` files) may be used to monitor runs in progress.

Grid graphics are generated from a `GRID` file created by ECLIPSE at the start of a simulation run. `GRID` files contain the locations of the corners of each cell in the reservoir, and may be used to display the grid in a variety of ways.

Dynamic solution displays are generated from `RESTART` files created by ECLIPSE at prescribed times. `RESTART` files contain detailed information such as saturations and pressures for each grid cell. Their main purpose is to enable ECLIPSE to be restarted at the prescribed times. However, they may also be used in conjunction with the `GRID` file to generate color displays of saturations and pressures. You can step forwards or backwards through color sequences creating animated displays of water floods, gas coning events etc.

As well as color-filled solution displays, GRAF provides options for contours, arrow plots and transmissibility displays. Hard copy output of GRAF can be generated on a a wide range of plotters and printers.

Additional programs

Programs, which may be purchased as special additions to the basic ECLIPSE system, are described below.

ECLIPSE Options

ECLIPSE Options consists of a series of special extensions to ECLIPSE that increase the modeling capability of the program. These are:

- Polymer Flood model,
- Surface Networks,
- Flux Boundary Conditions,
- Local Grid Refinement and Coarsening,
- Wellbore Friction option,
- Gas Field Operations model,
- Solvent model,
- GI Pseudo-compositional model,
- Surfactant model,
- Gas Lift Optimization,
- Parallel options,
- Environmental Tracers,
- Coal Bed Methane option,
- Foam model,
- Reservoir Coupling,
- Gas Calorific Value Control,
- Multi-segment Wells.
- Thermal Option

The special extensions may be purchased separately as required.

Polymer flood model

ECLIPSE 100

The Polymer Flood option uses a fully implicit five-component model (oil/water/gas/polymer/brine) to allow the detailed mechanisms involved in polymer displacement processes to be studied. The model accounts for the increase in viscosity of the aqueous phase due to the addition of the polymer solution, and also for the viscosity degradation due to non-Newtonian shear effects at high velocity.

The adsorption of polymer to the rock surfaces and the resulting reduction in aqueous phase relative permeability can be treated by the model. The effects of inaccessible pore volume on the flow of the polymerized solution are also included.

You may optionally use the model to study the loss of polymer solution viscosity due to interaction with brine. A fully implicit formulation is employed to solve the coupled oil/water/gas/polymer/brine equations, to ensure a completely stable solution for all time steps.

See "The Polymer Flood Model" on page 777 for further information.

Surface networks

The Network option is designed to provide variable tubing head pressure limits to groups of wells, which vary with the groups' production rates according to a set of pipeline pressure loss relationships.

A simple application would be to model a sub-sea completion, where several wells produce into a single manifold which is connected to a separator by a flow line. The pressure in the manifold will depend on the total production rate, GOR and water cut of the wells.

For more complicated applications, a multi-level hierarchical network of pipelines can be built up. Each pipeline is given a vertical flow performance table (which can be produced by VFP*i*) to determine the pressure losses along it, in the same way that well bore pressure losses are calculated.

A network balancing calculation is performed at user-defined intervals to update the pressure at each node in the network. Separate networks can also be set up for water and gas injection, which can be useful for modeling pump or compressor performance into injection manifolds.

See "The Network Option" on page 681 for further information.

Flux boundaries

The Flux Boundary option allows runs to be performed on a small section of a field. A base run is performed for the full field study and time dependent flows across the boundary of the small section of the field are written to a flux file. Subsequent runs for the small section are performed, reading the flux file to establish boundary conditions for consistency with the full field model. This option is useful for history matching when only a few wells remain to be matched, and reduces the cost of repeated runs when only a small sector of the field remains to be resolved.

ECLIPSE 100

It may also be used in conjunction with the drainage region facility (switch 25 in the `OPTIONS` keyword) to reduce computing costs in sensitivity studies.

See "Flux Boundary Conditions" on page 309 for further information.

Local grid refinement

The Local Grid Refinement option allows enhanced grid definition near wells. The local models may be 2D radial, 3D radial or 3D cartesian. Local models may have more layers than the global model. Transmissibilities between the local models and the global model are computed automatically.

ECLIPSE 100

Greater efficiency may be achieved by solving each local model individually. In general, local models have shorter time-steps than the global model so that detailed coning effects can be resolved near troublesome wells without holding up the progress of the global field simulation.

It is also possible to solve the combined local and global system fully implicitly as a coupled system ("LGR in-place"). In ECLIPSE 300 all LGRs are solved simultaneously with the global grid.

Grid coarsening can also be used to amalgamate cells in regions of the field where accuracy is not so important.

ECLIPSE 100

An autorefinement facility may be used to refine an existing global model as the reverse process to grid coarsening.

See "Local Grid Refinement and Coarsening" on page 563 for further information.

Wellbore friction

ECLIPSE 100

The Wellbore Friction option models the effects of pressure loss due to friction in the well tubing along the perforated length and between the perforations and the bottom hole reference point of the well. The facility is primarily intended for use with horizontal wells, in which frictional pressure losses may be significant over the horizontal section of the wellbore. It is important to consider friction losses when deciding on the optimum length and diameter of a horizontal well. The friction calculation is handled fully implicitly, to retain numerical stability.

See "The Wellbore Friction Option" on page 1163 for further information.

Gas field operations model

The Gas Field Operations model is a collection of facilities designed to model gas field production constraints and operations. These comprise:

- Using a seasonal profile of monthly multipliers to the mean rate or DCQ (Daily Contracted Quantity) to set the production target,
- Adjusting the DCQ each contract year to obey a swing factor requirement,
- Control and output of sales gas production,
- Estimation of delivery capacity at each time step, allowing for the increased pressure losses through the network,
- *ECLIPSE 100* Automatic operation of compressors in the network, whenever they are needed to achieve the production target.

See "Gas Field Operations Model" on page 359 for further information.

ECLIPSE 100

Solvent model

The Solvent model is a 4-component extension of the Todd and Longstaff empirical miscible flood model. ECLIPSE solves 4 coupled equations fully implicitly for oil/water/gas/solvent. The oil/water/gas system in the absence of the solvent component reduces to the traditional black oil model. A solvent phase can be introduced into the model to miscibly displace the reservoir hydro-carbons. You can control the degree of miscibility using the Todd-Longstaff parameter, and the transition to miscibility with an input table of miscibility versus solvent saturation. Most potentially miscible fluids are only miscible at high pressure. This pressure effect can be modeled in ECLIPSE by defining a table of miscibility versus pressure.

See "The Solvent Model" on page 955 for further information.

GI pseudo-compositional model

ECLIPSE 100

The GI Pseudo-compositional model is a 4-equation model which improves the modeling of volatile oil and gas condensate systems, without moving to a fully compositional description. The GI parameter is a measure of the amount of dry gas that has contacted the original hydrocarbons. The saturated R_s and R_v values and the B_o and B_g of the oil and gas are supplied to ECLIPSE as functions of both pressure and GI. For example the saturated R_v value of the gas will reduce with GI; thus the gas is able to contain less vaporized oil. This effect overcomes one of the problems associated with black oil models, namely that of predicting over-optimistic recoveries in gas cycling studies. The data required by ECLIPSE can be generated from an equation-of-state description of the reservoir fluids using the PVT program.

See "Gi Pseudo-compositional Model" on page 449 for further information.

Surfactant model

ECLIPSE 100

The Surfactant model allows modeling of enhanced water floods where the added surfactant reduces the capillary pressure between the water and oil phases. The reduction in capillary pressure allows a greater proportion of the oil to be displaced, thus reducing the residual oil saturation. The Surfactant flood option models the reduction in the residual saturation and the associated modification of the relative permeability curves as a function of the capillary number (that is the ratio of viscous to capillary forces). The effect of the surfactant concentration on the water viscosity, and the adsorption of surfactant onto the rock, can be treated by the model.

See "The Surfactant Model" on page 989 for further information.

Gas lift optimization

The Gas Lift Optimization facility determines how much lift gas to allocate to each well in order to meet well, group or field production targets. If production targets cannot be met, it determines how to make the best use of the available lift gas resources by allocating lift gas preferentially to the wells that can make the best use of it. The facility can also be used with the Network option to optimize the lift gas distribution within a small network of a few wells, taking account of the extra pressure losses in the flowlines due to the increased flows.

See "Gas Lift Optimization" on page 385 for further information.

Parallel options

The Parallel options allow ECLIPSE to take advantage of computers with more than one processor. The facility provides an efficient parallel solution method, using a domain decomposition technique, which divides the simulation grid into domains which are solved in parallel.

The facility can run on both shared memory and distributed memory machines.

See "Parallel option" on page 749 for further information.

Environmental tracer

ECLIPSE 100

The Environmental Tracer option is a set of extensions of the tracer tracking facility that can be used to model environmental problems. For example, the transport of a radioactive contaminant in ground water flow. At present the extra physical models are tracer adsorption and tracer decay. Up to fifty tracers can be tracked independently, each with its own adsorption and/or decay model.

See "Environmental Tracers" on page 269 for further information.

Coal bed methane

ECLIPSE 100

The Coal Bed Methane option allows the modeling of gas production (methane) from coal seams. Gas present in the coal matrix diffuses into a fracture system and can be produced using traditional oil field technology. The option models the diffusion process and the flow of water and gas within the fracture system. It is also possible to model enhanced recovery mechanisms using a third conservation equation to represent an injected gas, typically CO_2.

See "The Coal Bed Methane model" on page 161 for further information.

Foam model

ECLIPSE 100

The Foam model is a simple model of the foam injection process, which can be used to screen fields for possible foam injection projects. The foam is modeled as a tracer in the gas phase, which reduces the gas mobility. The foam quality can degrade due to adsorption and decay, the decay rate being a function of both the water and oil saturations. In addition, the effectiveness of the gas mobility reduction is a function of pressure and the shear rate.

See "The Foam Model" on page 319 for further information.

Reservoir coupling

ECLIPSE 100

The Reservoir Coupling facility provides a means of coupling a number of separate ECLIPSE 100 simulation models to allow for constraints on their overall production and injection rates, and optionally the sharing of a common surface network. An oil or gas field may comprise a number of isolated reservoir units that are coupled only by global production and injection constraints. These may have been studied separately (for example, for history matching) with their own simulation models. The Reservoir Coupling facility provides a convenient way of combining these models to obey global production and injection constraints, with minimal change to their input data. The simulation models are run as separate processes, under the control of a master process which dictates their production and injection constraints at each time step. Communication between the processes is handled by the PVM message passing system, which allows the individual simulation models to be run in parallel on different UNIX computers.

See "Reservoir Coupling" on page 811 for further information.

Gas calorific value control

ECLIPSE 100

The Gas Calorific Value Control option controls the mean calorific value of gas produced from the field, while simultaneously controlling the gas production rate. The calorific value of the produced gas can either be specified independently for each well, or equated to the concentration of a gas phase tracer. A target mean calorific value can be set for groups that are operating at a rate target under guide rate group control (keyword `GCONPROD`). ECLIPSE attempts to keep the group operating at both targets simultaneously by dynamically adjusting the well guide rates to give an appropriate mixture of high and low calorific value gas. The range of well calorific values must of course span the group's target value. An additional option is the ability to specify a target energy rate (that is flow rate multiplied by calorific value) for the group. The facility is compatible with the Gas Field Operations model, in that a group or field target for either gas production rate or energy production rate can be set according to the DCQ and the swing or profile factors.

See "Gas Calorific Value Control" on page 417 for further information.

Multi-segment wells

The Multi-segment Well model provides a detailed description of fluid flow in the well bore. The facility is specifically designed for horizontal and multi-lateral wells, and to model 'smart' wells containing various types of flow control device, although it can of course be used to provide a more detailed analysis of fluid flow in standard vertical and deviated wells. A down-hole separator model is also available. The detailed description of the fluid flowing conditions within the well is obtained by dividing the well bore (and any lateral branches) into a number of segments. Each segment has its own set of four independent variables to describe the local fluid pressure, the flow rate and the flowing fractions of water and gas. The pressure drop along the well bore is calculated from the local hydrostatic, friction and acceleration pressure gradients. The pressure drop can be calculated from a choice of two built-in multi-phase flow models: a homogeneous flow model where all the phases flow with the same velocity, or a 'drift flux' model which allows slip between the phases. Alternatively, the pressure drop may optionally be derived from pre-calculated VFP tables, which can potentially offer greater accuracy and provide the ability to model chokes and other flow control devices. The multi-segment well model also enables complex crossflow regimes to be modeled more accurately, as the re-injected fluid mixture can vary with position along the well bore.

See "Multi-segment Wells" on page 635 for further information.

Thermal option

ECLIPSE 300

The Thermal option turns ECLIPSE 300 into a thermal simulator that models the flow of steam as well as the flow of hot or cold water in an oil reservoir.

The thermal options runs in fully implicit mode. The production system model is the multi-level hierarchy in ECLIPSE 300, with an implicit cross-flow well model. The thermal model will run in either a dead-oil or a live-oil mode. In the dead-oil model, hydrocarbon components are only present in the oil phase, and the gas phase consists only of steam. The live-oil model allows hydrocarbon components in both the oil and the gas phase, and water components in both the water and the gas phase.

See "Thermal Option" on page 1023 for further information.

Open-ECLIPSE

Open-ECLIPSE enables ECLIPSE 100 and ECLIPSE 300 to be controlled by, and communicate interactively with, other applications. Although ECLIPSE is primarily used in a batch stand-alone mode, controlled solely by the contents of the input data file, there are many situations in which it would be advantageous to have ECLIPSE controlled by another software application. Examples include the requirement to have ECLIPSE tightly coupled to a surface gathering system model or a specialized production optimization application. It could also be useful to couple ECLIPSE to an interactive controller, to allow you to view the current status of the simulation and make well management decisions during the course of the run.

Open-ECLIPSE was developed to address these requirements. It consists of a set of subroutines in the code that communicate with an external application. Activation of Open-ECLIPSE puts ECLIPSE into 'listening mode', awaiting commands from a suitably configured controlling program. The controlling program may set well and group operating constraints, interrogate well and group flows, control the advancement of the simulation run and control the output of reports and `RESTART` files. Communication between ECLIPSE and the controlling program is handled by PVM (Parallel Virtual Machine), which must be present on your system.

Those wishing to develop their own applications to control ECLIPSE may obtain the Open-ECLIPSE Developer's Kit. This consists of full documentation of the communications dialog, a copy of the message passing systems annex with which to link their application, and a simple demonstration controller for testing the Open-ECLIPSE interface.

FloGrid

FloGrid is an interactive 3D product that constructs fluid flow simulation grids and properties.

FloGrid imports geological data in most popular map and 3D geological model formats, including the POSC Rescue format. Simulation grids can be exported in ECLIPSE and other formats. FloGrid provides full 3D visualization of the data - wells, maps, geocellular models, faults and simulation grids.

FloGrid supports the generation of both structured and unstructured grids. Structured (Cartesian) grids are optimized to fit to selected faults and the boundary, with remaining grid nodes placed to minimize orthogonality errors. Faults that are not explicitly gridded may be automatically zig-zagged. For 3D geological models, simulation layering can, optionally, be determined using algebraic and flow based techniques to automatically identify simulation layers that best capture the flow characteristics of the fine scale models.

Unstructured PEBI and tetrahedral (PetraGrid) grids can be tailored to honor all boundaries, wells and faults. They support rectangular and radial refinements around wells, and general refinements consisting of a rectangular, triangular or hexagonal grid within a specified polygonal region.

FloGrid also provides a suite of algebraic and flow based single phase upscaling tools that calculate appropriate simulation block properties from the fine scale geologic or stochastic property grids.

PVTi

PVT*i* is an interactive Equation of State (EoS) package used for the analysis of laboratory measurements performed to determine the phase behavior of reservoir fluids. The quality of the laboratory measurements can be tested through material balance checks. Laboratory experiments can be simulated using a variety of cubic EoS, and any discrepancies between calculated and measured data can be minimized by regression of one or more EoS parameters. The EoS model can then be used to generate data suitable for use in ECLIPSE 100 or 300 and VFP*i*.

SCAL

SCAL is a tool to help you effectively use laboratory derived relative permeability and capillary pressure measurements in reservoir simulation. The program has facilities to read in laboratory data, perform quality control such as curve smoothing, group data according to lithological parameters and end-point values, transform the laboratory data into rock curves suitable for ECLIPSE, and automatically assign these curves to grid cells on the basis of rules which you set up (for example, as a function of porosity / permeability / lithological parameters). The

output consists of a series of INCLUDE files, for both the PROPS and REGIONS sections. The program has facilities for 3D visualization of simulation grids and 3-phase relative permeabilities, and for experimenting with ECLIPSE end point scaling options. The behavior of SCAL can be extended or modified by the use of user-programmable command scripts to, for example, implement a company confidential algorithm.

Schedule

Schedule imports production data from a variety of common sources including PA, OilField Manager and Finder, and generates the corresponding ECLIPSE production control keywords. Production data can also be extracted from existing ECLIPSE models. The program has advanced graphic display features, which simplify the editing, validating and averaging of production data. Time steps on which rates are averaged are based on a user defined combination of calendar periods and reservoir events such as well completions, shut-ins and simulations.

All the main categories of production data necessary for simulation can be handled by Schedule. These typically take the form of well deviation surveys, historical production and injection volumes and completion data. A key feature of Schedule is the capability to generate accurate and representative COMPDAT keywords with time-varying connection factors calculated from perforation data specified in terms of measured depths and formations. Corrections are made for deviated wells, partial penetrations and multiple completions within a single cell.

Schedule has comprehensive facilities for creating prediction run controls for ECLIPSE. Controls can be set for wells, groups and the field. Wells can be created by defining IJK locations or by digitizing in the 3D Viewer, where simulation results can be used as a backdrop to aid placement.

Schedule can also prepare data for input to the Multi-segment Well model (see "Multi-segment wells" on page 43). It can read data describing casing, liner and tubing characteristics and locations of chokes, packers and inflow control valves. This information can be used, with perforation data, to generate WELSEGS and COMPSEGS/COMPSEGL keywords describing the multi-segment well.

FloViz

FloViz is an interactive 3D visualization system for the display and analysis of reservoir simulation results. It replaces RTView as the product of choice for reservoir visualization. FloViz has a simple to operate graphical user interface and provides faster 3D interaction and animation of simulation results. It runs on both PCs and workstations.

FloViz incorporates most of the RTView functionality with some key new features. The program allows multiple views (which can be tailored), through the use of slave viewers, of the reservoir with independent slicing, property, and time animation. Model rotation can be shared across the views. FloViz can display both structured and unstructured grids, and, also, streamlines. The program contains a host of new options to allow easier visual interpretation of the grid and its results.

Weltest 100

Weltest 100 is an interactive well test analysis program containing a wide range of analytic reservoir models. It provides a variety of diagnostic plots to help you select the reservoir model, which best fits the data. Straight lines can be fitted interactively to determine “first estimates” of the model parameters, which are passed on to the optimizer. The optimizer then determines the values of the reservoir parameters that give the best match between the chosen model and the observed data. The program also contains facilities for type curve analysis, gas well test analysis and report generation.

Weltest 200

Weltest 200 is a powerful integrated numerical well test analysis package which combines the features of Weltest 100 and the ECLIPSE reservoir simulator. It offers a wide range of features including the ability to build reservoir geometry interactively, specify formation properties by layer and region and carry out extended well tests.

The numerical solutions produced by Weltest 200 have been verified against analytical models. Wellbore storage, no-flow and constant pressure boundaries, dual-porosity systems, hydraulically fractured wells, partially penetrating wells and multi-layer systems can all be handled effectively and swiftly by the program. Weltest 200 also models multi-phase flow effects and offers a tractable method for analyzing tests below the bubble point.

By packaging the numerical capabilities of ECLIPSE inside Weltest 200 an unlimited range of models for well test analysis are available. In contrast, more traditional well testing analysis packages compute the time-dependent pressure response using analytical methods, which restricts the application of well testing to a few ‘standard’ models.

Well testing specialists will appreciate the time-saving brought about by the Weltest 200 automatic grid-building features. An innovative type of gridding technique is employed, based on perpendicular bisection (PEBI) or Voronoi grids. The grid is made up of polygonal cells offering flexibility and ease of refinement.

ECLIPSE Office

ECLIPSE Office provides an interactive environment for the creation and modification of Black Oil and Compositional models, the submission and control of runs, the analysis of results and report generation. Data sets may be created using a PEBI gridding module, correlations for PVT and SCAL data, keyword panels, or input from other pre-processors. Panels exist for all ECLIPSE keywords. Tools are provided for the management of cases within a project. Run management is provided for submitting, monitoring and controlling simulator execution. New interactive graphics are used to view results.

SimOpt

SimOpt is aimed at reducing the man time taken during the history matching phase of a reservoir engineering study. It provides a means for obtaining sensitivities of ECLIPSE 100 and ECLIPSE 300 simulation results with respect to user-defined parameters, which may be specified on a regional basis. Parameters currently available include pore volumes, transmissibilities, fault transmissibilities, aquifer strength and dual porosity sigma factors. Sensitivities can be viewed directly, or used by a regression algorithm to improve the quality of a history match.

PlanOpt

PlanOpt is an interactive tool to assist you in choosing the potential locations of vertical production wells during development. PlanOpt applies pre-defined screening criteria to each column of grid blocks in a 3D simulation model to determine which could sensibly accommodate a production well. A vertical well is completed in each column of cells that satisfies the criteria. A simulation run is then used to rank the wells and the least effective wells are eliminated. This cycle is repeated until the optimum well locations have been identified.

FrontSim

FrontSim is a three-dimensional, two-phase fluid flow simulator based on a state-of-the-art streamline concept. FrontSim can perform simulations on large and complex reservoir models several orders of magnitude faster than standard finite difference simulators, and can achieve this without grid orientation effects or numerical dispersion.

The streamline concept is based on an IMPES (Implicit Pressure Explicit Saturation) solution. First the pressure is solved with an implicit numerical method and then the saturation equation is solved using an explicit method. The pressure is used to compute a velocity field which is, in turn, used to compute streamlines. The saturation equations are solved on the streamlines using a front tracking method.

Licenses

For more information see "Licenses" on page 557.

Adjoint Gradient Screening

Chapter 2

Introduction

This chapter describes the adjoint gradient screening facilities within ECLIPSE 300.

For analysis of simulation models, methods that use gradient information can be prohibitively expensive if the gradients are derived numerically, from multiple runs of the 'black box'.

ECLIPSE production profile gradients can be calculated much more efficiently from within the simulator itself (see "The Gradient Option" on page 463). The SimOpt interactive application successfully makes use of these ECLIPSE gradients in an external gradient-based optimization loop, as an aid in the history matching phase of a study. With the classic type of gradient calculation used in the SimOpt workflow, there is still a practical limit on the number of parameters that can be used, as the simulation run time increases as the number of parameters increases.

An alternative gradient calculation approach, the adjoint method, allows cost-effective calculation of sensitivities of an objective function with respect to a large number of static grid parameters relevant to the flow equations, such as porosity or permeability of every active grid block. Here, an objective function is defined as a dynamic flow-related parameter (component rate, flowing bottom-hole pressure) of a particular well. The adjoint gradient method for static parameters is available in ECLIPSE 300.

The adjoint method for gradient calculation is also used in the optimization of well control settings to maximize production from a simulation model (see “Optimization” on page 727). The adjoint production optimization method cannot be used at the same time as this adjoint gradient screening option.

The sections that follow give an overview of the keywords necessary to use this option, describe the output from the option, list the restrictions and limitation of the option, and give a description of the formulation.

Keywords

Activating the option

In the RUNSPEC section, the AJGRADNT keyword must be specified to activate the option.

Specifying the objective functions

In the SCHEDULE section, the required objective functions are specified using the AJGWELLS keyword, which consists of a list of well names and mnemonics for dynamic parameters.

If an adjoint gradient screening run terminates early (for instance, because all the wells are shut, or because an economic limit has been violated) and this keyword has not been specified by the early exit time, then no adjoint gradients will be generated as none will have been requested by that stage of the simulation. In order to avoid this problem, the AJGWELLS keyword should be specified right at the beginning of the SCHEDULE section.

Note This advice on the positioning of the AJGWELLS keyword is different to the advice given pre-2007.1. Previously it was recommended that this keyword should be placed just before the END keyword).

Specifying the parameters

Also in the SCHEDULE section, the required static properties are specified using the AJGPARAM keyword. The sensitivities will be calculated with respect to these properties on each simulation grid cell.

If an adjoint gradient screening run terminates early (for instance, because all the wells are shut, or because an economic limit has been violated) and this keyword has not been specified by the early exit time, then no adjoint gradients will be generated as none will have been requested by that stage of the simulation. In order to avoid this problem, the AJGPARAM keyword should be specified right at the beginning of the SCHEDULE section.

Example

```
SCHEDULE

AJGPARAM
'PERMXY' 'PORO' /

AJGWELLS
'P1'  'WBHP'  /
'P2'  'WBHP'  /
'I1'  'WBHP'  /
/

RPTRST
BASIC=2  /

WELSPECS
'P1'    'GRUP'   1  1    3000.0   'OIL'    1*   'NO'  /
'P2'    'GRUP'   1  5    3000.0   'OIL'    1*   'NO'  /
'I1'    'GRUP'   5  1    3000.0   'WATER'  1*   'NO'  /
/

COMPDAT
'P1'    1  1   1  1   'OPEN'  0    1*    0.2   1*  0.0    1*   'Z' /
'P2'    1  5   1  1   'OPEN'  0    1*    0.2   1*  0.0    1*   'Z' /
'I1'    5  1   1  1   'OPEN'  0    1*    0.2   1*  0.0    1*   'Z' /
/

WCONPROD
'P*'   'OPEN'  'LRAT'   3*       100    1*     1*  /
/

WCONINJE
  'I1'   'WAT'  'OPEN'  'RATE'   400    1*     1*
/

TSTEP
50*5.0 /

END
```

Output

In the SCHEDULE section, the RPTRST keyword must be specified with BASIC=2 or BASIC=6 so that the restart information is saved for every report/time step.

At the end of the simulation, the sensitivities are calculated for all objective functions and at every report step. These are stored as real arrays in an additional section at the end of each restart file. The sensitivities can be plotted by loading restart data into FloViz or other visualization packages, and then displaying AJGFN* restart properties.

Note The sensitivities output to the restart file will be encoded unless the UNCODHMD keyword is present in the RUNSPEC section. The encoded sensitivities are not decoded by ECLIPSE Office or FloViz and so are not meaningful when visualized in those programs.

Example

For the example in "Keywords" on page 50, which has 25 simulation grid cells, the section added to the restart file for report step 49 will be as shown:,

```
'AJGRADNT'           0 'MESS'
'AJGNAMES'          24 'CHAR'
'AJGFN  1' 'P1    ' 'WBHP    ' 'PERMXY  ' 'AJGFN  2' 'P2    ' 'WBHP   '
'PERMXY  ' 'AJGFN  3' 'I1    ' 'WBHP   ' 'PERMXY  ' 'AJGFN  4' 'P1    '
'WBHP    ' 'PORO    ' 'AJGFN  5' 'P2    ' 'WBHP    ' 'PORO ' 'AJGFN  6'
'I1      ' 'WBHP    ' 'PORO    '
'AJGPARA '           5 'INTE'
49          1           1          25           0
'AJGFN  1'          25 'REAL'
0.26733625E+00   0.28940764E+00   0.15707164E+00   0.82344688E-01
0.53730132E-02  -0.60202714E-01  -0.42874895E-01  -0.30267414E-01
-0.30209363E-01  -0.54189377E-01  -0.10426258E-01  -0.54382261E-01
-0.51501323E-01  -0.38832009E-01  -0.45974124E-01   0.63967048E-02
-0.14613148E-01  -0.23543639E-01  -0.24201822E-01  -0.31580176E-01
-0.64227316E-02  -0.27032575E-01  -0.31063568E-01  -0.24502732E-01
-0.17454501E-01
'AJGPARA '           5 'INTE'
49          2           1          25           0
'AJGFN  2'          25 'REAL'
0.12758227E+00   0.28352138E+00   0.15206486E+00   0.75946450E-01
0.50332956E-02  -0.70431128E-01  -0.43791033E-01  -0.33293936E-01
-0.32356218E-01  -0.46265345E-01   0.21185523E-01  -0.51040705E-01
-0.59991647E-01  -0.43821692E-01  -0.33047505E-01   0.98482676E-01
-0.37209673E-02  -0.28060049E-01  -0.27350528E-01  -0.15706640E-01
0.32918173E+00   0.45060366E-01   0.36429397E-02  -0.28943263E-02
-0.19188058E-02

...
Lines skipped here
...

'ENDAJG  '           0 'MESS'
```

The new records appended to the restart file are:

- AJGRADNT - the header marking the start of the section

- AJGNAMES - a character array consisting of blocks of four fields containing the following information.
  - Name of the sensitivity (internally assigned by ECLIPSE)
  - Name of the well
  - Name of the dynamic parameter on the well
  - Name of the static grid property
- AJGPARA - an integer array containing the following information for the record that appears next in the file.
  - Report step number
  - Well number
  - Parameter number
  - Number of sensitivity values that will be output
  - Reserved for future use
- AJGFN 1 etc. - a real array containing one sensitivity value for every active grid cell
- ENDAJG - the header marking the end of the adjoint gradient section.

Restrictions

This section lists the current restrictions and known limitations of the option.

- This option is only available in ECLIPSE 300.
- This option cannot be used with the PARALLEL option.
- This option is not compatible with IMPES, IMPSAT or AIM solution methods. The FULLIMP solution method must be used.
- This option cannot be used with reservoir optimization (see the RESOPT keyword).
- The current version requires multiple restart files format (see the MULTOUT keyword). Also, restart information must be saved at every report step (BASIC=2 or BASIC=6 options in RPTRST keyword).
- In order for visualization of adjoint gradient properties to be available in post-processing programs such as ECLIPSE Office Results Viewer or FloViz, the current version of ECLIPSE 300 requires that a restart index file (which has an RSSPEC extension) is **not** written out. For this reason, the presence of the AJGRADNT keyword automatically turns on NORSSPEC.
- The option does not allow adjoint gradient calculations for wells under reservoir voidage rate or THP control, or wells under active group control (see the WCONPROD keyword).
- Simulation grids with non-neighbor connections cannot be used (faults, numerical aquifers, local refinement or coarsening, dual-porosity). It is advisable to explicitly use NONNC keyword in RUNSPEC section.
- Any well used for objective function definition must be on a surface rate or bottom-hole pressure control during the entire simulation run. When a requested dynamic parameter is used as well control, or the well is shut or stopped, the derivative is set to zero.

Formulation

ECLIPSE forces the fully implicit residual in each simulator grid block, R, to be zero (see"Formulation of the Equations" on page 341) by solving for solution variables $X(t)$ at discrete points in time, $t_1, t_2,..., t_N$, so that for a dynamic well parameter, the following vector notation can be used.

$$W[X(t)] \rightarrow \underline{W} = \begin{bmatrix} W[X(t_1)] \\ W[X(t_2)] \\ \dots \\ W[X(t_N)] \end{bmatrix} \equiv \begin{bmatrix} W_1 \\ W_2 \\ \dots \\ W_N \end{bmatrix}$$

If we assume that a parameter p, which is normally a constant in the flow equations, can be varied, then $W = W(p,X(t,p))$ which has the following differential

$$dW = \left(\frac{\partial W}{\partial p}\right) \cdot dp + \left(\frac{\partial W}{\partial X}\right) \cdot dX \qquad \text{[EQ 2.1]}$$

Remembering that the flow equations still have to be honoured:

$$\begin{aligned} dR_1(X_1, X_0, p) &= 0 \\ dR_2(X_2, X_1, p) &= 0 \\ dR_3(X_3, X_2, p) &= 0 \\ \dots \quad & \quad \dots \\ dR_N(X_N, X_{N-1}, p) &= 0 \end{aligned} \qquad \text{[EQ 2.2]}$$

then we can eliminate the term containing solution variables in [EQ 2.1]. To do this, we apply Lagrange multipliers, Ψ , for each equation in [EQ 2.2] and insert them into [EQ 2.1] to give the following:

$$\frac{dW}{dp} = \left\{\left(\frac{\partial W}{\partial p}\right) + \Psi \cdot \left(\frac{\partial R}{\partial p}\right)\right\} + \left\{\left(\frac{\partial W}{\partial X}\right) + \Psi \cdot \left(\frac{\partial R}{\partial X}\right)\right\} \cdot \frac{dX}{dp} \qquad \text{[EQ 2.3]}$$

We are free to choose the Lagrange multipliers so that the right hand term in [EQ 2.3] is equal to zero - this choice has the benefit that we do not need to calculate the (expensive) dX/dp term.

These Lagrange multipliers can be represented as a lower triangular matrix:

$$\Psi = \begin{bmatrix} \Psi_{1,1} & 0 & 0 & 0 & \dots \\ \Psi_{2,1} & \Psi_{2,2} & 0 & 0 & \dots \\ \Psi_{3,1} & \Psi_{3,2} & \Psi_{3,3} & 0 & \dots \\ \dots & \dots & \dots & \dots & \dots \\ \Psi_{N,1} & \dots & \Psi_{N,N-2} & \Psi_{N,N-1} & \Psi_{N,1N} \end{bmatrix}$$

where both row and column indices indicate the report step number.

So, at each time level, k, the Lagrange multipliers can be calculated by solving the set of "adjoint" equations, [EQ 2.4] and [EQ 2.5].

$$\frac{\partial W_k}{\partial X_k} + \Psi_{k,k} \cdot \frac{\partial R_k}{\partial X_k} = 0 \qquad \text{[EQ 2.4]}$$

$$\Psi_{k,m} \cdot \frac{\partial R_m}{\partial X_m} + \Psi_{k,m+1} \cdot \frac{\partial R_{m+1}}{\partial X_m} = 0 \qquad \text{[EQ 2.5]}$$

where $m = 1,..., k\text{-}1$.

The residual derivatives required in this system are the same as the ones already used in the non-linear iterations inside the simulator, so they are already calculated and available for this method to use.

Note With this adjoint method, the number of equations that must be solved grows quadratically with the number of report steps. However, for a given report step, we are not limited by the number of parameters as the residual derivatives required in [EQ 2.3] are relatively cheap to obtain.

The Alkaline Model

Chapter 3

Introduction

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL

Alkaline flooding requires the injection of alkaline chemicals (lye or caustic solutions, that is high pH solutions) into a reservoir that react with petroleum acids to form in-situ surfactants that help release the oil from the rock by reducing interfacial tension, changing the rock surface wettability, and spontaneous emulsification. The oil can then be moved more easily through the reservoir to production wells. Alkaline flooding is usually more efficient if the acid content of the reservoir oil is relatively high.

When used in conjunction with surfactant and polymer to perform an Alkaline-Surfactant-Polymer (ASP) flooding, the alkaline can reduce the adsorption of both surfactant and polymer on the rock surface, therefore enhancing the effectiveness of the surfactant and polymer drive.

In this release, we provide a simplified model that does not take into account the in-situ surfactant creation and the phase behavior:

- Model 1; this simplified model is targeted at providing an easy way to take into account some effects of the alkaline on an ASP flooding performance. The focus here is to provide the user with some facility to analyze the effect of the alkaline on the water-oil surface tension and adsorption reduction of surfactant and polymer.

The alkaline model 1

In this section we describe the simplified alkaline model for ASP flooding.

Alkaline conservation equation

The alkaline is assumed to exist only in the water phase a concentration in a water injector (see the WALKALIN keyword). The distribution of the injected alkaline is modeled by solving a conservation equation:

$$\frac{d}{dt}\left(\frac{VS_wC_a}{B_rB_w}\right)+\frac{d}{dt}\left(V\rho_rC_a^a\frac{1-\phi}{\phi}\right) = \sum\left[\frac{Tk_{rw}}{B_w\,\mu_{s\,\mathrm{eff}}}(\delta P_w-\rho_w gD_z)\right]C_a+Q_wC_a \qquad \text{[EQ 3.1]}$$

where

ρ_w, ρ_r denotes the water and rock density respectively

Σ denotes the sum over neighboring cells

C_a denotes the alkaline concentration

C_a^a denotes the adsorbed alkaline concentration

$\mu_{s\mathrm{eff}}$ denotes the effective viscosity of the salt

D_z is the cell center depth.

B_w, B_r is the water and rock formation volume respectively

T is the transmissibility

k_{rw} is the water relative permeability

S_w is the water saturation

V is the block pore volume

Q_w is the water production rate

P_w is the water pressure

g is the gravity acceleration

By default, the alkaline concentrations are updated at the end of a time step after the inter-block phase flows have been determined. If the alkaline effects of the flow properties are very strong, the alkaline concentrations can be solved at each Newton iteration so that the tracer concentrations are updated simultaneously with the phase equations by setting the 126th item of the OPTIONS keyword to 1. If the ECLMC keyword is found in the RUNSPEC section, the default solution is to update the alkaline concentration at each Newton iteration. The new alkaline concentrations are then used to update the flow properties for the subsequent iterations.

Treatment of adsorption

The adsorption of alkaline is assumed to be instantaneous. The isotherm adsorption can be specified as:

- either a look-up table of adsorbed alkaline as a function of alkaline concentration using the ALKADS keyword.
- or by a generic analytical adsorption model using the ADSORP keyword. This model enables the adsorption to depend on rock permeability as well.

If desorption is prevented (see the first argument of the ALKROCK keyword) then the adsorbed alkaline concentration may not decrease with time. If desorption is allowed then each grid block retraces the adsorption isotherm as the alkaline concentration falls in the cell.

Note Because this simplified model is targeted at looking at some of the effects of the alkaline on Alkaline-Surfactant-Polymer flooding (ASP) the rock mass density is taken from either the PLYROCK keyword or the SURFROCK keyword.

Alkaline effect on water-oil surface tension

We model the effect of alkaline on the water-oil surface tension as a combined effect with surfactant by modifying the water-oil surface tension as follows:

$$\sigma_{wo} = \sigma_{wo}(C_{\text{surf}})A_{st}(C_{\text{alk}}) \qquad \text{[EQ 3.2]}$$

where

$\sigma_{wo}(C_{\text{surf}})$ is the surface tension at surfactant concentration and zero alkaline concentration (SURFST keyword).

$A_{st}(C_{alk})$ is the surface tension multiplier at alkaline concentration (ALSURFST keyword).

Alkaline effect on surfactant/polymer adsorption

The alkaline can reduce the adsorption of both surfactant and polymer on the rock surface. This is modeled by modifying the mass of adsorbed surfactant or polymer as follows:

$$V\rho_r C^a_{s,p}\frac{1-\phi}{\phi}A_{ad}(C_{\text{alk}}) \qquad \text{[EQ 3.3]}$$

where

V is the pore volume of the cell

ϕ is the porosity

ρ_r is the mass density of the rock (see the SURFROCK/PLYROCK keyword)

$C^a_{s,p}$ is the surfactant/polymer adsorbed concentration obtained from SURFADS/PLYADS/ ADSORP keywords.

$A_{ad}(C_{alk})$ is the adsorption multiplier at alkaline concentration (see the ALSURFAD/ALPOLADS keyword).

When the Alkaline is assumed to be adsorbed irreversibly, by preventing desorption in the ALKROCK keyword, its effect on Surfactant/Polymer adsorption is assumed to be irreversible as well. Using the previous notations, the mass of adsorbed surfactant or polymer becomes:

$$V\rho_r C^a_{s,p}\frac{1-\phi}{\phi}A_{ad}(C_{\text{alk max}}) \qquad \text{[EQ 3.4]}$$

where

C_{alkmax} is the maximum alkaline concentration reached in the block, corresponding to the actual effect on Surfactant/Polymer adsorption.

Note If surfactant and polymer are used in the same run the PLYROCK keyword will superseed the SURFROCK keyword

Using the Alkaline model 1

This model is activated by specifying the ALKALINE keyword in the RUNSPEC section. The Polymer Flood Model or the Surfactant Flood Model should be active as well. The presence of at least one of the following keyword in the PROPS section is obligatory.

Table 3.1 Alkaline model 1 keywords

Keyword	Description	Notes
ALSURFST	Table of oil/water surface tension as a function of alkaline concentration	Obligatory if the Surfactant Flood Model is active.
ALSURFAD	Table of surfactant adsorption as a function of alkaline concentration	Obligatory if the SURFADS keyword is used.
ALPOLADS	Table of polymer adsorption as a function of alkaline concentration	Obligatory if the Polymer Flood Model is active.
ALKADS	Tables of adsorption functions	Optional
ALKROCK	Specifies alkaline-rock properties	Obligatory if ALKADS is used.

The concentration of the injected alkaline in a water injector is specified using the WALKALIN keyword. For the list of outputs associated of the alkaline model see the SUMMARY and SCHEDULE sections below.

The SUMMARY outputs follow the passive tracer format with a keyword concatenated with the tracer name. Here, alkaline is referred to as ALK. For example the keyword for the field total alkaline production would be FTPRALK.

Summary of keywords

RUNSPEC section

The Alkaline model 1 RUNSPEC keywords are

Table 3.2 RUNSPEC section

Keyword	Description
ALKALINE	Activates the Alkaline model.

PROPS section

The Alkaline model 1 PROPS keywords are

Table 3.3 PROPS section

Keyword	Description
ALSURFST	Tables of oil/water surface tension as a function of alkaline concentration Only if the surfactant option is activated
ALSURFAD	Tables of surfactant adsorption as a function of alkaline concentration Only if the surfactant option is activated
ALPOLADS	Table for polymer adsorption as a function of alkaline concentration Only if the polymer option is activated
ALKADS	Table for alkaline adsorption function as a function of alkaline concentration
ADSORP	Analytical adsorption isotherm (alternative to ALKADS).
ALKROCK	Alkaline-rock adsorption/desorption properties

SUMMARY section

The Alkaline model 1 SUMMARY keywords are

Table 3.4 Alkaline SUMMARY output controls

Field	Group	Well	Connection	Region	Block	Information
			CTFRALK			Flow Rate (+ or -)
FTPRALK	GTPRALK	WTPRALK				Production Rate
FTPTALK	GTPTALK	WTPTALK	CTPTALK			Production Total
FTIRALK	GTIRALK	WTIRALK				Injection Rate
FTITALK	GTITALK	WTITALK	CTITALK			Injection Total
					BTCNFALK	Concentration

Table 3.4 Alkaline SUMMARY output controls

Field	Group	Well	Connection	Region	Block	Information
					BTADSALK	Adsorption
					BTSTMALK	Surface tension multiplier
					BTPADALK	Polymer adsorption multiplier
					BTSADALK	Surfactant adsorption multiplier

SCHEDULE section

The Alkaline model 1 SCHEDULE keywords are:

Table 3.5 SCHEDULE section

Keyword	Description
RPTSCHED	Controls output from the SCHEDULE section. Mnemonic ALKALINE outputs the Alkaline concentrations and the alkaline adsorption when active.
WALKALIN	Specifies the alkaline concentration for a water injector. This concentration can be set as an UDA (User Defined Argument).

Example problem

In this example only sections where alkaline data (input/output) are used are mentioned.

RUNSPEC Example

RUNSPEC section

```
RUNSPEC
TITLE
Example Alkaline and Surfactant Case.
DIMENS
 5    5    1  /
NONNC
OIL
WATER
SURFACT
ALKALINE
FIELD

TABDIMS
 2    2   20    5    6   20 /
REGDIMS
 6    1    0    0    0    1 /
WELLDIMS
 2    1    1    2 /
START
 1 'JAN' 1983  /
NSTACK
 8 /
```

PROPS Example

PROPS section

```
PROPS
SWFN
-- Table 1 - Immiscible curves - SATNUM
    .25    .0    4.0
    .7    1.0     .0
/
-- Table 2 - Miscible curves   - SURFNUM
    .25    .0    4.0
    .99    1.0     .0
/
```

```
SOF2
-- Table 1 - Immiscible curves - SATNUM
    .3000  .0000
    .7600 1.0000
/
-- Table 2 - Miscible curves   - SURFNUM
    .0100  .0000
    .7600 1.0000
/
PVTW
 4000.0  1.0  3.03E-06  .5  0.0 /
 4000.0  1.0  3.03E-06  .5  0.0 /
PVDO
  .0     1.0     2.0
 8000.0   .92    2.0
/
-- Table 2 defaulted
/
ROCK
 4000.0         .30E-05 /
 4000.0         .30E-05 /
DENSITY
 52.0000  64.0000   .04400 /
 52.0000  64.0000   .04400 /
-- Water Viscosity vs surfactant concentration tables.
SURFVISC
-- Surfactant      Water viscosity at 4000 psi
-- concentration
--   LB/STB             CENTIPOISE
      0.0                  1.0
      0.1                  1.1
      0.2                  1.2
      0.8                  1.3  /
      0.0                  0.5
      0.1                  0.6
      0.2                  0.7
      0.8                  0.8  /
```

```
-- Surfactant Adsorption tables
SURFADS
-- Surfactant       Adsorbed saturation
-- concentration
--   LB/STB              LB/LB
     0.0                  0.00000
     0.1                  0.00005
     0.2                  0.00005
     1.0                  0.00005
/
-- Table 2 defaulted
/
-- Alkaline adsorption
ALKADS
--   Alkaline            Adsorbed
-- concentration        saturation
--   LB/STB
     0.0                  0.00000
     0.2                  0.00002
     1.0                  0.00005 /
/
-- Alkaline adsorption properties
ALKROCK
-- De-adsorption model
           1              /
/
-- Alkaline multupliers for Surfactant Adsorption
ALSURFAD
-- alkaline             adsorption
-- concentration        multiplier
--   LB/STB
     0.0                  1.00000
     0.1                  0.99995
     0.2                  0.99500
     1.0                  0.50000 /
/
```

```
-- Water/Oil surface tension vs Surfactant concentration
SURFST
-- Surfactant        Water/Oil Surface Tension
-- concentration
--   LB/STB              LB/IN
      0.0                100.0
      0.1                50.0
      0.2                10.0
      0.5                0.0  /
 /
-- Water/Oil surface tension multipliers vs alkaline concentration
ALSURFST
-- Alkaline        Water/Oil Surface Tension
-- concentration        multiplier
--   LB/STB
      0.0                1.0
      0.1                0.9
      0.2                0.9
      0.5                0.9  /
 /
-- Capillary de-saturation curve
SURFCAPD
-- LOG10(Capillary Number)     Miscibility
--                             Function
        -9                       0.0
         2                       0.0
         5                       1.0
        10                       1.0 /
/
-- Extra rock properties.
SURFROCK
-- De-adsorption model            Rock mass
-- indicator                      Density
--   1 - reversible
--   2 - irreversible              LB/RB
      1                            1000 /
      1                            1000 /
```

SUMMARY Example

SUMMARY section

```
SUMMARY
CTFRALK
 'P' 5 5  1 /
/
FTPRALK
FTPTALK
FTIRALK
FTITALK
WTPTALK
'P' /
```

```
WTITALK
'I' /
BTCNFALK
1 1 1 /
2 2 1 /
5 5 1 /
/
BTADSALK
1 1 1 /
2 2 1 /
5 5 1 /
/

FTPRSUR
WTPRSUR
'P' /
WTPTSUR
'P' /
WTITSUR
'I' /
BTCNFSUR
1 1 1 /
2 2 1 /
5 5 1 /
/
FTIPTSUR
FTIPFSUR
RTIPTSUR
1 /
RTIPFSUR
1 /
BTIPTSUR
1 1 1 /
/
FTADSUR
RTADSUR
 1 /
BTADSUR
1 1 1 /
/
BTCASUR
1 1 1 /
/
```

SCHEDULE Example

SCHEDULE section

```
SCHEDULE
RPTSCHED
  ALKALINE/
```

```
WELSPECS
'I'  'G'   1  1  4000  'WAT'  /
'P'  'G'   5  5  4000  'OIL'  /
/
COMPDAT
'I       '   1   1   1   1 'OPEN'   0  .0   1.0 /
'P       '   5   5   1   1 'OPEN'   0  .0   1.0 /
/
WCONPROD
'P' 'OPEN' 'BHP' 5* 3999.0 /
/
WCONINJ
'I' 'WAT' 'OPEN' 'RATE' 200.0 /
/
-- Start injecting the Surfactant Slug
WSURFACT
 'I'   0.8 /
 /
-- Start injecting the Alkaline
WALKALIN
 'I'   0.8 /
 /
TSTEP
  2*100
/
-- Water chase
WSURFACT
 'I'   0.0 /
 /
WALKALIN
 'I'   0.0 /
 /
TSTEP
 5*100 /
END
```

API Tracking

Chapter 4

Introduction

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL

The API Tracking facility enables ECLIPSE to model the mixing of different types of oil, having different surface densities and PVT properties.

Without the API Tracking facility, the presence of different types of oil in the reservoir could be handled with the aid of PVT region numbers. Oil in PVT region 1 would have its properties determined from PVT table number 1, and so on. However, this method cannot model the mixing of oil types. Oil flowing from region 1 into region 2 would appear to take on the properties associated with region 2.

The API Tracking facility essentially replaces the concept of PVT regions for oil. The PVT tables used for determining the oil properties are selected at each time step according to the average API of the oil in each grid block (or to be more precise, its average surface density). A mass conservation equation is solved at the end of each time step to update the oil surface density in each grid block, to model the mixing of the different oil types.

The API Tracking facility is turned on by the keyword `API` in the `RUNSPEC` section. In dead oil systems the keyword `RSCONSTT` cannot be used with API tracking, but `RSCONST` (defining a uniform R_s value over the whole field) can be used.

The PVT properties

Two or more sets of PVT tables should be supplied, each set being associated with a particular value of the API. The oil property tables are entered in the usual way (with keyword `PVTO` or `PVCO` for live oil problems, or keyword `PVDO` for dead oil problems), but with the requirement that all the `PVTO` or `PVCO` tables must have the same upper and lower R_S values. From these tables, ECLIPSE constructs a set of internal tables that have a common set of R_S nodes, which includes the R_S nodes of all the input tables. This allows ECLIPSE to use an efficient method of interpolating between tables, for intermediate values of the API. The restriction that the upper and lower R_S values must be the same for each table prevents extrapolation of the input data during this process, which may cause unphysical behavior.

The API values associated with each PVT table are set using the keyword GRAVITY, which inputs the oil, water and gas gravities for each PVT table number. The API values are converted internally to oil surface densities, using the formula given in "Conversion factors" on page 1136. The oil API values must **decrease** monotonically with the table number. Alternatively the surface densities associated with each table number can be input directly using the keyword DENSITY. In this case the oil surface densities must **increase** monotonically with the table number.

The PVT properties of water and gas are not affected by the API Tracking option. The same number of tables must be entered in all the PVT data keywords and the ROCK keyword, but normally only the first table is actually used in all except the oil PVT data. The unused PVT and rock compressibility tables can be defaulted by typing a single slash (/) on a new line for each unused table in the keyword.

Grouping tables into PVT regions

The API Tracking facility over-rides the concept of PVT table regions for oil. The tables used to determine the oil PVT properties are selected according to the surface density of the oil in the grid block, instead of the block's PVT region number. Thus the PVTNUM keyword in the REGIONS section is not normally required with the API tracking facility.

If however the concept of PVT regions for different types of oil is still required in an API Tracking run, the oil PVT tables can be partitioned into groups for use in different regions of the reservoir. The keyword APIGROUP must be entered, to specify the maximum number of groups of oil PVT tables. The grouping of the oil PVT tables is then determined by the order in which they are entered. Within each group, the oil API gravities should decrease (or the oil surface densities should increase) monotonically with the table number. A break in the required monotonicity is taken to indicate that the subsequent tables belong to the next API group. An error is flagged if the total number of API groups exceeds the maximum specified in the APIGROUP keyword.

The separate groups of oil PVT tables can then be used in different areas of the reservoir. The REGIONS Section keyword PVTNUM is used to specify which API group is to be used for each grid block. For example, a cell in PVTNUM region 2 uses the second API group to obtain the oil properties.

For water and gas PVT properties, the PVTNUM regions correspond to the actual table numbers, as in runs that do not use API tracking. So a cell in PVTNUM region 2 still uses table number 2 for its water and gas properties.

Setting the initial conditions

The initial API distribution throughout the reservoir is specified in the SOLUTION section. If the initial conditions are determined by equilibration, the API can vary with depth independently in each equilibration region. Keyword APIVD supplies a table of API values versus depth for each equilibration region.

When setting the initial conditions by enumeration (keywords PRESSURE, SWAT etc.), the keyword OILAPI is used to supply the initial API values for each grid block.

The API tracking calculation

The initial API values in the grid blocks are immediately converted into oil surface density. The hydrostatic pressure gradient calculation takes account of the varying surface density of the oil. The variation of other PVT properties with surface density (bubble point pressure, formation volume factor and viscosity) is handled by interpolating between PVT tables. The two adjacent PVT tables whose oil surface densities straddle the oil surface density value in the grid block are located, and their properties ($1/B_o$, $1/(B_o\mu_o)$) are interpolated linearly in oil surface density.

To calculate the bubble point, the two PVT tables corresponding to oil surface gravities either side of the current grid block API value are located. The bubble point is then obtained by linearly interpolating between the bubble points in these PVT tables at the given grid block dissolved gas-oil ratio.

The oil surface density in each grid block and well is held constant over the time step. When a converged solution for the time step has been found, and the inter-block flows determined, a mass conservation equation is solved to update the oil surface densities. The new densities are used in the next time step.

The oil surface densities are converted back into API values for the output reports. The mnemonic `OILAPI` in the `RPTSCHED` keyword provides an output of the grid block API values, and the well reports include the API values in the wells.

API tracking in wells

The oil surface density in each well is calculated at the end of the time step to reflect a flow-weighted average surface density of the inflowing oil. A crossflowing well reinjects oil of this average surface density back into the formation. However, the hydrostatic head calculation in the wellbore is based on a (more accurate) flow-weighted average of **upstream** connection inflows if `WELSPECS` item 12 is '`SEG`', allowing the oil density to vary with position in the wellbore. (The pre-98A treatment of wellbore hydrostatic head, which uses the well's average oil surface density, can be restored if required by setting switch 35 in the `OPTIONS` keyword.)

If the well uses a VFP table to calculate the tubing pressure losses, this should be calculated using a suitable value of the oil surface density. However, it is possible to take account of the variation of the well's producing API over time by defining the fifth lookup variable, the ALQ, to represent the surface density of the produced oil (see keyword `WALQCALC`). The VFP table should be prepared using the same definition of the ALQ, with values spanning the expected range of oil surface densities. The VFP*i* program contains an option to prepare tables with variable oil surface density.

Behavior in wet gas systems

By default, in wet gas systems, the surface density property (API) of the oil is transported in both the oil and gas phases, that is the API is interpreted as a property of the oil component rather than the liquid oil phase. However, it is the API in the liquid oil phase that is used in subsequent calculations involving the surface density of oil.

By using `OPTIONS` switch `58` this behavior can be modified. When the switch is set greater than zero the API property is only transported in the liquid oil phase. In this case the API property can be thought of as a heavy component that does not vaporize. Care needs to be taken when using this option as it is possible to vaporize all the oil in a cell, leading to high surface densities and negative API values.

Aquifer Modeling Facilities

Chapter 5

Introduction

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

This chapter describes the aquifer models available in both ECLIPSE 100 and ECLIPSE 300.

The models available are:

- Numerical aquifers
- Carter-Tracy aquifers
- Fetkovich aquifers
- Constant flux aquifers
- Constant head aquifers.

Numerical aquifers are represented by a one-dimensional row of cells within the simulation grid. The other four types of aquifer, classed as ‘analytic aquifers’, are represented by computed source terms in the reservoir grid cells with which they connect.

Numerical aquifer

A numerical aquifer is modeled by a one-dimensional row of cells. A set of cells in the simulation grid is nominated to represent the aquifer, which may then connect to specified faces of the reservoir (using the AQUCON keyword).

The properties of the aquifer grid blocks (length, cross-sectional area, porosity, permeability, initial pressure, depth, PVT and saturation table numbers) may be declared explicitly using keyword AQUNUM in the GRID section, or if the properties for the aquifer cells are defaulted, they are taken from the grid block values using data entered in the GRID and EDIT sections. This enables the aquifer properties to be independent of its actual position within the grid.

The aquifer should be connected to a face of the reservoir using AQUCON. The first cell of the aquifer declared in AQUNUM connects to the specified faces of the reservoir by non-neighbor connections. The set of cells defining the aquifer are connected together (possibly by non-neighbor connections) in the order specified in AQUNUM. The aquifer cells are isolated from the grid except for these connections. In dual porosity runs the aquifer should be placed in the lower half (that is the fracture part) of the grid.

The one dimensional rows of cells each have cross-section, length and depth, but no further information concerning their shape.

The aquifer pore volume is calculated from the data entered in the AQUNUM keyword using the equation

$$\mathrm{PORV}_i = \mathrm{PORO}_i \cdot \mathrm{LENGTH}_i \cdot \mathrm{AREA}_i \qquad \text{[EQ 5.1]}$$

where

PORV_i is the pore volume of the aquifer cell

PORO_i is the porosity of the aquifer cell

LENGTH_i is the length of the aquifer cell

AREA_i is the area of the aquifer cell.

The porosity of the aquifer cell is taken from the grid block if it is defaulted in the AQUNUM keyword and any values of MULTPV entered in the GRID or EDIT sections will not apply to the numerical aquifer cells. Any edits to PORV in the EDIT section for numerical aquifer cells are ignored and the aquifer pore volume is always calculated from equation [EQ 5.1].

ECLIPSE 100

Transmissibility values are calculated using the OLDTRAN block centered method (see "Transmissibility Calculations" on page 1103).

The transmissibility between two aquifer cells is thus given by:

$$\mathrm{TR} = \frac{\mathrm{CDARCY}}{1/T_i + 1/T_j} \qquad \text{[EQ 5.2]}$$

with

$$T_i = \frac{2 \cdot \mathrm{PERMX}_i \cdot \mathrm{XSECT}_i}{\mathrm{LENGTH}_i}$$

where

PERMX_i is the permeability of aquifer cell i

XSECT_i its cross-section.

This expression is used for transmissibilities between aquifer cells in both radial and cartesian geometries. However, in connecting the first aquifer cell to a grid block, the appropriate radial or cartesian transmissibility from the block edge to the center is used (see the AQUCON keyword).

ECLIPSE 100

If required, a transmissibility multiplier may be applied to the calculated transmissibility between the first aquifer block and the nominated connecting face of the reservoir, using keyword AQUCON, in order to aid history matching.

No multiplier is applied to the transmissibility of the inter-block connections within the one-dimensional aquifer. The values of transmissibility multipliers for the grid (MULTX and so on) entered in the GRID section do not apply to any numerical aquifer connections.

ECLIPSE 100

If the initial aquifer pressure is not entered in keyword AQUNUM, then the initial pressure is calculated to be in equilibrium with the reservoir by using the initial pressures for the underlying grid blocks obtained by equilibration. Note that, when initializing the solution in numerical aquifer blocks, ECLIPSE sets the water saturation to unity, so the initial state is not then quiescent if the representative cells do not lie completely in the water zone.

ECLIPSE 300

For the AQUNUM keyword, if the initial aquifer pressure is not entered (by choosing the default value) or even if a negative value is specified, then the initial pressure will be calculated to place the aquifer as near as possible in equilibrium with the reservoir.

Summary quantities for a numerical aquifer may be obtained by defining each aquifer as a fluid-in-place region. It is then possible to examine the aquifer inflow performance and pressure support supplied to the reservoir.

In order to use the facility, the maximum number of numerical aquifer blocks and the maximum number of lines of connection data in AQUCON should be set using the RUNSPEC section keyword AQUDIMS.

Carter-Tracy aquifer

The Carter-Tracy aquifer model is a simplified approximation to a fully transient model, which avoids the need for superposition. The method uses a table that supplies a constant terminal rate influence function. Although the theory has been developed for a radially symmetric reservoir surrounded by an annular aquifer, the method is applicable to arbitrarily-shaped reservoirs.

The two main parameters that govern the behavior of the aquifer are the time constant (with the dimension of time):

$$T_c = \frac{\mu_w \phi C_t r_o^2}{k_a c_1} \qquad \text{[EQ 5.3]}$$

where

k_a is the aquifer permeability

ϕ is the aquifer porosity

μ_w is the viscosity of water in the aquifer

C_t is the total (rock + water) compressibility

r_o is the outer radius of the reservoir (or inner radius of the aquifer)

c_1 is 0.008527 (`METRIC`, `PVT-M`); 0.006328 (`FIELD`); 3.6 (`LAB`)

and the aquifer influx constant (with the dimension of total influx per unit pressure drop)

$$\beta = c_2 h \theta \phi C_t r_o^2 \qquad \text{[EQ 5.4]}$$

where

h is the aquifer thickness

θ is the angle subtended by the aquifer boundary from the center of the reservoir, in degrees divided by 360

c_2 is 6.283 (`METRIC`, `PVT-M`); 1.1191 (`FIELD`); 6.283 (`LAB`).

The time constant T_c is used to convert time t into dimensionless time t_D through

$$t_D = \frac{t}{T_c} \qquad \text{[EQ 5.5]}$$

The Carter-Tracy model uses tables of dimensionless time versus a dimensionless pressure influence function. The default table is for an infinite aquifer with constant terminal rate as given by van Everdingen and Hurst, but you may supply alternative tables using `AQUTAB` in the `PROPS` section.

The Carter-Tracy model expresses the pressure drop at the aquifer boundary in terms of the dimensionless pressure influence function PI_D by

$$p_{a0} - \bar{p} = \frac{Q_a}{\beta} \mathrm{PI}_D(t_D) \qquad \text{[EQ 5.6]}$$

where

Q_a is the aquifer inflow rate

p_{a0} is the initial pressure of water in the aquifer

$\bar{p}$ is the average water pressure on the aquifer/reservoir boundary.

The average inflow rate from the aquifer to a grid block i over a simulator time interval $[t, t+\Delta t]$ is calculated as

$$\overline{Q_{ai}} = \alpha_i\{a - b[p_i(t+\Delta t) - p_i(t)]\} \qquad \text{[EQ 5.7]}$$

where

$$a = \frac{1}{T_c}\left\{\frac{\beta\Delta p_{ai} - W_a(t)\mathrm{PI}_D'(t+\Delta t)_D}{\mathrm{PI}_D(t+\Delta t)_D - t_D\mathrm{PI}_D'(t+\Delta t)_D}\right\} \qquad \text{[EQ 5.8]}$$

and

$$b = \frac{\beta}{T_c[\mathrm{PI}_D(t+\Delta t)_D - t_D\mathrm{PI}_D'(t+\Delta t)_D]} \qquad \text{[EQ 5.9]}$$

where

Δp_{ai} is the pressure drop $p_{a0} + \rho g(d_i - d_a) - p_i(t)$

PI_D' is the derivative of PI_D with respect to t_D

α_i is the area fraction for each connection.

Here, the area fraction for each connection is given by:

$$\alpha_i = \frac{m_i A_i}{\sum m_i A_i}$$

where

A_i is the area of the block face communicating with the aquifer

m_i is an aquifer influx coefficient multiplier.

The aquifer influx rates calculated from [EQ 5.7] contribute the residual for the implicit equations solved by ECLIPSE at time t. The cumulative aquifer influx $W_a(t)$ used in [EQ 5.8] is updated explicitly at the end of the time step.

The aquifer properties (compressibility, porosity, initial pressure, depth, radius etc.) should be defined using keyword `AQUCT`, and the aquifer connections to one or more faces of the reservoir should be made using the keywords `AQUANCON` and `AQANCONL`.

ECLIPSE 100

If the initial aquifer pressure is defaulted, it is calculated from [EQ 5.12] such that the aquifer is in initial equilibrium with the reservoir.

The aquifer should also be given a table number for the influence function table that it should use. Influence function tables may be supplied with the `AQUTAB` keyword in the `PROPS` section. ECLIPSE has a built-in default influence function table (table number 1), which represents the constant terminal rate solution for an infinite aquifer as given by van Everdingen and Hurst.

For Tracer Tracking runs, the initial concentrations of water phase tracers in the aquifer may be supplied using the keyword `AQANTRC` in the `SOLUTION` section.

ECLIPSE 100

For the Brine Tracking option, the initial salt concentration may be supplied using the `AQUCT` keyword in the `SOLUTION` section.

Summary file quantities for the aquifer may be obtained using AAQR (influx rate) and AAQT (cumulative influx) in the SUMMARY section.

In order to use the facility, the maximum number of analytic aquifers and the maximum number of grid block connected to any single analytic aquifer should be set using the RUNSPEC section keyword AQUDIMS.

Fetkovich aquifer

The Fetkovich aquifer model uses a simplified approach based on a pseudosteady-state productivity index and a material balance relationship between the aquifer pressure and the cumulative influx.

This model assumes that the pressure response is felt uniformly throughout the entire aquifer.

The Fetkovich model is best suited for smaller aquifers that may approach a pseudo steady-state condition quickly.

The aquifer inflow is modeled by the equation:

$$Q_{ai} = \frac{d}{dt}(W_{ai}) = J\alpha_i[p_a + p_c - p_i + \rho g(d_i - d_a)] \qquad \text{[EQ 5.10]}$$

where

Q_{ai}

is the inflow rate from the aquifer to the connecting grid block i

W_{ai} is the cumulative influx from the aquifer to grid block i

J is the specified Productivity Index of the aquifer

α_i is the area fraction for the connection to grid block i

p_a is the pressure in the aquifer at time t

p_i is the water pressure in a connecting grid block i

ρ is water density in the aquifer

d_i is the grid block depth

d_a is the datum depth of the aquifer.

p_c is the capillary pressure.

ECLIPSE 100

If the Polymer model is in use, similar equations are applied for the polymer and brine components.

The area fraction for each grid block connection is given by:

$$\alpha_i = \frac{m_i A_i}{\sum m_i A_i} \qquad \text{[EQ 5.11]}$$

where

A_i is the area of the block face communicating with the aquifer

m_i is an aquifer influx coefficient multiplier.

For vertical faces, the areas A_i are multiplied by the net-to-gross ratio of the connecting grid blocks. You may specify the influx multiplier m_i to help with the history matching process.

The pressure response in the aquifer is determined by the material balance equation

$$W_a = C_t V_{w0}(p_{a0} - p_a) \qquad \text{[EQ 5.12]}$$

where

W_a is the cumulative total influx from the aquifer

C_t is the total (rock + water) compressibility of the aquifer

V_{w0} is the initial volume of water in the aquifer

p_{a0} is the initial pressure of water in the aquifer.

If the brine tracking or passive tracer options are used, ECLIPSE 100 also accounts for the inflow of the tracer or brine from the aquifer.

The aquifer performance essentially depends on two parameters, the aquifer time constant and the productivity index. The aquifer time constant is given by

$$T_c = \frac{C_t V_{w0}}{J} \qquad \text{[EQ 5.13]}$$

and has the dimension of time.

Under the assumption of uniform reservoir pressure in the connecting grid blocks, and by integrating [EQ 5.10] and [EQ 5.12], the average influx rate over the time interval Δt is expressed as

$$\overline{Q_{ai}} = \alpha_i J(p_a - p_i + \rho g(d_i - d_a))\left(\frac{1 - \exp(-\Delta t / T_c)}{\Delta t / T_c}\right) \qquad \text{[EQ 5.14]}$$

which is the form used by the simulator compute the influx rates.

At the end of each time step, the aquifer's cumulative total influx is incremented and its pressure is updated using [EQ 5.12].

By varying the aquifer volume and the productivity index, the Fetkovich model can encompass a range of aquifer behavior from steady state to the 'pot' aquifer. If the product $C_t V_{w0}$ is given a large value, so that the time constant is very large, the behavior approaches that of a steady state aquifer, in which the pressure on the external boundary does not change. Alternatively, if the productivity index is given a large value so that the time constant is small, the behavior approaches that of a 'pot' aquifer, in which the pressure is in approximate equilibrium with the reservoir at all times.

The aquifer properties (compressibility, porosity, initial pressure, depth, productivity index etc.) should be defined using keyword `AQUFETP`, and the aquifer connections to one or more faces of the reservoir should be made through the keywords `AQUANCON` and `AQANCONL`.

ECLIPSE 100

Alternatively, both the aquifer properties and the connection data may be defined with a single keyword `AQUFET`, but this is less versatile as it allows the aquifer to connect to only one face of the reservoir.

If the initial aquifer pressure is defaulted, it is calculated from [EQ 5.10] such that the aquifer is in initial equilibrium with the reservoir.

For Tracer Tracking runs, the initial concentrations of water phase tracers in the aquifer may be supplied using the keyword `AQANTRC`. For the Brine Tracking option, the initial salt concentration may be supplied using `AQUFET` (ECLIPSE 100 only) or `AQUFETP`.

`SUMMARY` file quantities for the aquifer may be obtained using `AAQR` (influx rate), `AAQT` (cumulative influx) and `AAQP` (aquifer average pressure) in the `SUMMARY` section.

In order to use the facility, the maximum number of analytic aquifers and the maximum number of grid block connected to any single analytic aquifer should be set using the `RUNSPEC` section keyword `AQUDIMS`. (For more information, see [Ref. 1].)

Constant flux aquifer

A constant flux aquifer has its water influx rate specified directly by the user, instead of being calculated by an analytic aquifer model. But for the purpose of dimensioning etc., it is classed with the other analytic aquifer models. One use of a constant flux aquifer is to model rainfall influx for environmental applications.

The water flow rate into a grid block from a constant flux aquifer is given by:

$$Q_{ai} = F_a \cdot A_i \cdot m_i \qquad \text{[EQ 5.15]}$$

where

Q_{ai} is the inflow rate from the aquifer to the connecting grid block i

F_a is the aquifer flux, entered by the user

A_i is the area of the connecting face of grid block i, which is calculated directly from the cell geometry

m_i is the aquifer influx multiplier.

The aquifer should be defined with the AQUFLUX keyword in the SOLUTION section, and the aquifer connections to one or more faces of the reservoir should be made via the keywords AQUANCON and AQANCONL. To achieve time-dependency, the flux may be modified during simulation by re-entering the AQUFLUX keyword in the SCHEDULE section

For Tracer Tracking runs, the concentrations of water phase tracers in the aquifer may be supplied using the keyword AQANTRC.

SUMMARY file quantities for the aquifer may be obtained using AAQR (influx rate) and AAQT (cumulative influx) in the SUMMARY section.

In order to use the facility, the parameters NANAQU (the maximum number of analytic aquifers) and NCAMAX (the maximum number of grid block connected to any single analytic aquifer) should be set using the RUNSPEC section keyword AQUDIMS.

ECLIPSE 100

The keyword RAINFALL can be used to model a monthly varying flux rate for a constant flux aquifer.

Constant head aquifers

A constant head aquifer can be defined in the SOLUTION section by the AQUCHWAT keyword for a water aquifer and by the AQUCHGAS keyword for a gas aquifer. The aquifer connections to one or more faces of the reservoir should be made via the keywords AQUANCON and AQANCONL to achieve time-dependency, the conditions may be modified during simulation by re-entering the AQUCHWAT or AQUCHGAS keywords in the SCHEDULE section.

For Tracer Tracking runs, the concentrations of water or gas phase tracers in the aquifer may be supplied using the keyword AQANTRC.

SUMMARY file quantities for the aquifer may be obtained using AAQR (influx rate) and AAQT (cumulative influx) in the SUMMARY section.

In order to use the facility, the parameters NANAQU (the maximum number of analytic aquifers) and NCAMAX (the maximum number of grid block connected to any single analytic aquifer) should be set using the RUNSPEC section keyword AQUDIMS.

The aquifer inflow equations are given in the description of AQUCHWAT and AQUCHGAS respectively.

Multi-component analytic aquifers

ECLIPSE 300

Constant head gas aquifers defined by the AQUCHGAS keyword can be composed of any components that can be present in the gas phase. Refer to the AQUCHGAS description for more information.

When using one of the CO2SOL, CO2STORE, COMPW, GASWAT options, any water analytic aquifers can be composed of any component that can be present in the water phase.

Please refer to:

- "Analytic water aquifers" on page 138 for the CO2STORE and GASWAT options,
- "Aquifer properties" on page 146 for the CO2SOL option,
- "Aquifer properties" on page 1159 for the COMPW option.

For the multi-component aquifers listed above, the summary keywords AACMR and AACMT allow the user to access the component molar influx rates and totals.

List of keywords

RUNSPEC

The RUNSPEC keywords are:

Table 5.1 Aquifer RUNSPEC keywords

E100	E300	Keyword	Description
x	x	AQUDIMS	Dimensions for aquifers

GRID

The GRID keywords are:

Table 5.2 Aquifer GRID keywords

E100	E300	Keyword	Description
x	x	AQUCON	Specifies connection data for numerical aquifers
	x	AQUCT	Specifies the property data for Carter-Tracy aquifers
x		AQUNNC	Set non-neighbor connection values explicitly for numerical aquifers
x	x	AQUNUM	Assigns a numerical aquifer to a block
x	x	INIT	Output aquifer connection data to the INIT file. Aquifer connection arrays AQUIFERN, AQUIFERA and AQUIFERG are written to the init file depending on whether numerical or analytic aquifers are present.
x	x	RPTGRID	Controls output from the GRID section to the print file: Use AQUCON and AQUNUM to output numerical aquifer connections and definitions respectively.

PROPS

The PROPS keywords are:

Table 5.3 Aquifer PROPS keywords

E100	E300	Keyword	Description
	x	AQUCT	Specifies the property data for Carter-Tracy aquifers
x	x	AQUTAB	Influence function tables for Carter-Tracy aquifers
x		RPTPROPS	Controls output from the PROPS section to the print file: AQUTAB outputs Carter-Tracy influence tables.

SOLUTION

The SOLUTION keywords:

Table 5.4 Aquifer SOLUTION keywords

E100	E300	Keyword	Description
x	x	AQANCONL	Specifies local grid refinement connection data for analytic aquifers
x		AQANNC	Set connection values explicitly for analytic aquifers
x	x	AQANTRC	Sets initial tracer concentrations for analytic aquifers
	x	AQSTREAM	Set composition of injected gas stream for AQUCHGAS aquifer or set composition of AQUCHGAS aquifer or set composition of GASWAT water analytic aquifer
	x	AQSTREAW	Set composition of multi-component water analytic aquifer
x		AQUALIST	Sets up lists of aquifers for output
x	x	AQUANCON	Specifies connection data for analytic aquifers
x	x	AQUCHGAS	Specifies the property data for constant head/pressure gas aquifers
x	x	AQUCHWAT	Specifies the property data for constant head/pressure water aquifers
x	x	AQUCT	Specifies the property data for Carter-Tracy aquifers
x		AQUFET	Specification data for Fetkovich aquifers
x	x	AQUFETP	Specifies the property data for Fetkovich aquifers

Table 5.4 Aquifer SOLUTION keywords

E100	E300	Keyword	Description
x	x	AQUFLUX	Specifies a constant flux aquifer
x		RAINFALL	Specifies a monthly varying flux rate for a constant flux aquifer
x	x	RPTSOL	Controls output from the SOLUTION section to the print file: Use AQUFET or AQUFETP or AQUCT or AQUFLUX or AQUANCON or AQANCONL to output analytic (Fetkovich, Carter-Tracy, Constant Flux or Constant Head) aquifer data (entered using AQUFET, AQUFETP, AQUCT, AQUCHWAT, AQUCHGAS and AQUFLUX). ECLIPSE 100: If set to 2: in addition, data for each aquifer-grid block connection will be printed (input using AQUANCON). ECLIPSE 300: Data for each aquifer-grid block connection will be printed (input using AQUANCON). Note: AQUFET applies only for ECLIPSE 100 Note: AQANCONL applies only for ECLIPSE 300

SUMMARY

The SUMMARY keywords for analytical aquifers are:

Table 5.5 Analytic aquifer quantities

E100	E300	Field	Aquifer	Aquifer List (E100)	Description
x	x	FAQR	AAQR	ALQR	Aquifer influx rate
x	x	FAQT	AAQT	ALQT	Cumulative aquifer influx
x	x	FAQRG	AAQR		Aquifer influx rate (for Constant Head Gas aquifer)
x	x	FAQTG	AAQT		Cumulative aquifer influx rate (for Constant Head Gas aquifer)
	x		AACMR		Aquifer component molar influx rate (for multi-component aquifer)
	x		AACMT		Aquifer component molar influx totals (for multi-component aquifer)
x	x		AAQP		Aquifer pressure
	x	FAQER	AAQER		Aquifer thermal energy influx rate
	x	FAQET	AAQET		Cumulative aquifer thermal energy influx
	x		AAQTEMP		Aquifer temperature
	x		AAQENTH		Aquifer molar enthalpy

The SUMMARY keywords for numerical aquifer quantities are:

Table 5.6 Numerical aquifer quantities

E100	E300	Field	Aquifer	Description
x	x	FNQR	ANQR	Aquifer influx rate
x	x	FNQT	ANQT	Cumulative aquifer influx
x	x		ANQP	Aquifer pressure (average pressure weighted by water volume)
	x		ANCMR	Aquifer component molar influx rate
	x		ANCMT	Aquifer component molar influx totals

SCHEDULE

The SCHEDULE keywords are:

Table 5.7 Aquifer SCHEDULE keywords

E100	E300	Keyword	Description
	x	AQSTREAM	Set composition of injected gas stream for AQUCHGAS aquifer or set composition of AQUCHGAS aquifer or set composition of GASWAT water analytic aquifer
	x	AQSTREAW	Set composition of multi-component water analytic aquifer
x	x	AQUCHGAS	Specifies the property data for constant head/pressure gas aquifers
x	x	AQUCHWAT	Specifies the property data for constant head/pressure water aquifers
x	x	AQUCT	Specifies the property data for Carter-Tracy aquifers
x		AQUCWFAC	Changes the property for all constant head aquifers
x	x	AQUFETP	Specifies the property data for Fetkovich aquifers
x	x	AQUFLUX	Specifies a constant flux aquifer
x		RAINFALL	Specifies a monthly varying flux rate for a constant flux aquifer
	x	RPTPRINT	Controls output of printed reports. Use item 13 of keyword to report aquifer flows
x	x	RPTSCHED	Controls output from the SCHEDULE section to the print file: Use AQUCT or AQUFET or AQUFETP to output Fetkovich or Carter-Tracy aquifer status. Also outputs AQUCHWAT, AQUCHGAS and AQUFLUX status. Note: AQUFET applies only to ECLIPSE 100

The Asphaltene Option

Chapter 6

Introduction

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

The purpose of the Asphaltene option is to model asphaltene precipitation and its impacts on reservoir performance. There are a number of asphaltene models currently in existence, but there is still no consensus about the characterization of asphaltene behavior. This release of the Asphaltene Option implements a simplified but practical dissolution-like model that provides users with sufficient flexibility to model a wide range of processes associated with asphaltene behavior.

Given that the terminology of these processes in the literature may be different, here we call

- precipitation: the process that triggers the appearance of the asphaltene particles (“fines”).
- flocculation: the process whereby the precipitated fines aggregate to form bigger particles, (“flocs”).
- deposition: the mechanistic process of exchanging flocs between the liquid solution (oil) and the solid matrix making up the reservoir rock.

Basically, the asphaltene modelling is decomposed into different stages as depicted below.

The double arrow indicates reversibility (partial or total). Precipitation triggers the sequence of flocculation deposition and damage, including a viscosity effect, as detailed below.

The asphaltene modeling option is activated with `ASPHALTE` keyword in the `RUNPSEC` section.

Asphaltene Characterization and Precipitation

In this release, we model the precipitation, flocculation, deposition and damage processes using separate keywords. The basis of this model is that asphaltene is characterized by the weight percentage of component(s) making up the asphaltene that can remain dissolved in the oil as a function of pressure.

Asphaltene as weight percentage

In this approach the asphaltene is defined as a set of component(s) that can precipitate depending on their percentage molar weight in the solution. The percentage limit is defined by the user as a function of pressure using the `ASPREWG` keyword.

This limit can be set against the pressure or temperature or molar fraction of a specified component using the `ASPP1P` keyword. The default setting is against pressure.

This limit can also be set as a function of two variables chosen from pressure (P), temperature (T) and molar fraction (Z). This is specified using keywords `ASPP2P` and `ASPPW2D`. Keyword `ASPP2P` sets the variables (P,T) or (P,Z) or (T,Z). If the molar fraction Z is chosen the 3rd argument of `ASPP2P` must be the component number. Keyword `ASPPW2D` takes a 2D table where the first row represents the temperature if (P,T) is chosen, and the molar fraction if (P,Z) or (T,Z) is chosen. The first column of the 2D table represents the pressure if (P,T) or (P,Z) is chosen and the temperature if (T,Z) is chosen.

The amount of precipitate corresponds to the excess of a specified component in the oil phase with respect to the limit defined by the user.

The `ASPREWG` keyword like the`ASPPW2D` keyword is used in conjunction with the `ASPFLOC` keyword, which is used to specify the range of components that can precipitate. This approach can be used to model some aspects of the polydispersivity that has been reported in the literature. It can also be interpreted to some extent as a dissolution model, where the oil phase can only dissolve a certain amount of asphaltene component(s) at a given pressure, temperature or molar fraction.

Note Note that the precipitation process does not change the fluid composition. It only flags the fraction of a precipitate component that will contribute to the flocculation process.

Assuming that the list of components is enumerated as *1,2,...,i,i+1,...j,...k,..., N*. If the first component number to precipitate is *i,* the last component is *j*, and *k* is the flocs component then the dissolved fraction of asphaltene is simply:

$$w_{adis} = \frac{\sum_{l=i}^{j} m_{wl} x_l}{\sum_{l=1}^{N} m_{wl} x_l} \qquad \text{[EQ 6.1]}$$

and the total fraction of asphaltene (dissolved and flocs) is calculated as:

$$w_{apre} = \frac{m_{wk} x_k + \sum_{l=i}^{j} m_{wl} x_l}{\sum_{l=1}^{N} m_{wl} x_l} \qquad \text{[EQ 6.2]}$$

where

m_{wl} is the molecular weight of component l

x_l is the liquid fraction of component l

At a given pressure, temperature or concentration the precipitate fraction coming out from the dissolved asphaltene gives the excess of w_{adis} with respect to user limit input entered using ASPREWG keyword, as:

$$E = w_{adis} - asp_{lim}(P)$$

The $asp_{lim}(P)$ limit is interpreted as a fraction of asphaltene component(s):

$$asp_{lim}(P) = \frac{\sum_{l=i}^{j} m_{wl} x_l^a}{\sum_{l=1}^{N} m_{wl} x_l} \qquad \text{[EQ 6.3]}$$

and the composition of the precipitate is calculated by assuming that x_l^a is the same for all the asphaltene component(s), which means that

$$x_l^a = asp_{lim}(P) \frac{\sum_{l=1}^{N} m_{wl} x_l}{\sum_{l=i}^{j} m_{wl}} \qquad \text{[EQ 6.4]}$$

Hence the fraction of precipitate from the dissolved asphaltene writes:

$$E = \frac{\sum_{l=i}^{j} m_{wl}(x_l - x_l^a)}{\sum_{l=1}^{N} m_{wl} x_l} \qquad \text{[EQ 6.5]}$$

The 2nd argument of ASPREWG can take values from 0 (no dissolved asphaltene component(s)) to 100 (no precipitate), however the actual upper can be estimated from the expression of w_{adis} if the molar fraction(s) of the precipitating component(s) is known. The $asp_{lim}(P)$ limit and w_{adis} can be output to check out the evolution of precipitate fraction. A list of outputs for the precipitation process is provided in "Outputs" on page 100.

An example

Let consider a case with 5 components, and assume component 4 is precipitating and component 5 is the flocs component, then the amount of precipitate is calculated as depicted below:

Asphaltene Flocculation-Dissociation

Flocculation is the process whereby fines obtained from the precipitation process aggregate to form bigger particles referred to as flocs. This process is modeled by a set of two kinetic reactions to allow reversibility (partial or total) between aggregation of the fines into flocs and dissociation of the flocs into fines.

Let C_i denote the concentration of the fines coming from component *i* and C_a the concentration of the flocs. Once precipitation occurs, the aggregation rate of the fines *i* into flocs *a* is modeled by:

$$R_a = \frac{\partial C_a}{\partial t} = r_{ia} C_i - r_{ai} C_a \qquad \text{[EQ 6.6]}$$

where r_{ia} is the flocculation rate coefficient of the fines and r_{ai} is the dissociation rate coefficient of the flocs.

The range of components that precipitate and the flocs component *a* are specified by the ASPFLOC keyword. This keyword is used in conjunction with the ASPFLRT keyword that specifies rates r_{ia} and r_{ai}. This process is depicted in the figure below.

In the case where the asphaltene is seen as single pure component, this flocculation reduces to two kinetic reactions only.

The maximum number of flocculation is set using argument 20 of TABDIMS keyword. The default value is 5.

Data input

The flocculation-Dissociation process can generate significant changes in the composition, which may result in un-physical behavior if data input is not carefully set up. Given that the flocs component is a result of the flocculation process, it is expected that the flocs properties (molar weight, critical properties, and other EOS parameters) to be some average values of the components that precipitate. These average values should also take into account the flocculation rates to have the right weight contribution of the different components that precipitate.

In this release, it is expected that the user provides flocculation properties that are consistent with the components that precipitate. A helpful tool to provide an estimate of the flocculation properties would be a PVT package with a grouping capability that derives the properties of a pseudo-component from its parts.

As the composition of the oil can be changed by the flocculation and/or deposition process(es), the oil/gas densities will change, and therefore the pressure too.

Asphaltene Deposition

Deposition represents the process whereby the flocs are exchanged between the solution (oil phase) and the rock surface. The flocs can adsorb on the rock surface, can be trapped within the porous media because of their size (plugging) or can be entrained and returned to the oil phase because of high, local velocity (shear).

Note Note, however, changes in the rock density are not taken into account as they are not thought to contribute to the process at this stage.

The deposition in the flow direction *i* is modeled as follows

$$\frac{\partial \varepsilon_i}{\partial t} = \frac{\alpha}{d}\phi C_a + \gamma |F_{oi}| C_a - \beta(|U_{oi}| - U_{cr})^{+} \varepsilon_i \qquad \text{[EQ 6.7]}$$

where *d* is the dimension of the problem (1,2 or 3) and:

ε_i is the volume fraction of deposit in the *i* direction of the flow

α is the adsorption or static deposition coefficient

ϕ is the current porosity (time *t*)

C_a is the volumetric concentration of the flocs in the oil phase (flowing flocs)

F_{oi} is the oil Darcy flux

γ is the plugging coefficient

β is the entrainment coefficient

U_{oi} is the oil phase velocity ($F_{oi}/(A\phi)$), A is the section area between connecting cells

U_{cr} is the user input critical velocity.

The "+" sign around the bracket for the entrainment part means that the entrainment will be zero if the velocity $|U_{oi}|$ is smaller than the critical value U_{cr}. The overall volume fraction deposit is the sum of the deposits in each direction *i*:

$$\varepsilon = \sum_{i=1}^{d} \varepsilon_i \qquad \text{[EQ 6.8]}$$

The flocs mole fraction that deposits is taken as:

$$n_a^{dp} = \varepsilon n_a^f \qquad \text{[EQ 6.9]}$$

and the flocs mole fraction that remains in solution is corrected to:

$$n_a^f = n_a^f - n_a^{dp} \qquad \text{[EQ 6.10]}$$

The concentration of the flowing part of the flocs is calculated as:

$$C_a = \frac{n_a^f S_o x_a}{b_o} \qquad \text{[EQ 6.11]}$$

where b_o is the oil molar density, S_o the oil saturation and x_a the molar fraction of the flocs in the liquid phase.

Input parameters α, γ ,β , and U_{cr} are user input, and are specified using the ASPDEPO keyword in the PROPS section. This parameters should be obtain from core flood experiments.

Plugging control

Plugging can be controlled by using the ASPLCRT keyword to specify a critical volume fraction of asphaltene deposit ε_{cr} and a critical concentration of flocs C_{acr} above which the plugging happens. If the ASPLCRT keyword is absent the pre-2008.1 modelling is recovered. It actually corresponds to using the defaults values of $\varepsilon_{cr} = 0$ and/or $C_{acr} = 0$. The transport equation is then slightly altered as

$$\frac{\partial \varepsilon_i}{\partial t} = \frac{\alpha}{d}\phi C_a + \gamma |F_{oi}| (C_a - C_{acr})^+ - \beta(|U_{oi}| - U_{cr})^+ \varepsilon_i \qquad \text{[EQ 6.12]}$$

Basically the coefficient γ is set as:

$$\gamma = \begin{cases} \gamma & \text{if } \varepsilon > \varepsilon_{cr} \text{ and/or } C_a > C_{acr} \\ 0 & \text{if not} \end{cases} \qquad \text{[EQ 6.13]}$$

This means that plugging will occur only if the flocs concentration gets above the critical value and/or the volume fraction deposit gets bigger than the critical value. Note that when the concentration gets above critical only the concentration difference will contribute to the plugging.

The critical volume fraction of asphaltene deposit can be interpreted as a reduction in the average pore radius. Using the empirical correlation and [EQ 6.16] a critical radius can be estimated as

$$r \sim \sqrt{\frac{K}{\phi}} \rightarrow r_{cr} \sim r_0 \left(1 - \frac{\varepsilon_{cr}}{\phi_0}\right)^{\frac{\delta - 1}{2}} \qquad \text{[EQ 6.14]}$$

so the remain flowing radius is $r = r_0 - r_{cr}$. Exponent δ is the same as in [EQ 6.16] below. The critical floc concentration can be interpreted as a likelihood that above that value the size of some of the flowing flocs is big enough to plug some pore throats.

The ASPLCRT keyword takes three arguments: the critical volume fraction of asphaltene deposit ε_{cr}, the critical concentration of the flocs C_{acr}, and a third argument equal to “AND” or “OR” for the combination of the two first arguments.

Asphaltene Damage

Porosity

One of the damage associated with asphaltene deposition is a reduction of the porosity due to asphaltene deposition, that can be written as:

$$\phi = \phi_0 - \int_0^t \frac{\partial \varepsilon}{\partial t} dt \qquad \text{[EQ 6.15]}$$

where ϕ_0 is the initial porosity, is ε the volume fraction of asphaltene deposit.

Permeability

If the rock permeability is correlated to the porosity, a reduction in permeability can also be taken into account using a parametrized power law relationship given the ratio of the permeability K at time t with respect to the initial permeability K_0:

$$\frac{K}{K_0} = \left(1 - \frac{\varepsilon}{\phi_0}\right)^{\delta} \qquad \text{[EQ 6.16]}$$

where δ is a user input (based on core experiment data). To use this model the user needs to specify "PORO" in the second argument of `ASPHALTE` keyword, and provide the exponent with the `ASPKDAM` keyword.

Alternatively, if the rock permeability is independent of the porosity, or data giving a relationship between the permeability and the amount of asphaltene deposit are available, this can be directly used within a look-up table. In that case the second argument of the `ASPHALTE` keyword should be set to "TAB" and the `ASPKDAM` keyword takes a table as an argument where the first column represents the volume fraction of asphaltene deposit and the second column is the corresponding permeability multiplier.

Oil viscosity

When precipitation occurs, asphaltene fines can be considered as colloids in the oil phase, which can alter the viscosity of the transporting phase, i.e. oil phase.

Using the `ASPVISO` keyword in conjunction with the 3rd argument of the `ASPHALTE` keyword, the user can model the viscosity alteration in three different ways:

1 Generalized Einstein model (one parameter)

$$\frac{\mu}{\mu_0} = 1 + aC_p \qquad \text{[EQ 6.17]}$$

where the default for a is $a = 2.5$. and C_p is the volume concentration of precipitate, and μ_0 is the oil viscosity at $C_p=0$. The user can specify the coefficient a using `ASPVISO`.

2 Krieger and Dougherty model (two parameter)

$$\frac{\mu}{\mu_0} = \left(1 - \frac{C_p}{C_{p0}}\right)^{-\eta C_{p0}} \qquad \text{[EQ 6.18]}$$

where:

η is the intrinsic viscosity, $\eta = 2.5$ for spherical colloids

C_{p0} is the volumetric concentration for maximum packing, equal to 0.65 for spheres packing.

The user can specify the coefficient η and the maximum volumetric concentration C_{p0} using ASPVISO.

3 Look-up table that gives the oil viscosity multiplier as a function of the volume fraction of the asphaltene precipitate. The first column is the volume fraction and the second column is the viscosity multiplier. The number of rows can be set in item 19 of TABDIMS keyword.

Depending on the model chosen, the ASPVISO keyword takes the appropriate number of entries.

Wettability change

Asphaltene deposition can trigger a wettability change. The exact physics on how this alteration occurs is still a research topic, however it has been reported that some of its effect can be captured by a relative permeability shift from a water-wet system to an oil-wet system. This is how we propose to model the wettability change in this version of ECLIPSE.

In this version of ECLIPSE, the wettability type is based on the water-oil capillary pressure values. If the water-oil capillary pressure is zero or negative there is no wettability change.

Assuming that the system is initially water-wet,i.e a positive water/oil capillary pressure, the relative permeabilities (entered using for example SWFN and SOF3) are combined with the oil-wet saturation functions entered using the ASPKROW keyword as follows:

1 First the critical oil in water and water saturations are scaled

$$S_{ocr} = FS_{ocra} + (1 - F)S_{ocri}$$
$$S_{wcr} = FS_{wcra} + (1 - F)S_{wcri}$$

where S_{ocra} and S_{ocri} are the critical oil in water saturations from ASPKROW table and SOF3 table respectively, and S_{wcra} and S_{wcri} are the critical water saturations from ASPKROW table and SWFN table respectively. The weighting factor F is a function of the volume fraction of asphaltene deposit, and must be provided using the ASPWETF keyword.

2 Then a look-up for relative permeability is carried out on the scaled saturations from step 1, followed by a linear interpolation between the resulting water-wet and oil-wet relative permeabilities as:

$$k_{rw} = Fk_{rwa} + (1 - F)k_{rwi}$$
$$k_{row} = Fk_{rowa} + (1 - F)k_{rowi}$$

where k_{rwa} and k_{rwi} and the oil-wet and water-wet relative permeabilities to water respectively (from ASPKROW and SWFN), and k_{rowa} and k_{rowi} and the oil-wet and the water-wet relative permeabilities to oil respectively (from ASPKROW and SOF3).

The number of tables in the ASPKROW keyword is the same as the number of saturation tables (argument 1 of the TABDIMS keyword), and the maximum number of rows in each table is also the same as for the saturation tables (argument 3 of the TABDIMS keyword).

The number of tables in the ASPWETF keyword is the same as the number of saturation tables, and the maximum number of rows in each table is set by argument 23 of the TABDIMS keyword.

General comments on viscosity behavior

Viscosity behavior

Oil viscosity changes are due to two competing effects:

- the precipitation, as modeled above, tends to increase the oil viscosity
- the deposition of heavier component(s) tends to make the oil lighter and therefore reduces its viscosity.

This can be checked by running a case with deposition parameters set to zero, and running the same case with non-zero deposition parameters.

List of keywords

RUNSPEC section

The RUNSPEC keywords are:

Table 6.1 RUNSPEC keywords

Keyword	Description
ASPHALTE	Activates the asphaltene option. It takes three arguments that select the precipitation model, the permeability damage model and the oil viscosity model.

PROPS section

The PROPS section keywords are:

Table 6.2 PROPS keywords

Keyword	Description
ASPP1P	Sets the variable asphaltene precipitation is tabulated against: pressure P, or molar fraction Z, or temperature T.
ASPREWG	Sets the table for the dissolution-precipitation asphaltene model based on percentage molar weight of asphaltene content as a function of one variable: P or T or Z.
ASPP2P	Sets the variables asphaltene precipitation is tabulated against: (P,T) or (P,Z) or (T,Z).
ASPPW2D	Sets the 2D table for the dissolution-precipitation asphaltene model based on percentage molar weight of asphaltene content as a function of two variables: (P,T) or (P,Z) or (T,Z).
ASPFLOC	Sets the precipitating components range and the flocs component number.
ASPFLRT	Sets the flocculation-dissociation rates.
ASPDEPO	Sets the deposition parameters.
ASPKDAM	Sets the permeability damage model: either an exponent if the permeability is correlated to the porosity OR a table of permeability multiplier as a function of volume fraction of asphaltene deposit.
ASPVISO	Sets the oil viscosity model parameters

Table 6.2 PROPS keywords

Keyword	Description
ASPLCRT	Sets the critical values for the volume fraction of asphaltene deposit and the critical value for flocs concentration for plugging control, and how to use these critical values (AND or OR).
ASPKROW	Sets the water and oil wet relative permeability as a function of water saturation for the wettability change due to asphaltene deposition.
ASPWETF	Sets the wettability weighting factor between the water-wet and oil-wet relative permeability as a function of the volume fraction of asphaltene deposit.

Outputs

The following mnemonics are associated with the RPTRST/RPTSCHED keywords for the ASPHALTENE option

Table 6.3 RPTRST/RPTSCHED keywords mnemonics

Input Keyword or Mnemonic	Output	E300
ASPADS	Asphaltene cumulative adsorption volume* fraction	x
ASPDOT	Asphaltene cumulative net deposition volume fraction	x
ASPENT	Asphaltene instantaneous entrainment volume fraction	x
ASPFL	Asphaltene flocculation-dissociation rates	x
ASPKDM	Asphaltene permeability damage multiplier	x
ASPLUG	Asphaltene instantaneous plugging volume fraction	x
ASPREW	Asphaltene precipitate fraction	x
ASPRET	Asphaltene precipitate fraction including flocs	x
ASPLIM	Asphaltene precipitate fraction limit	x
ASPFRD	Asphaltene fraction precipitated and dissolved	x
ASPVOM	Asphaltene oil viscosity multiplier	x
ASPFLO	Oil phase velocities used in the entrainment calculations. The mnemonics output to the restart file are VELOILI+,VELOILJ+, and VELOILK+ depending on the dimension of the problem.	x

*.The volume in these reported values is considered to be the grid block volume (bulk volume).

The following SUMMARY section keywords are associated with the ASPHALTENE option

Table 6.4 ASPHALTENE option SUMMARY section keywords - block values

E300	Block	Description
x	BASPADS	Asphaltene cumulative adsorption volume* fraction
x	BASPDOT	Asphaltene volume fraction net deposit
x	BASPFL	Asphaltene flocculation/dissociation rate. The number of the flocculation/dissociation needs to follow the block coordinates
x	BASPREW	Asphaltene precipitation fraction (no flocs)
x	BASPRET	Asphaltene precipitation fraction including flocs
x	BASPLIM	Asphaltene precipitate fraction limit
x	BASPFRD	Asphaltene fraction precipitated and dissolved
x	BASPKDM	Asphaltene permeability multiplier (damage)
x	BASPVOM	Asphaltene oil viscosity multiplier
x	BASPLUG	Asphaltene instantaneous plugging contribution to deposition
	BASPENT	Asphaltene instantaneous entrainment contribution to deposition

*.The volume in these reported values is considered to be the grid block volume (bulk volume).

Note When reporting the flocculation rates using the BASPFL or ASPFL mnemonic, the fourth argument is the number of the flocculation/dissociation reaction, not the component number as in BXMF for example. The flocculation rates are reported as *1,..,MFLOCU* and the dissociation rates are reported as *i+MFLOCU*, where *i* is the index of the ith flocculation and *MFLOCU* is the maximum number of flocculation specified in argument 20 of the TABDIMS keyword. For example if there are 2 components precipitating, then the index of the flocculation rates will be 1 and 2, and the index for the dissociation rates will be *1+MFLOCU*, and *2+MFLOCU*

Example

In the RUNSPEC section, switch on Asphaltene deposition model by

```
ASPHALTE
 WEIGHT  /
Then the PROPS section specifies the asphaltene modelling parameters
(e.g. with 7 components):
CNAMES
           C1       C3      C6       C10      C15     C20    ASPH   /
-- Asphaltene parameters
-- ... asphaltene floc components
ASPFLOC
-- first last floc
    6     6    7  /
-- ... define asphaltene concentration limits
ASPREWG
--  pres   %_wt
  1000.0    30.0
 10000.0    100.0  /
-- ... ashphaltene floc rates
ASPFLRT
-- C20 <->ASPH
    0.001
    0.0001 /
-- ... asphaltene deposition (set here to avoid deposition)
ASPDEPO
-- adsorp plug entrain Vcr
5.0E-6  0.0 0.00 1.0E6 /
-- ... asphaltene damage ratio
ASPKDAM
-- deposit mult (set here to avoid permeability damage)
     0.0     1.0
     1.0     1.0  /
-- ... ashphaltene viscosity change (set here for no viscosity change)
ASPVISO
-- vfrac  mult
    0.00  1.00
    1.00  1.00  /
```

Brine Tracking

Chapter 7

Introduction

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

The Brine Tracking facility enables ECLIPSE to model the mixing of waters with different salinities, as well as the effect of low versus high salinity on the flow performance with the Low Salinity option.

The salinities can be made of different salt species. In that case different effective salinities are calculated depending on the features used in conjunction with the brine option. For the Polymer and Surfactant options, ions exchanges are solved to determine the effective salinities used in the adsorption isotherms.

The concentration of salt in the water phase can modify both the water density and the viscosity, and, with the low salinity option, the saturation and relative permeability end points for oil and water and the water-oil capillary pressure.

For example, if low salinity water is injected into a reservoir with a significantly higher natural salinity, the lighter injected water would be expected to over-ride the initial aquifer water.

The Brine Tracking facility is activated by the keyword `BRINE` in the `RUNSPEC` section, and the Low Salinity option is activated by the `LOWSALT` in the `RUNSPEC` as well.

ECLIPSE 100 only

The Multi-Component Brine option is activated by the keyword `BRINE` in conjunction with the keyword `ECLMC` in the `RUNSPEC` section.

The Brine Model

Brine conservation equation

The distribution of brine is modeled by solving a mass conservation equation for the salt concentration in each grid block. Brine is assumed to exist solely in the water phase and is modeled internally as a water phase tracer using the equation

$$\frac{d}{dt}\left(\frac{VS_wC_s}{B_w}\right) = \sum\left[\frac{Tk_{rw}}{B_w\,\mu_{s\,\mathrm{eff}}}(\delta P_w - \rho_w g D_z)\right]C_s + Q_wC_s \qquad \text{[EQ 7.1]}$$

where

ρ_w denotes the water density

Σ denotes the sum over neighboring cells

C_s denotes the salt concentration in the aqueous phase

$\mu_{s\mathrm{eff}}$ denotes the effective viscosity of the salt

D_z is the cell center depth.

B_w is the water formation volume

T is the transmissibility

k_{rw} is the water relative permeability

S_w is the water saturation

V is the block pore volume

Q_w is the water production rate

P_w is the water pressure

g is the gravity acceleration

By default, the salt concentrations are updated at the end of a time step after the inter-block phase flows have been determined. If the salinity effect of the flow properties is very strong, the salt concentrations can be solved in each Newton iteration so that the tracer concentrations are updated simultaneously with the phase equations by setting the 126th item of the `OPTIONS` keyword to 1. See "Solving the Conservation Equations" on page 112 for more details. The new salt concentrations are then used to update the water properties for the subsequent iterations.

Multi-Component Brine model

ECLIPSE 100 only

In the standard Brine Tracking option all salt species are lumped into a single salt. The Multi-Component Brine model enables the modelling of processes where different salt species affect fluid properties in different ways. For example, when used in conjunction with the Surfactant and Polymer options, effective salinities are calculated based on the transport and exchange of different ions. These effective salinities can then be used when modeling surfactant phase behavior and polymer/surfactant adsorption.

In the current Multi-Component Brine model three different ion species are considered: anions, monovalent cations, and divalent cations. All salts are assumed to be fully dissociated into their ion constituents, and salt precipitation/dissolution is therefore not accounted for. In this release, the ion species are transported as water phase tracers according to:

$$\frac{d}{dt}\left(\frac{VS_w(C_{++}+C^s{}_{++})}{B_w}+VC^r{}_{++}\right) = \sum\left[\frac{Tk_{rw}}{B_w\,\mu_w}(\delta P_w-\rho_w g D_z)\right]C_{++}+Q_wC_{++}\ , \qquad \text{[EQ 7.2]}$$

$$\frac{d}{dt}\left(\frac{VS_wC_-}{B_w}\right) = \sum\left[\frac{Tk_{rw}}{B_w\,\mu_w}(\delta P_w-\rho_w g D_z)\right]C_-+Q_wC_-\ , \qquad \text{[EQ 7.3]}$$

$$C_+ = C_- - C_{++} \qquad \text{[EQ 7.4]}$$

where

C_{++} denotes the equivalent molar concentration of divalent cations

C_+ denotes the equivalent molar concentration of monovalent cations

C_- denotes the equivalent molar concentration of anions

$C^s{}_{++}$ denotes the concentration of divalent cations associated with surfactant

$C^r{}_{++}$ denotes the concentration of divalent cations associated with rock

The condition of electroneutrality in the water phase, $C_- = C_{++}+C_+$, is eliminated explicitly, and therefore only two of the three ion concentration transport equations need solving. Hence, when activating the Multi-Component Brine model an additional two conservation equations ([EQ 7.2] and [EQ 7.3]) are solved for the anion and divalent cation concentrations, respectively. As for the standard Brine Tracking option the ion concentrations are updated at the end of a time step after solving the primary equations for the phase flows.

The Multi-Component Brine model is activated using keyword `BRINE` in the `RUNSPEC` section in conjunction with the `ECLMC` keyword. The salts are specified in the `BRINE` keyword by their chemical formula and must be chosen from the list given in Table 7.1. Internally, all salt concentrations are converted and lumped into equivalent molar concentrations of anions, monovalent cations, and divalent cations. The equivalent molar concentration of an ion is the molar concentration multiplied by the ion valence. Thus, a calcium ion (Ca^{2+}) concentration of 0.1 kg moles/m^3 corresponds to an equivalent concentration of 0.2 kg moles/m^3. All reporting of ion concentrations is done in equivalent concentrations.

Table 7.1 List of available salts in the Multi-Component Brine model.

Salt name	Chemical formula	Molecular weight [kg/kg-mole]
Sodium Chloride	NaCl	58.44
Potassium Chloride	KCl	74.55
Calcium Chloride	CaCl2	110.98
Magnesium Chloride	MgCl2	95.21
Sodium Carbonate	Na2CO3	105.99
Potassium Carbonate	K2CO3	138.20
Calcium Carbonate	CaCO3	100.09
Magnesium Carbonate	MgCO3	84.31

Table 7.1 List of available salts in the Multi-Component Brine model.

Salt name	Chemical formula	Molecular weight [kg/kg-mole]
Sodium Sulfate	Na2SO4	142.04
Potassium Sulfate	K2SO4	166.97
Calcium Sulfate	CaSO4	136.14
Magnesium Sulfate	MgSO4	120.37

Cation exchange

In surfactant processes the activity of the surfactant changes with changing salt concentrations. Divalent cations have a much greater effect on surfactant activity, and thereby on phase behavior, than monovalent cations. The cation exchange with rock surfaces will change the flowing concentrations of monovalent and divalent cations and is therefore important to the performance of surfactant floods. The exchange capacity of the rock generally depends on the clay content in the reservoir. In the same way that rock surfaces act as cation exchangers, anionic surfactant micelles can also act as cation exchangers. The cation exchange model accounts for exchange of cations on both rock surfaces and surfactant micelles.

The exchange equilibrium reactions, exemplified here for sodium and calcium, are

$$2Na^{+}(aq) + \overline{Ca^{++}(r)} \leftrightarrow 2\overline{Na^{+}(r)} + Ca^{++}(aq), \qquad \text{[EQ 7.5]}$$

$$2Na^{+}(aq) + \overline{Ca^{++}(s)} \leftrightarrow 2\overline{Na^{+}(s)} + Ca^{++}(aq), \qquad \text{[EQ 7.6]}$$

where the bar indicates that the ion is associated with either the rock surface (r) or surfactant micelles (s) and (aq) stands for aqueous. The exchange equilibria are modeled as

$$\frac{(C_{++})(C_{+}^{r})^{2}}{(C^{r}{}_{++})(C_{+})^{2}} = \beta^{r} q^{r}, \qquad \text{[EQ 7.7]}$$

$$\frac{(C_{++})(C_{+}^{s})^{2}}{(C^{s}{}_{++})(C_{+})^{2}} = \beta^{s} C^{s}, \qquad \text{[EQ 7.8]}$$

where

q^{r} denotes the rock cation exchange capacity

C^{s} denotes the equivalent molar concentration of surfactant

β^{r} denotes the cation exchange constant for rock

β^{s} denotes the cation exchange constant for surfactant

Together with the conditions of electroneutrality on both rock surfaces, $q^{r} = C_{+}^{r} + C_{++}^{r}$, and surfactant micelles, $C^{s} = C_{+}^{s} + C_{++}^{s}$, [EQ 7.7] and [EQ 7.8] are solved to obtain the equilibrium partitioning of monovalent and divalent cations.

The cation exchange model can be activated independently for rock exchanges and surfactant micelle exchanges using keywords IONXROCK and IONXSURF in the PROPS section, respectively. For rock ion exchanges the cation exchange constant, β^r, should be provided using the IONXROCK keyword. The rock exchange capacity must be entered for each grid block using keyword IONROCK in the GRID section. For surfactant micelle exchanges the surfactant option must first be activated using keyword SURFACT in the RUNSPEC section. The cation exchange constant for surfactant, β^s, should be provided using the IONXSURF keyword along with the equivalent molecular weight of the surfactant. Since we solve for massic surfactant concentration, an equivalent molecular weight is required to calculate the number of active sites available on surfactant micelles for ion exchange.

Note The equivalent molecular weight refers to the weight of surfactant per mole of "active sites". A surfactant molecule may have more than one active anionic site. For example, a divalent surfactant with a molecular weight of 500 kg/kg mole has an equivalent weight of 250 kg/kg mole.

When the flowing concentrations of cations are zero, the exchange equilibria are ignored and the rock and surfactant micelle associated concentrations are zero. This corresponds to ignoring the exchange capacity of the rock and surfactant along with the respective electroneutrality conditions. To ensure continuous behavior in the vicinity of zero cation concentrations, we impose a damping function on the exchange capacities. The damping function satisfies $W(0) = 0$ and $W(\varepsilon) = 1$, where ε is a small threshold value. The damping is only imposed for $C_{++} \in [0, \varepsilon]$. The threshold value is set according to

$$\varepsilon = THIONX \times C_{++}^{ref} \qquad C_{++}^{ref} = \max(C_{++}^{inj}, 10^{-2}),$$

where C_{++}^{inj} is the maximum injection concentration and THIONX is a relative threshold concentration, which can be set using the TUNING keyword. The default value for THIONX is 10^{-2}.

The PVT properties of brine

The PVT properties of salt water solution are specified using the PVTWSALT and BDENSITY keywords in the PROPS section.

The PVTWSALT keyword replaces PVTW in brine models, and is used to specify the water properties at different values of the salt concentration C_s. The first record of each table contains two items of data: the reference pressure P_{ref} and the reference salt concentration C_{sref}. The reference salt concentration defines the formation volume factor $B_w(P_{ref}, C_{sref})$ for stock tank water. The reference salt concentration defaults to the minimum salt concentration in the table.

The BDENSITY keyword tabulates the brine surface density against the salt concentration (as specified in the PVTWSALT data). The BDENSITY keyword is optional. If the keyword is not used the brine surface density is taken as the water surface density (input using the DENSITY or GRAVITY keywords) for all salt concentrations.

The fluid properties of the salt water phase are computed from the information entered in the PVTWSALT keyword in different ways depending on whether ECLIPSE 100 or ECLIPSE 300 is used.

ECLIPSE 100

ECLIPSE 100

ECLIPSE 100 first calculates the values of formation volume factor B_w, compressibility C, viscosity μ_w and viscosibility C_v for the water phase as functions of the salt concentration at the previous time step by linear interpolation of C and C_v and the **reciprocals** of B_w and $B_w\mu_w$. Then it calculates B_w and $B_w\mu_w$ at the current pressure P as follows:

$$B_w(P, C_s) = \frac{B_w(P_{ref}, C_s)}{1 + X + (X^2/2)}, \text{ where } X = C(P - P_{ref}) \qquad \text{[EQ 7.9]}$$

$$B_w(P, C_s)\mu_w(P, C_s) = \frac{B_w(P_{ref}, C_s)\mu_w(P_{ref}, C_s)}{1 + Y + (Y^2/2)}, \text{ where } Y = (C - C_v)(P - P_{ref}) \qquad \text{[EQ 7.10]}$$

ECLIPSE 300

ECLIPSE 300

ECLIPSE 300 also calculates B_w using [EQ 7.9]. The water viscosity μ_w is then calculated similarly, using the expression:

$$\mu_w(P, C_s) = \frac{\mu_w(P_{ref}, C_s)}{1 + Y + (Y^2/2)}, \text{ where } \quad Y = -C_v(P - P_{ref}) \qquad \text{[EQ 7.11]}$$

Low salinity option

ECLIPSE 100 only

This option allows you to modify the saturation and relative permeability end points for water and oil phases as a function of the salt concentration as well as the water-oil capillary pressure. This option is activated by the `LOWSALT` keyword in the `RUNSPEC` section.

Given two sets of saturation functions, one for the low salinity and one for the high salinity, the saturation end points are first modified as:

$$\begin{aligned} S_{wco} &= F_1 S^L_{wco} + (1 - F_1) S^H_{wco} \\ S_{wcr} &= F_1 S^L_{wcr} + (1 - F_1) S^H_{wcr} \\ S_{wmax} &= F_1 S^L_{wmax} + (1 - F_1) S^H_{wmax} \\ S_{owcr} &= F_1 S^L_{owcr} + (1 - F_1) S^H_{owcr} \end{aligned} \qquad \text{[EQ 7.12]}$$

where

F_1 is a function of the salt concentration, and correspond to the second column of the `LSALTFNC` keyword.

S_{wco} is the connate water saturation

S_{wcr} is the critical water saturation

S_{wmax} is the maximum water saturation

S_{owcr} is the critical oil saturation in water

and where index H stands for high salinity, and index L stands for low salinity. The F_1 factor is a function of brine concentration and must be provided as a look-up table using the LSALTFNC keyword. Then the relative permeabilities for water and oil, and oil-water capillary pressure are first found by look-up table at the scaled saturations, and then interpolated similarly as:

$$\begin{aligned} k_{rw} &= F_1 k_{rw}^L + (1 - F_1) k_{rw}^H \\ k_{ro} &= F_1 k_{ro}^L + (1 - F_1) k_{ro}^H \\ P_{cow} &= F_2 P_{cow}^L + (1 - F_2) P_{cow}^H \end{aligned} \qquad \text{[EQ 7.13]}$$

where

F_2 is a function of the salt concentration, and corresponds to the third column of the LSALTFNC keyword.

k_{rw} is the water relative permeability

k_{ro} is the oil relative permeability

P_{cow} is oil-water capillary pressure

In the three-phase case, the oil in gas relative permeability at connate water k_{rog}, is interpolated in a same way as in [EQ 7.13] between the scaled low salinity and high salinity curves. The scaling uses a modified endpoints S_{ogcr} and $1 - S_{wco} - S_{gco}$ given by averaging S_{ogcr} and S_{wco} as in [EQ 7.12]. The low salinity model can be used with the two Stone's 3-phase relative permeability models or the default model.

The gas relative permeability k_{rg} is supposed salinity independent even if the maximum gas saturation may increase in the low salinity situation. Only high-salinity tables are used from SGFN, and k_{rg} is taken constant for gas saturations bigger than $S_{gmax}^H = 1 - S_{wco}^H$.

Defining Low and High salinity curves

The usual SATNUM keyword defines the high salinity saturation functions input with keywords SWFN, SOF2, SOF3 or SWOF. The user needs to define additional low salinity saturation functions by entering additional tables within these keywords. The keyword LWSLTNUM in the REGIONS section must then be used to associate a low salinity table number to each grid block.

When the endpoints scaling is used (keywords SWL, SWU, KRW, KRO and so on.) the **high salinity** tables are firstly scaled to these points before being used in the low salinity model described previously. The scaling **does not apply** to the low salinity tables.

When the surfactant option is active, the salinity is assumed to have no effect in the miscible situation. The SATNUM keyword defines the immiscible high salinity saturation functions and LWSLTNUM defines the immiscible low salinity saturation functions. Thus, for a given salinity, the immiscible curves are defined by interpolating between the low salinity and high salinity situations using the procedure described in [EQ 7.12] and [EQ 7.13].

When using the SURFACTW option, the SATNUM and LWSLTNUM keywords correspond to the oil-wet situation. The actual oil-wet curves are defined by interpolating between these two sets of tables using the current salinity. The SURFWNUM keyword defines regions of water-wet high salinities curves and a new set of tables has to be added for the water-wet low salinity situation. The table number for the water-wet low salinity curves should be set by the keyword LSLTWNUM in the REGION section. The water-wet curves used are defined by interpolating between the SURFWNUM and LSLTWNUM tables using the current salinity following [EQ 7.12] and [EQ 7.13].

Note The low salinity option is not supported with directional relative permeabilities option or the HYSTER option. ECLIPSE uses the same low-salinity curves for all directions and for both imbibition and drainage curves. See keyword SATOPTS.

Setting the initial conditions

The initial salt distribution throughout the reservoir is specified in the SOLUTION section. If the initial conditions are determined by **equilibration**, the salt concentration can vary with depth independently in each equilibration region. The SALTVD keyword supplies a table of salt concentrations versus depth for each equilibration region.

When setting the initial conditions by **enumeration** (keywords PRESSURE, SWAT and so on.), the SALT keyword is used to supply the initial salt concentration for each grid block.

Multi-Component Brine

ECLIPSE 100 only

When the Multi-Component Brine model is activated the initial mass concentration of each salt specified in the BRINE keyword in the RUNSPEC section must be entered. For initialization by equilibration the SALTVD keyword supplies a table of salt mass concentrations versus depth with one column of data for each salt. For initialization by enumeration the SALT keyword supplies the initial salt concentration of each salt for every grid block.

For both initialization options the salt mass concentrations are converted into equivalent molar concentrations of anions, monovalent cations, and divalent cations, which are then tracked during simulation.

Wells, rivers and aquifers

The salinity of injected water can be specified independently for each water injector using the WSALT keyword (the argument(s) can be used as a User Defined Argument). The default salt concentration is zero.

ECLIPSE 100 only

Brine economic limits can be specified for production connections, wells and groups by using the keywords CECONT, WECONT and GECONT respectively. It is also possible to set brine cutback limits for wells and groups by using the keywords WCUTBACT and GCUTBACT respectively.

Note The name ‘BRI’ should be used as the name of the tracer for these keywords when setting production and cutback limits for brine.

The salinity of water can be specified for individual river branches and reaches by using the RIVSALT keyword. The default salt concentration is zero.

The initial salinity of any analytic aquifers can be set in the keyword used to specify the aquifer (AQUFET, AQUFETP, AQUCT, AQUFLUX or AQUCHWAT). The default salt concentration for all analytic aquifers is zero.

Tracer related options

ECLIPSE 100 only

As brine is modeled as a tracer it is possible to use some of the tracer related options for brine. It is possible to use a form of velocity and concentration dependent dispersion using the keywords TRDIS, DISPERSE and DIAGDISP. The tracer dispersion option is activated by specifying DISPDIMS in the RUNSPEC section.

For a brine tracer it is possible to replace the tracer concentration dispersion dependence by water density dispersion dependence by using the keyword DSPDEINT.

The Polymer flood option

ECLIPSE 100 only

The Polymer flood model (see "The Polymer Flood Model" on page 777), activated using the option POLYMER in the RUNSPEC section, allows you to investigate the effect of varying brine concentrations on the efficiency of the polymer flood. The study of the interaction of brine with the polymer solution is, however, optional within the model.

For models which have the polymer and brine options activated, the viscosity of a fully mixed polymer solution needs to be entered as a function of the salt concentration. The keywords SALTNODE and PLYVISCS should be used to enter two-dimensional tables for the viscosity of the solution. The maximum polymer and salt concentrations to be used in calculating the effective fluid component viscosities are entered using the keyword PLYMAX.

The polymer adsorption can be set to depend on the brine concentration and on the rock permeability using the ADSORP keyword with POLYMER as an argument

The concentration of polymer and salt for the injection wells can be specified using the WPOLYMER keyword. The default polymer and salt concentrations are zero.

Multi-Component Brine and Polymer

ECLIPSE 100 only

When the Multi-Component Brine model is used in conjunction with the Polymer option, an **effective salinity** is calculated with respect to polymer properties. This effective salinity can be used in the polymer adsorption isotherms, and is computed according to

$$C_{SEP} = C_{-} - (\alpha^{p} - 1)C_{++} \ , \qquad \text{[EQ 7.14]}$$

where α^{p} is an input parameter supplied using the PLYESAL keyword in the PROPS section. The salinity dependent adsorption isotherms can then be specified in the ADSORP keyword with POLYMER as an argument.

The Surfactant flood option and Multi-Component Brine

ECLIPSE 100 only

The multi-component brine option can be used to make the surfactant adsorption dependent on the salinity. Based on ion transport and exchanges an **effective salinity** is computed, and then used in the surfactant adsorption isotherm. The effective salinity is an empirical quantity incorporating the association between the surfactant and divalent cations, and calculated according to

$$C_{SE} = \frac{C_{-}}{1 - \alpha^s X^s}, \qquad X^s = \frac{C^s_{++}}{C^s}, \qquad \text{[EQ 7.15]}$$

where

C_{SE} denotes the surfactant effective salinity

X^s denotes the fraction of surfactant sites occupied by divalent cations

α^s denotes a user input effective salinity parameter ($\alpha^s \in [0, 1]$)(SURFESAL)

The effective salinity calculation is activated using keyword SURFESAL in the PROPS section, which also specifies the effective salinity parameter, α^s, in [EQ 7.15]. This salinity is then used in the surfactant adsorption isotherms specified in the ADSORP keyword with SURFACT as an argument.

Solving the Conservation Equations

ECLIPSE 100 only

For the Multi-Component Brine Model the ion components are solved as active tracers. The ion concentrations determine the effective salinities, which in turn have a strong impact on surfactant-water-oil phase behavior and surfactant/polymer adsorption, and hence affect the phase flows. By default tracers are solved only at the end of a time step once the equations for phases have converged. In this way the salinity is treated explicitly in the calculations which may lead to numerical instability if the time steps are large and the dependence on salinity is strong. For this reason active tracers can now be solved in each Newton iteration so that the tracer concentrations are updated simultaneously with the phase equations.

When the ECLMC keyword is present in the RUNSPEC section the Multi-Component Brine equations are solved in each Newton iteration. The standard behavior of solving the Brine equations at the end of a time step can be restored by setting the 126th item of the OPTIONS keyword to 2. When solved in each Newton iteration the Brine concentration used when calculating effective salinities is computed as a weighted average of the concentration at time t and the concentration at time $t + \Delta t$ corresponding to the current iteration:

$$C = (1 - \theta)C(t) + \theta C(t + \Delta t), \qquad \text{[EQ 7.16]}$$

where θ is a weighting factor. $\theta = 0$ corresponds to the standard, explicit treatment of Brine concentration, and $\theta = 1$ corresponds to a fully coupled approach. The value of θ can be set in Item 13 in the second record of the TUNING keyword (Default value: 1).

Summary of keywords

RUNSPEC

The brine RUNSPEC keywords are:

Table 7.2 Brine RUNSPEC keywords

E100	E300	Keyword	Description
x	x	BRINE	Activates the Brine Tracking option
x		LOWSALT	Activates the low salinity option
x		ECLMC	Activates multi-component brine

GRID

The brine GRID keywords are:

Table 7.3 Brine GRID keywords

E100	E300	Keyword	Description
x		IONROCK	Specify the rock ion exchange capacity
x		RPTGRID	Controls output from the GRID section to the print file: Use mnemonic IONROCK to output rock ion exchange capacity.

PROPS

The brine PROPS keywords are:

Table 7.4 Brine PROPS keywords

E100	E300	Keyword	Description
x	x	BDENSITY	Specify the brine density
x		LSALTFNC	Specify the low salinity fraction as function of the salt concentration in the grid block.
x	x	PVTWSALT	Specify water PVT functions with salt present
x	x	RPTPROPS	Controls output from the PROPS section to the print file: Use mnemonics BDENSITY and PVTWSALT to output brine properties to the print file.
x		IONXROCK	Activate cation exchange on rock surfaces and specify cation exchange constant for rock

Table 7.4 Brine PROPS keywords

E100	E300	Keyword	Description
x		IONXSURF	Activate cation exchange on surfactant micelles and specify cation exchange constant for surfactant and equivalent molecular weight of surfactant
x		SURFESAL	Specify parameter for effective salinity calculation with respect to surfactant phase behavior
x		PLYESAL	Specify parameter for effective salinity calculation with respect to polymer properties

REGIONS

The brine REGIONS keywords are:

Table 7.5 Brine REGIONS keywords

E100	E300	Keyword	Description
x		SATNUM	Specify the high salinity table number for each block. For Surfactant option, this correspond to the immiscible high salinity oil-wet tables.
x		LWSLTNUM	Specify the low salinity table number for each block. For Surfactant option, this correspond to the immiscible low salinity oil-wet tables.
x		SURFWNUM	For Surfactant option, it specifies the immiscible high salinity water-wet tables.
x		LSLTWNUM	For Surfactant option, it specifies the immiscible low salinity water-wet tables.

SOLUTION

The brine SOLUTION keywords are:

Table 7.6 Brine SOLUTION keywords

E100	E300	Keyword	Description
x	x	RPTSOL	Controls output from the SOLUTION section to the print file: Use mnemonic SALT to output initial grid block salt concentration values. Use mnemonic SALTVD to output tables of initial salt concentration. For the Multi-Component Brine model use SALTVD to output tables of initial anion and divalent cation concentrations. (ECLIPSE 100 only).
x	x	SALT	Specify initial salt concentration(s)
x	x	SALTVD	Specify salt concentration(s) versus depth for equilibration

SUMMARY

The brine SUMMARY outputs are:

Table 7.7 Brine summary quantities

E100	E300	Field	Group	Well	Connection	Region	Block	Description
x	x				CSFR			Salt Flow Rate (+ or -)
x	x	FSPR	GSPR	WSPR	CSPR			Salt Production Rate
x	x	FSPT	GSPT	WSPT	CSPT			Salt Production Total
x	x	FSIR	GSIR	WSIR	CSIR			Salt Injection Rate
x	x	FSIT	GSIT	WSIT	CSIT			Salt Injection Total
x	x						BSCN	Salt Cell Concentration
x	x	FSPC	GSPC	WSPC	CSPC			Salt Production Concentration
x	x	FSIC	GSIC	WSIC	CSIC			Salt Injection Concentration
x	x	FSIP				RSIP (not E300)	BSIP (not E300)	Salt In Place
x						RSFT		Salt Flow Total
x							BEWV_SAL	Effective brine viscosity

ECLIPSE 100 only

The Multi-Component Brine `SUMMARY` options are:

Table 7.8 Multi-Component Brine summary quantities

Field	Group	Well	Connection	Region	Block	Information
FTPRANI	GTPRANI	WTPRANI				Anion Prod. Rate
FTPTANI	GTPTANI	WTPTANI	CTPTANI			Anion Prod. Total
FTIRANI	GTIRANI	WTIRANI				Anion Inj. Rate
FTITANI	GTITANI	WTITANI	CTITANI			Anion Inj. Total
					BTCNFANI	Anion Flowing Concentration
FTPRCAT	GTPRCAT	WTPRCAT				Cation Prod. Rate
FTPTCAT	GTPTCAT	WTPTCAT	CTPTCAT			Cation Prod. Total
FTIRCAT	GTIRCAT	WTIRCAT				Cation Inj. Rate
FTITCAT	GTITCAT	WTITCAT	CTITCAT			Cation Inj. Total
					BTCNFCAT	Cation Flowing Concentration
					BTRADCAT	Cation Rock Associated Conc.
					BTSADCAT	Cation Surfactant Associated Conc.
					BESALSUR	Effective Salinity for Surfactant
					BESALPLY	Effective Salinity for Polymer

SCHEDULE

The SCHEDULE keywords are:

Table 7.9 SCHEDULE keywords

E100	E300	Keyword	Description
x		RIVSALT	Sets the salt concentrations in the upstream flow of river branches
x	x	RPTSCHED	Controls output from the SCHEDULE section to the print file: Use SALT to output grid block salt concentration values. ECLIPSE 100 only: - FIPSALT outputs salt fluid in place reports - ANIONS and CATIONS output grid block anion and divalent cations concentrations for the Multi-Component Brine model. - ESALSUR and ESALPLY output grid block effective salinities for surfactant and polymer, respectively, in the Multi-Component Brine model.
		RPTRST	Mnemonic VISC allows to output extra viscosity when or Brine is active: - EWV_SAL: effective water viscosity due to salt concentration
x	x	WSALT	Set salt concentration for injection wells (the argument can be set as a User Defined Argument)
x		TUNING	Item 12 is the second record sets THIONX, the relative threshold for damping in the ion exchange calculation.

Examples

This example shows only sections where data related to the brine option are mentioned.

RUNSPEC Example

RUNSPEC section example:

```
RUNSPEC
TITLE
Example Brine Model
-- Phases present
OIL
WATER
BRINE
FIELD
DIMENS
15  1  10  /
START
1 'JAN' 1983 /
-- Well dimensions
WELLDIMS
2  10  1  2  /
-- Aquifer dimensions
AQUDIMS
0  0  1  36  2  10  /
```

PROPS Example

PROPS section example

```
PROPS
-- Rock properties
ROCK
4000.0  0.000008  /
-- Fluid densities at surface conditions
DENSITY
45  64  0.09  /
```

```
-- PVT data for oil
PVDO
 400  1.012  1.160
 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 1.190
/
-- PVT data for water with salt present
PVTWSALT
3000.0 /
 0.0  1.000  3.0E-6  1.00  1.0E-6
10.0  0.8    3.0E-6  1.00  1.0E-6  /
-- Saturation functions
SWFN
0.20  0.0  0.0
0.40  0.1  0.0
0.80  0.5  0.0
1.0   1.0  0.0
/
SOF2
0.2  0.0
0.4  0.12
0.6  0.35
0.7  0.68
0.8  1.0
/
```

SOLUTION Example

`SOLUTION` section example

```
SOLUTION
-- Equilibration data
EQUIL
5270  4000.0  5270.0  0.0  400.0  0.0  0  0  5  /
-- Salt concentration versus depth table
SALTVD
 5000.0  9.0
 5500.0  9.0 /
-- Aquifer data
AQUFETP
 1  5400  1*  2.0E9  3.0E-5  540.96  1  10.0  /
/
AQUCT
 2  5400.0  1*  20.0  0.10  3.0E-5  2400.0  140.0  4.8  1  1  9.5  /
/
```

```
AQUANCON
 1  15  15  1  1  1   5  'I+'  /
 2  15  15  1  1  6  10  'I+'  /
/
```

SUMMARY Example

SUMMARY section example

```
SUMMARY
-- Field salt production rates and totals
FSPR
FSPT
-- Field salt injection rates and totals
FSIR
FSIT
-- Field salt production and injection concentrations
FSPC
FSIC
-- Field salt in place
FSIP
-- Group salt production and injection rates
GSPR
'G' /
GSIR
'G' /
-- Well salt production and injection rates
WSPR
'PRODUCER' /
WSIR
'INJECTOR' /
-- Block salt concentration
BSCN
1 1 1 /
/
```

SCHEDULE Example

SCHEDULE section example

```
SCHEDULE
-- Well specification data
WELSPECS
    'PRODUCER'  'G'   1  1  5200  'OIL'  /
    'INJECTOR'  'G'  14  1  5400  'WAT'  /
/
```

```
COMPDAT
    'PRODUCER'    1  1  1  10  'OPEN'  1  -1  0.5  /
    'INJECTOR'   14  1  5  10  'OPEN'  1  -1  0.5  /
/
WCONPROD
    'PRODUCER'  'OPEN'  'ORAT'  200.0  4*  1500.0  /
/
GCONINJE
    'FIELD'  'WAT'  'VREP'  3*  0.7  /
/
WCONINJE
    'INJECTOR'  'WAT'  'OPEN'  'GRUP'  /
/
-- Salt concentration for injection well
WSALT
  'INJECTOR'  1.0  /
/
-- Time-step data
TSTEP
1.0 9.0 90.0 100.0 100.0 100.0 100.0 250.0 250.0 1000.0 1000.0 /
END
```

CO_2 Injection: Geological Storage, EOR and ECBM

Chapter 8

Introduction

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL

One of the major issues of CO_2 injection problems is the chemical reactivity between the supercritical fluid (referred to as the gas phase in this document) and the liquid phase. There are several facilities available in ECLIPSE 300, depending on the site and operational conditions that need to be modeled.

Geological Storage

In Aquifers

- Using the `CO2STORE` option, two phases are considered: a CO_2-rich phase (labeled gas phase), and an H_2O-rich phase (labeled liquid phase).

  This option gives accurate mutual solubilities of CO_2 in water (xCO_2) and water in the CO_2-rich phase (yH_2O). Salts are described as components of the liquid phase.

- The `GASWAT` option provides a gas/aqueous phase equilibrium method. The liquid mole fraction x CO_2 is accurately predicted; however, the gas phase mole fraction yH_2O is less accurate. The gas composition is not restricted to CO_2/H_2O, as other gases and their solubility in water can be included.

In Oil Depleted Reservoirs and EOR (Enhanced oil recovery)

- The option best suited for this case is `CO2SOL`. Water is not considered in the oil or gas phase, but the solubility of CO_2 in the water phase is accounted for. The gas/oil composition is not restricted to CO_2 but only CO_2 is considered soluble in water.

In Gas Depleted Reservoirs

- This case can be modeled using the GASWAT option by adding gas components, such as CH_4 and H_2S for example.

ECBM

- The COAL option provides a diffusive flow and sorption model for enhanced coal bed methane recovery (ECBM). A full description of this option is provided in "The Coal Bed Methane model" on page 161.

Since the CO2STORE, CO2SOL and GASWAT options estimate the fluid properties differently, they can not be used together.

CO2STORE

The mutual solubilities of CO_2 and H_2O are calculated to match experimental data for typical CO_2 storage conditions: typically 12-250 degrees C and up to 600 bars. They are calculated following the procedure given by Spycher and Pruess [Ref. 110] and [Ref. 132], based on fugacity equilibration between water and a CO_2-phase. Water fugacity is obtained by Henry's law, while CO_2 fugacity is calculated using a modified Redlich-Kwong equation of state.

See "CO_2 Storage" on page 135.

- The salts are assumed to stay in the liquid phase.
- The gas density is obtained by an accurately tuned cubic equation of state.
- The brine density is first approximated by the pure water density and then corrected for salt and CO_2 effects by Ezrokhi's method [Ref. 111].
- The CO_2 gas viscosity is calculated from [Ref. 113] and [Ref. 114].

Using CO2STORE

RUNSPEC section

The option switched on using the CO2STORE keyword. The number of components is specified by the COMPS keyword. Five components are currently allowed: CO_2, H_2O, NaCl, $CaCl_2$ and $CACO_3$.

PROPS section

Fluid component properties are assigned internally based on the names given in the CNAMES keyword (labeled 'CO2', 'H2O', 'NACL', 'CACL2', ’CACO3’).

The water relative permeability is input using the WSF keyword, while the gas relative permeability and the gas-water capillary pressure are entered using the GSF keyword.

Initial state constructed by equilibration

If an initial gas-water contact exists within the reservoir you may specify either the vapor or the aqueous phase composition at the contact; either ZMFVD or COMPVD can be used.

If only a single phase is present the composition may be entered using ZI or with respect to depth using ZMFVD.

SOLUTION section

The EQUIL keyword is used for initial state equilibration.

- Initial state set by enumeration
- Liquid, gas and total compositions are specified explicitly with keywords XMF, YMF and ZMF respectively.

SCHEDULE section

The composition of the injected gas can be specified using the keyword WINJGAS.

Mole fractions xCO_2 and yH_2O are output by writing XMF and YMF in RPTSCHED.

Note See the Note about salt precipitation in "Using the CO2 storage model" on page 139.

CO2SOL

CO2SOL is a three-phase compositional option for simulating CO_2 flooding in oil reservoirs (EOR, storage in oil depleted reservoirs). Water is not allowed to dissolve in the gas or oil phases, CO_2 can be present in the three phases, and other components in the oil and gas phases.

See "CO_2/H_2S Solution in the Aqueous Phase" on page 143.

CO_2 partitioning between the oil and gas phases is calculated by a fugacity equilibration method. A cubic equation of state is used to model oil and gas phase densities and fugacities.

The amount of CO_2 dissolved in water, and other aqueous phase properties, are computed using solubility data entered by the user with either the SOLUBILI, SOLUBILS or SOLUBILT keywords or defaulted following Chang et al. [Ref. 4]. When using SOLUBILI or SOLUBILT the salting-out effect is taken into account by correcting solubility in pure water with the Chang et al. correlation for brine [Ref. 4]. The SOLUBILS keyword can be used to describe a more complex salt dependency using the BRINE tracking. The SOLUBILT keyword can be used to describe a more complex temperature dependency. To obtain details on the solubility data used in a simulation the SOLU mnemonic may be specified in the RPTPROPS keyword.

Molecular diffusion between gases can be modeled using the DIFFUSE option. See also "Diffusion" on page 211.

It is possible to set initial dissolved CO_2 as a function of depth using the RSWVD or RSW keywords. This initial solution honors this value unless it exceeds the saturated CO_2 concentration in which case the saturated value is used.

Notes

1 The initial CO_2 concentration may not be in exact equilibrium with the hydrocarbon phase. In such a case the initial flash slightly modifies the input values.

2 The first line of the solubility table defined using the SOLUBILI keyword represents water at standard conditions, when the dissolved CO_2 amount is at a minimum. The simulator does not assume that the amount of CO_2 in water at the lowest pressure is zero; if you want to specify it as such then you should set it to zero in the first line of the table.

3 Instantaneous equilibrium is assumed between the oil, gas and water phases.

Using CO2SOL

RUNSPEC section

This option is switched on using the CO2SOL keyword. Unlike in the CO2STORE option, water is not defined as a component in COMPS and CNAMES, as it is absent in the oil and gas phase.

The DIFFUSE option allows for the molecular diffusion of gases. See also"The GASWAT Option" on page 405.

The Brine tracking option is activated by the BRINE keyword.

PROPS section

The gas relative permeability and oil-gas capillary pressure are input using the SGFN keyword, while the water relative permeability and the gas-water capillary pressure are entered using the SWFN keyword. Oil relative permeabilities are defined with the SOF3 keyword.

Fluid component properties can be generated from the PVTi database. The component 'CO2' must be present in CNAMES.

The initial dissolved CO_2 may be specified as a function of depth using the RSWVD keyword.

The SOLUBILI keyword allows the user to enter a solubility data set as a function of reservoir pressure. The SOLUBILT keyword is similar to SOLUBILI but allows the user to enter a temperature dependent table. For both keywords, the table will be automatically corrected for the salinity effect following Chang et al. [Ref. 4] if either the SALINITY or SALINITR keyword has been used. In this case, the temperature used in that correction will be taken from the RTEMP keyword in the case of SOLUBILI and from table itself for SOLUBILT. The density of water at surface condition can be adjusted using the DENSITY or GRAVITY keywords.

The SOLUBILS keyword is similar to SOLUBILI but the user enters a salt concentration dependent table. If the Brine option is active, SOLUBILS supersedes PVTWSALT and BDENSITY and the salt concentration is taken from the brine tracer.

If none of the SOLUBILI, SOLUBILS or SOLUBILT keywords are used the solubility data is defaulted as detailed in the description of SOLUBILI.

The coefficients for molecular diffusion must be set with DIFFCGAS if the DIFFUSE option is used.

If CO_2 appears as liquid, it will be labelled as oil phase. In such a case, make sure that the SOF3 keyword (relative permeability for CO_2 liquid) is appropriate.

Initial state constructed by equilibration

If an initial oil-gas contact exists within the reservoir you may specify either the vapor or the liquid phase composition at the contact, either ZMFVD or COMPVD can be used.

If only a single phase is present the composition may be entered using ZI or with respect to depth using ZMFVD.

SOLUTION section

The EQUIL keyword is used for equilibration.

- Initial state set by enumeration
- The oil phase composition is set with XMF, the gas phase composition with YMF, water composition with AMF and the total oil and gas composition with ZMF. Water content in CO_2 can be defined by RSW or RSWVD.

SCHEDULE section

The composition of the injected gas can be specified using the keyword WINJGAS.

The aqueous mole fractions a CO_2 and aH_2O can be output by adding the mnemonic AMF in RPTSCHED.

CO2SOL/H2SSOL/GASSOL

CO2SOL/H2SSOL is a three-phase compositional option for simulating CO_2 and H_2S flooding in oil reservoirs (EOR, storage in oil depleted reservoirs).

GASSOL is a three-phase compositional option for simulating multi-component gas flooding in oil reservoirs (EOR, storage in oil depleted reservoirs). Water is not allowed to dissolve in the gas or oil phases; any gas component can be present in all three phases; all other components can be present in the oil and gas phases.

See "CO_2/H_2S Solution in the Aqueous Phase" on page 143 and "GASSOL Solution in the Aqueous Phase" on page 401.

Multi-component gas partitioning between the oil and gas phases is calculated by a fugacity equilibration method. A cubic equation of state is used to model oil and gas phase densities and fugacities.

The amount of multi-component gas dissolved in water, and other aqueous phase properties, is computed using solubility data entered by the user using SOLUCO2H, SOLUH2SC or SOLUAQA. This solubility data may also be defaulted. To obtain details regarding the solubility data used in a simulation, the SOLU mnemonic may be specified using the RPTPROPS keyword.

Molecular diffusion between gases can be modeled using the DIFFUSE option. See also "Diffusion" on page 211.

It is possible to set the initial concentrations of dissolved gases as a function of depth using the RSWVD or RSW keywords. The initial solution honors these values unless they exceed the saturated multi-component gas concentration, in which case the saturated value is used.

Notes

1. The initial multi-compound gas concentration may not be in exact equilibrium with the hydrocarbon phase. In such a case, the initial flash slightly modifies the input values.
2. Instantaneous equilibrium is assumed between the oil, gas and water phases.

Using GASSOL

RUNSPEC section

This option is switched on using the GASSOL keyword. Unlike in the CO2STORE option, water is not defined as a component in COMPS and CNAMES, as it is absent in the oil and gas phases.

The DIFFUSE option allows for the molecular diffusion of gases. See also "The GASWAT Option" on page 405.

PROPS section

The gas relative permeability and oil-gas capillary pressure are input using the SGFN keyword, while the water relative permeability and the gas-water capillary pressure are entered using the SWFN keyword. Oil relative permeabilities are defined with the SOF3 keyword.

Fluid component properties can be generated from the PVTi database. The dissolved gas components must be present in CNAMES.

The initial concentrations of dissolved gases may be specified as a function of depth using the RSWVD keyword.

The SOLUAQA keyword allows the user to enter a solubility data set as a function of reservoir pressure. The reservoir temperature is taken from the RTEMP keyword. Note that the solubility data may be defaulted for any given soluble component. In such a case, a default set of pressures and corresponding solubilities is generated. If the solubilities for all soluble components are defaulted, then each table comprises twenty rows of data. If the solubilities for some (but not all) soluble components are defaulted, then the number of rows in the defaulted solubility table(s) equals the maximum number of rows from the remaining input solubility tables. Since the default solubilities depend strongly on the data entered in the PROPS section, this may lead to unrealistic values if the component data is not accurate enough. After computing the dissolving hydrocarbons' mole fraction in the aqueous phase using the phase equilibrium algorithm mentioned previously, the water density and the water viscosity μ_{sat} can be calculated using the Ezrokhi data specified using DENAQA and VISCAQA, respectively.

The coefficients for molecular diffusion must be set with DIFFCGAS if the DIFFUSE option is used.

Initial state constructed by equilibration

If an initial oil-gas contact exists within the reservoir, the user may specify either the vapor or the liquid phase composition at the contact, using ZMFVD or COMPVD.

If only a single phase is present, the composition may be entered using ZI or (with respect to depth) using ZMFVD.

SOLUTION section

The EQUIL keyword is used for equilibration.

- Initial state set by enumeration.
- The oil phase composition is set with XMF, the gas phase composition with YMF, water composition with AMF and the total oil and gas composition with ZMF. Aqueous phase content can be defined by RSW or RSWVD.

SCHEDULE section

The composition of the injected gas can be specified using the keyword WINJGAS.

The aqueous mole fractions of dissolved gas components can be output by adding the mnemonic AMF in RPTSCHED.

GASWAT

The GASWAT option uses a modified Peng-Robinson equation of state in order to model gas/aqueous phase equilibriums.

GASWAT allows a multi-component gas phase. CO_2, N_2, and H_2S solubilities in water are treated with the Peng Robinson equation of state (EOS) modified by Soreide and Whitson. However, solubilities of other gases like methane are treated by the original Peng Robinson EOS by default. "The GASWAT Option" on page 405 provides an exhaustive description of this feature.

Molecular diffusion between gases can be modeled using the DIFFUSE option. See "Diffusion" on page 211.

Using GASWAT for CO_2 storage issues

RUNSPEC section

This option is switched on using the GASWAT keyword. The number of components is specified by the COMPS keyword. Water has to be defined as a component; salts are not components. The WATER keyword should not be present, as H_2O is a component specified in COMPS.

The Diffusion option allows for molecular diffusion of gases.

PROPS section

The water relative permeability is input using the WSF keyword, while the gas relative permeability and the gas-water capillary pressure are entered using the GSF keyword.

Salinity can be set up by the SALINITY keyword or on a regional basis with the SALINITR keyword

The coefficients for molecular diffusion must be set with DIFFCGAS if the Diffusion option is used.

Initial state constructed by equilibration:

- If an initial gas-water contact exists within the reservoir you may specify either the vapor or the aqueous phase composition at the contact, either ZMFVD or COMPVD can be used.
- If only a single phase is present the composition may be entered using ZI or with respect to depth using ZMFVD.

SOLUTION section

The EQUIL keyword is used for initial state equilibration.

- Initial state set by enumeration
- Liquid phase composition can be set with XMF, the gas phase composition with YMF and the total composition with ZMF.

SCHEDULE section

The composition of the gas injected can be specified using the keyword WINJGAS.

Mole fractions xCO_2 and yH_2O can be output by writing mnemonics XMF and YMF in RPTSCHED.

Boundary conditions

Several facilities are available.

- Numerical aquifers contain only a water phase. For CO2STORE and GASWAT cases, make sure that no supercritical CO_2 (CO_2 rich phase being labeled as gas phase) reaches the boundary. Likewise for the liquid CO_2 for a CO2SOL run.
- For CO2STORE and GASWAT cases, analytic aquifers can be used to let the CO_2 rich phase leave the reservoir. For this, use AQUCHWAT, AQUCHGAS and AQSTREAM and ensure that the composition of the gas aquifer specified by AQUCHGAS is as close as the composition of the gas reaching the boundary.

Note on conversions for salinity units

As the various CO_2 options use different frameworks within ECLIPSE, units for salinity input differ.

Salt concentrations in XMF and YMF must be entered in mole fraction. The SALINITY keyword unit is molality (mol/kg of solvent). Salinity in SOLUBILS must be expressed in kg/m^3 of brine.

Conversions between these different units are described below.

Molality to mole fraction

Mole fraction of component i in the aqueous phase: $x_i = molality_i \times \frac{M_{solvent}}{1000} \times x_{solvent}$

where:

- mole fraction of solvent in the aqueous phase: $x_{solvent} = \frac{n_{solvent}}{n_{total}} = \frac{n_{solvent}}{n_{solvent} + \sum_{i \neq solvent} n_i}$
- molality of component i : $molality_i = \frac{n_i}{m_{solvent}}$ [mol/kg],
- $M_{solvent}$ is the molecular weight of solvent [g/mol]

Thus:

- $molality_{solvent} = 1000 / M_{solvent}$ [mol/kg],
- In most of the cases, the solvent is H_2O. As M_{H_2O}=18.014 g/mol, $molality_{H_2O}$= 55.5 mol/kg.

Concentration (mg/L) to mole fraction

Chemistry analysis gives usually concentration in mg/L (=kg/m^3) and the initial composition of the brine/water in ECLIPSE must be entered in mole fraction.

Concentrations in mg/L can be converted to mole fraction through: $x_i = \frac{\left(\frac{c_i}{M_i}\right)}{\sum_j \left(\frac{c_j}{M_j}\right)} = \frac{[i]}{\sum_j [j]}$

where [i] denotes the molarity of component i in mol/L, c_i the concentration in mg/L and M_i the molecular weights in mg/mol.

With a standard water analysis at hand, a few additional considerations are required to process the conversion by this formula, as highlighted in the following example:

Example

A brine sample has been analyzed and the results are the following:

Table 8.1 Brine sample analysis

Species	Concentration (mg/L)	Species	Concentration (mg/L)
Na^+	90100	HCO_3^-	45
Ca^{2+}	1760	I^-	1
Mg^{2+}	2230	HS^-	nb
K^+	470	CO_3^{2-}	0
NH_4^{2+}	0	OH^-	0
Fe^{2+}	0	TDS (Total Dissolved Solid)	250000
Cl^-	141470	pH	5.3
SO_4^{2-}	6600	Water Density at reservoir conditions (kg/m^3)	1164
Br^-	64		

ECLIPSE CO2STORE

Targeted ion concentrations (for ECLIPSE CO2STORE option):

$c_{Na^+} = 90.11$ g/L

$c_{Ca^{2+}} = 1.76$ g/L

$c_{Cl^-} = 141.47$ g/L

For one liter of brine:

m_{H_2O} = reservoir brine density - TDS = 1164 - 250 = 914 g

$$n_{H_2O} = \frac{m_{H_2O}}{M_{H_2O}} = \frac{914(836)}{18.014} = 50.738 \text{ mol}$$

$$n_{Na^+} = \frac{90.11}{23} = 3.919 \text{ mol}$$

$$n_{Ca^{2+}} = \frac{1.76}{40} = 0.044 \text{ mol}$$

$$n_{Cl^-} = \frac{141.47}{35} = 4.042 \text{ mol}$$

For $Na^+ + Cl^- \Leftrightarrow NaCl$:

	Sodium (Na)	Chlorine (Cl)	Sodium Chloride (NaCl)
Initial	n_{Na^+0}	n_{Cl^-0}	0
At t time n_t	$n_{Na^+0} - \xi$	$n_{Cl^-0} - \zeta$	ξ
Final n_f	0	$n_{Cl^-f} = n_{Cl^-0} - n_{Na^+0}$	n_{Na^+0}

where ζ is the reaction advancement.

Here $n_{NaCl(aq)} = n_{Na^+0}$ because Na^+ is the limiting factor.

For $Ca^+ + 2Cl^- \Leftrightarrow CaCl_2$:

	Calcium (Ca)	Chlorine (Cl)	Calcium Chloride ($CaCl_2$)
Initial	$n_{Ca^{2+}0}$	n_{Cl^-f}	0
At t time n_t	$n_{Ca^{2+}0} - \psi$	$n_{Cl^-f} - 2\psi$	ψ
Final n_f	0	$n_{Cl^-f} - 2n_{Ca^{2+}0}$	$n_{Ca^{2+}0}$

where ψ is the reaction advancement.

Here $n_{CaCl_2(aq)} = n_{Ca^{2+}0}$ because Ca^{2+} is the limiting factor.

The ionic species are reconciled into neutral component in ECLIPSE. The neutral components in this case are NaCl and $CaCl_2$. These are formed from the ionic species in order of increasing solubility. Starting with the salt with lowest solubility, in this case NaCl, the limiting ionic species is Na^+, that is, 3.92 moles of NaCl can be formed, leaving (4.00 - 3.92 =) 0.08 mol/L of Cl^- for the formation of 0.04 mol/L of $CaCl_2$. It should be noted that the water analysis in this case is consistent with the Ca^{2+} concentration.

$$n_{total} = n_{NaCl(aq)} + n_{CaCl_2(aq)} + n_{H_2O} = 54.701 \text{ mol}$$

$$x_{NaCl} = \frac{n_{NaCl(aq)}}{n_{total}} = \frac{3.919}{54.701} = 0.0716$$

$$x_{CaCl_2} = \frac{n_{CaCl_2(aq)}}{n_{total}} = \frac{0.044}{54.701} = 0.0008$$

$$x_{H_2O} = \frac{n_{H_2O}}{n_{total}} = \frac{46.408}{54.701} = 0.9276$$

Note In this example, the molality of salt you have to input for the keyword `SALINITY` (as used in the `CO2SOL` and `GASWAT` options) is:

for one liter of solution, $molality_{salt} = \frac{\sum_{salt} n_i}{mass_{H_2O}} = \frac{3.963}{0.916} = 4.34$ mol/kg

Molality to mass per volume unity (kg/m^3)

$$\text{Mass per volume unity} = \frac{d}{B} \times \frac{\text{molality} \times 58.440 \times 10^{-3}}{1 + \text{molality} \times 58.440 \times 10^{-3}}$$

where

B = brine volume factor

d = surface brine density

CO_2 Storage

Chapter 9

Introduction

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL

In ECLIPSE 300 there are several options available in order to study the storage of CO_2 under various conditions. This chapter describes the CO2STORE option, for CO_2-H_2O systems with salts. In addition the following options are available.

- The CO2SOL option, typically used to study CO_2 storage in oil reservoirs - water is not allowed in the oil/gas phase.
- The GASWAT option, general equation of state for gas-water systems.
- The COAL option for coal bed methane reservoirs.

Since the CO2STORE, CO2SOL and the GASWAT options estimate the fluid properties differently, they can not be used together.

Note For CO2STORE, The “EOR and CO2 STORAGE” license is required.

Note In thermal mode the CO2STORE option can be used with any number of components and include an liquid oil phase, this is described in "Thermal CO2STORE" on page 140”.

For the CO2STORE option three phases are considered, a CO_2 rich phase, an H_2O rich phase and a solid phase. The CO_2 rich phase is labeled the gas phase while the H_2O rich phase is labeled the water phase (liquid phase). The mutual solubilities of CO_2 and H_2O are calculated to match experimental data for CO_2 -H_2O systems under typical CO_2 storage conditions: 12-100$°C$ and up to 600 bar. The model is also applicable at elevated temperatures as detailed in [Ref. 132], up to 250 $°C$. Fluid components are specified with the COMPS keyword. Allowed component names are currently CO2, H2O and the salts NACL, CACL2 and CACO3.

Table 9.1 Components that can be used with the CO2STORE option.

Component name	Phases
CO2	Water/Gas
H2O	Water/Gas
NACL	Water/Solid
CACL2	Water/Solid
CACO3	Water/Solid

The option is similar to the GASWAT option, but the salt components of the water or solid phase are tracked as part of the fluid system. Also the partitioning of water in the gas phase has been matched accurately. Fluid properties are assigned internally based on the names; this means that a minimum of user data is needed in order to supply the fluid properties.

Phase splitting

The partitioning of CO_2 and H_2O in the liquid and gas phase follows the procedure given by Spycher and Preuss, [Ref. 110] and [Ref. 132]. The salts are assumed to stay in the liquid phase unless the SOLID option is activated.

With the SOLID option the components NaCl, $CaCl_2$ and $CaCO_3$ can be present in both the aqueous phase and the solid phase according to the following reactions equilibrium:

$$H_2O \;\; <\!\!-\!\!> \;\; H^+ + OH^-$$
$$CO_2 + H_2O \;\; <\!\!-\!\!> \;\; HCO_3^- + H^+$$
$$HCO_3^- \;\; <\!\!-\!\!> \;\; CO_3^{--} + H^+$$
$$CaCl^+ \;\; <\!\!-\!\!> \;\; Ca^{2+} + Cl^-$$
$$NaCl(s) \;\; <\!\!-\!\!> \;\; Na^+ + Cl^-$$
$$CaCl_2(s) \;\; <\!\!-\!\!> \;\; CaCl^+ + Cl^-$$
$$CaCO3(s) \;\; <\!\!-\!\!> \;\; Ca^{2+} + CO_3^-$$

If only NaCl is present a simplified procedure can be used and the maximum NaCl solubility in the aqueous phase is calculated by [Ref. 116].

$$WtNaCl = 26.218 + 0.0072T + 0.000106T^2$$

where T is the temperature in $°C$.

It is recommended that the SOLIDMMS keyword is also used in order to model the decrease in mobility as a function of the solid saturations. The gas and water saturations are normalized to be between zero and unity, so the two phase water and gas relative permeability curves can be used, however it should be noted that the effective fluid volume is given by

$$FV = (1 - S)PV$$

where PV is the pore volume and S is the solid saturation. It is possible to specify a target solid saturation change for each time step by using item 21 of TSCRIT. A small value will usually enhance the stability.

Activity coefficient models

The CO_2 and H_2O activity are computed following Spycher and Preuss, [Ref. 110] and [Ref. 132]. This is done using Margules expressions:

$$\ln(\gamma_{H2O}) = (A_M - 2A_M x_{H2O})x_{CO2}^2$$

$$\ln(\gamma_{CO2}) = 2A_M x_{CO2} x_{H2O}^2 \exp(\gamma_{CO2}')$$

where:

$$A_M = 0 \quad (T \leq 100°C)$$

$$A_M = a \cdot (T - 100) + b \cdot (T - 100)^2 \quad (T > 100°C)$$

In these expressions, a and b are regressed parameters and x_{CO2} and x_{H2O} denote the aqua phase mole fractions. The CO_2 activity coefficient is modified for salting out effects by γ_{CO2}'.

There are two models that can be applied in order to compute γ_{CO2}':

1 The Rumpf et. al 1994 model as detailed in [Ref. 110], applicable up to 160 $°C$.

2 The model given in [Ref. 132], applicable up to 250 $°C$.

To choose the model, use the keyword ACTCO2S.

Density

The gas density is obtained by a cubic equation of state tuned to accurately give the density of the compressed gas phase, following Spycher and Preuss, [Ref. 110]. A modified Redlich-Kwong equation of state is used, where the attraction parameter is made temperature dependent:

$$P = \left(\frac{RT_K}{V - b_{mix}}\right) - \left(\frac{a_{mix}}{T_K^{1/2} V(V + b_{mix})}\right)$$

where V is the molar volume, P is the pressure, T_K the temperature in Kelvin, R is the universal gas constant and a_{mix} and b_{mix} are the attraction and repulsion parameters. For more information see also [Ref. 132].

The transition between liquid CO_2 and gaseous CO_2 will lead to rapid density changes of the gas phase, the simulator use a narrow transition interval between the liquid and gaseous density to represent the two phase CO_2 region.

For the brine density we first compute the density of pure water. The default model uses an analytical form of the water density as detailed by *The International Association for the Properties of Water and Steam* (IAPWS-IF97). In order to use the pre-2010.1 water density calculations as detailed in [Ref. 112], OPTIONS3 item 232 can be put to unity. The Ezrokhi's method is then applied to calculate the effect of salt and CO_2 as described in [Ref. 111].

Viscosity

The CO_2 gas viscosity is calculated from [Ref. 113] and [Ref. 114].

Aqueous speciation

It is possible to output the different ions and chemical components in the aqueous phase by adding AQSP to the RPTRST or RPTSCHED keyword. Similarly the pH is output by adding AQPH. Additionally, the SUMMARY keywords BAQSP and BAQPH can be used.

Names of the species considered are given below

Table 9.2 Aquas Species

Aquas species	
aH2O	Pure water
aCO2	Carbon dioxide
aH+	Hydrogen ion
aOH-	Hydroxide ion
aCO3-2	Carbonate ion
aHCO3-1	Bicarbonate ion
aCl-	Chloride ion
aNa+	Sodium ion
aCa++	Calcium ion
aCaCl+	Calcium monochloride ion

Note Note as an example that the total amount of aqueous NaCl is given by the molar density, named MLSC followed by the component index, while the speciation output the ions aNA+ and aCL-.

pH calculation

The pH is calculated as

$$pH = -\log(\zeta_H \cdot m_H)$$

where ζ_H is the activity coefficient of the H+ ion and m_H is the molality of the H+ ion.

Analytic water aquifers

An analytic water aquifer defined by AQUCHWAT, AQUCT, AQUFETP or AQUFLUX contains only the “H2O” component. It is possible to modify the aquifer composition using the AQSTREAM keyword. Any component is allowed to leave or enter the reservoir. Additionally, the aquifer water properties are calculated as described above.

If the item 131 of OPTIONS3 is set to 1, this reverts to the pre-2007.2 behavior where the water entering / leaving the reservoir is made up entirely of the “H2O” component. If the item 119 of OPTIONS3 is set to 1, this reverts to the pre-2007.1 behavior where the aquifer water properties were calculated using the water PVT tables. See "Aquifer Modeling Facilities" on page 73.

Using the CO_2 storage model

RUNSPEC section

The option is invoked by the CO2STORE keyword. If a solid phase is considered the SOLID keyword must be specified. The capillary pressure mode is selected by default so that the capillary pressure is added to the gas phase pressure. To subtract the capillary pressure from the water phase pressure use option switch 8, keyword OPTIONS3. To run with a saturation weighted capillary pressure option the keyword FORMOPTS can be used with the mnemonic PCSPLIT. For runs with large capillary pressure it is recommended to use option switch 8, so the system pressure is the gas phase pressure. Also note that the WATER keyword should not be used with the CO2STORE option.

PROPS section

The fluid components are set by specifying the names in the COMPS keyword.

The water relative permeability is input using the WSF keyword, while the gas relative permeability and the gas-water capillary pressure are entered using the GSF keyword.

If only a single phase is present the composition may be entered using ZI or with respect to depth using ZMFVD.

If an initial gas-water contact exists within the reservoir you may specify either the vapor or the aqueous phase composition at the contact (ZMFVD or COMPVD may be used).

To specify molecular diffusion the keywords DIFFCWAT and DIFFCGAS should be used. Cross phase diffusion can be included by the keywords DIFFCWG and DIFFCGW. Note that without the cross phase diffusion option two neighbor cells that are single phase, one being liquid and the other gas, no diffusive flow is computed. The interaction with the formation water could therefore be slightly under predicted without including the cross phase diffusion term. For more information see "Diffusion" on page 211.

SOLID properties needs to be assigned in the usual way. The component reference density is assigned using the keyword SDREF. To specify mobility multipliers the SOLIDMMS keyword can be used.

For temperatures above160 $°C$ an alternative activity coefficient model for the aqua CO_2 should be considered.This is achieved by the keyword ACTCO2S.

SUMMARY section

The keywords FWCD, FGCDI and FGCDM can be used to output the field total molar amount of CO_2. RWCD, RGCDI and RGCDM can be used to output the same quantities on a regional basis.

Table 9.3 CO_2 specific summary quantities

Summary quantity	Description
FWCD, RWCD	Dissolved in water
FGCDI, RGCDI	In an immobile gas phase
FGCDM, RGCDM	In a mobile gas phase

Notes

1 Quantities such as FGIP, FGIPL and FGIPG depend on the state of the CO_2 at standard conditions. This means that if all the CO_2 injected is in a dissolved state these quantities are zero. The field molar amount injected for a given component can be obtained using the keywords FCMIT and FCMIP.

2 Also the water phase is considered as "hydrocarbon" when calculating the "hydrocarbon weighted" field pressure reported by FPR and FPRH. For a comparison with a simulation involving an oil phase such as CO2SOL, the "pore volume weighted" pressure returned by FPRP can be used. Additionally, setting the 151st item of OPTIONS3 to 1 ensures that only the gas phase is treated as "hydrocarbon".

SOLUTION section

The EQUIL keyword is used for equilibration. An initial solid saturation and corresponding composition can be assigned by the keywords SSOLID and SMF. Some care must be taken as this might not be in equilibrium. The initial flash will however put the minerals in equilibrium with the formation water.

SCHEDULE section

To specify a CO_2 injector the keyword WINJGAS can be used to define the composition of the injector. The composition of a water injector can also be specified by using this keyword. However, to specify the composition of an injected water stream, the keywords WELLSTRW and WINJW are now available within the CO2STORE option. When using the SOLID option it is possible to specify a target solid saturation change by using item 21 of TSCRIT.

If the adaptive implicit solution technique is used, the AIM option, and large stable time steps are wanted, the AIMCON keyword can be used in order to increase the number of implicit cells.Typically this will be the case at the end of a simulation when CO_2 injection have stopped.

Compatibility

The CO2STORE option is currently not compatible with the multi-segmented well option.

Note If the solid option is not activated the salts are not allowed to precipitate. For such cases some residual water might become more and more concentrated, with salt resulting in unphysically dense water. However, this water will typically become immobile, so the overall simulation is only marginally affected. Typically this will only occur near a CO_2 injection well, due to injection of dry CO_2.

The model applies a maximum $NaCl$ concentration for the CO_2 solubility model, corresponding to the value of salting out while other salts are limited to a fixed value of 5 Molal in the CO_2 solubility model.

Thermal CO2STORE

The CO2STORE option can be used with the THERMAL option for live oil and gas.The fluid properties are defined similarly to a standard thermal case, but the CO_2 component is allowed to dissolve in the water phase. Using the CO2STORE option together with THERMAL it is assumed that the temperature range of the reservoir is limited to 12-250 ^{o}C.

If no oil K-values are supplied the phases assumed is as for a standard CO2STORE case, a liquid water rich phase and a CO_2 rich gas phase. The density of the CO_2 rich phase and the water vapor is computed as for the standard CO2STORE option. Additional components are taken as water components unless the user supply water K-values for these components or define them as gas only components. This is done by the keywords KWTABTn and CVTYPE. Note that the CO_2 and H_2O component is allowed to be defaulted when using the CO2STORE option. If this is done the usual computations for the solubilities of CO_2 and H_2O will be applied.

In THERMAL it is possible to define the H_2O components as part of the components defined in CNAMES, or if left out it will be added as the last component of the system. If the H_2O component is specified it is also possible for H_2O to be in the oil phase. However this is only allowed if oil K-values are supplied using KVTABTn, where you need to supply the split of H_2O between oil and vapor.

Note With thermal CO2STORE, the aqua mole fractions are named AMF, while XMF refers to the oil phase mole fractions.

Density and viscosity

The density of the aqua phase is computed as described by the keyword DENAQA, equation [EQ 27.2]) and VISCAQA, equation [EQ 27.3]. Default values will be assigned for CO_2/CH_4, NACL/$CACL_2$/$CACO_3$ component. The vapour phase density is computed as for a standard thermal case, but the CO_2 and H_2O contribution to the phase specific volume is computed internally.

Similarly for vapor phase viscosity the CO_2 and H_2O contribution when calculating the phase viscosity are computed internally unless a component specific value for the viscosity is assigned.

Enthalpy

The aqua enthalpy is computed by the parameters given in the keywords SPECHW1 and SPECHW2.

For the gas phase the usual parameters given for a thermal live oil run should be supplied, note that for CO_2 the parameters defining the enthalpies for both a oil and a vapor should be given even if not an oil phase is considered.

The H_2O molar enthalpy in the vapor phase is computed internally.

Relative permeability

The relative permeability need to be of the three phase type. If not an oil phase is considered, the oil relative permeability need to be assigned dummy values.

Note WSF and GSF cannot be used to specify the relative permeabilities.

Initializing the study

The typical conditions is that the reservoir is water. It is possible to assign the initial aqua mole fractions by the AMFVD keyword when using the EQUIL keyword. If the reservoir is initialized by enumeration the aqua mole fractions are assigned by the AMF keyword.

Compatibility

The CO2STORE option is currently not compatible with the multi-segmented well option.

CO_2/H_2S Solution in the Aqueous Phase

Chapter 10

Introduction

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

The CO_2 and H_2S solution algorithm allows carbon dioxide and/or hydrogen sulfide to dissolve in the aqueous phase. This is switched on using respectively the CO2SOL and/or H2SSOL keyword in the RUNSPEC section. The basic model is a fugacity function for aqueous CO_2 and H_2S which is constructed to match solubility data and which takes the form:

$$f^A_{co_2} = Pa_{CO_2}\phi_{CO_2}(P) \qquad \text{[EQ 10.1]}$$

$$f^A_{H_2S} = Pa_{H_2S}\phi_{H_2S}(P)$$

The phase equilibrium between the aqueous CO_2 or H_2S and the hydrocarbon phases is then defined by the conditions that the fugacity values are equal. It is also possible to construct a CO_2 and H_2S aqueous phase Gibbs energy contribution.

The solubility data may be entered using the SOLUBILI keyword. Alternatively, a default set of data is provided by the program for CO_2. To obtain details of the solubility data used in a simulation the SOLU mnemonic may be specified with the RPTPROPS keyword.

The functions $\phi(P)$ is constructed by considering a pure CO_2 or H_2S/aqueous mixture - the gas phase fugacity is obtained using the equation of state.

The default CO_2 aqueous phase properties follow the correlations given by Chang, Coats and Nolen [Ref. 4]. All or any of these properties may be overwritten; see the SOLUBILI keyword for details.

For H_2S no default values exist.

The initial dissolved CO_2 or H_2S solubility as a function of depth may be specified using the RSWVD keyword. This initial solution honors this value unless it exceeds the saturated CO_2 concentration in which case the saturated value is used.

Notes

1 The initial CO_2 or H_2S concentration may not be in exact equilibrium with the hydrocarbon phases and initial flash modifies the input values slightly.

2 The first line of the solubility table defined using the `SOLUBILI` keyword represents water at standard conditions, when the dissolved CO_2 or H_2S amount is at a minimum. The simulator does not assume that the amount of CO_2 or H_2S in water at the lowest pressure is zero; if you want to specify it as such then you should set it to zero in the first line of the table. The data of this first line can be set to obtain a satisfying density of water with a small amount of dissolved CO_2 or H_2S.

For further information see "CO2SOL" on page 125.

Aqua solubilities of both CO_2 and H_2S

If both components are made soluble this should be entered by the keywords `SOLUCO2H` and `SOLUH2SC` specifying the solubility of CO_2 in the presence of H_2S and the solubility of H_2S in the presence of CO_2. Please note that no default values exist in this case. If only CO_2 or H_2S is made soluble the keyword `SOLUBILI` is used to specify either CO_2 or H_2S as the component reacting with the aqueous phase.

Note The CO_2 component is always regarded as the first component followed by the H_2S component when entering composition dependent data with `RSW` or `AMF`.

Water component properties

After computing the CO_2 aqueous mole fraction using the phase equilibrium algorithm mentioned previously, the saturated formation volume factor FVF_{sat}, the water compressibility c_{sat}, the water viscosity μ_{sat} and the water saturation pressure P_{sat} can be calculated using the CO_2-saturated water properties tables given by either SOLUBILI, SOLUBILS or SOLUBILT using linear interpolation.

The formation volume factor FW at pressure P is calculated by:

$$\frac{1}{FVF} = \frac{1}{FVF_{sat}} \cdot (1 + X_c + 0.5X_c^2)$$

where the compressibility term X_c is given by

$$X_c = c_{sat} \cdot (P - P_{sat})$$

Density

The reservoir density of water as given by the mnemonic DENW of RPTRST or RPTSOL is given by

$$\frac{\rho_s}{MW} \cdot \frac{MW_r}{FVF}$$

where

ρ_s is the surface density, specified with either keywords DENSITY, GRAVITY or SOLUBILS.

MW is the molecular weight of pure water (18.015 kg/kg-mol).

MW_r is the molecular weight of water in the reservoir calculated using the aqueous mole fraction of CO_2 (AMF_{CO_2}) and water and their molecular weights ($AMF_{CO_2} \cdot MW_{CO_2} + (1 - AMF_{CO_2}) \cdot MW$). The molecular weight of CO_2 is specified using the keyword MW.

FVF is the formation volume factor calculated as explained earlier.

Viscosity

The viscosity of water at pressure P is given by

$$\frac{\mu_{sat}}{(1 + Y_c + 0.5Y_c^2)}$$

where the term Y_c is given by

$$Y_c = -c_V \cdot (P - P_{sat})$$

where

c_V is the "viscosibility", specified with keyword PVTW

Aquifer properties

An analytic water aquifer defined by AQUCHWAT, AQUCT, AQUFETP or AQUFLUX contains only the “WATER” component. Any water component is allowed to leave the reservoir. If the item 131 of OPTIONS3 is set to 1, this reverts to the pre-2008.1 behavior where the water entering / leaving the reservoir is made up entirely of the “WATER” component. Additionally, the water properties of the analytic aquifers are calculated as described above. If the item 119 of OPTIONS3 is set to 1, this reverts to the pre-2008.1 behavior where the water PVT tables were used. See "Aquifer Modeling Facilities" on page 73.

Chemical Reactions

Chapter 11

Introduction

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

This chapter describes how chemical reactions are modeled in the ECLIPSE 300 simulator. Chemical reactions can only be used in fully implicit runs. They can be used in blackoil runs and compositional runs (EoS, K-value and thermal).

The keywords for chemical reactions can be used to model a number of other processes, for example:

- combustion
- biodegradation
- decay of radioactive tracers
- non-equilibrium reactions.

Chemical reaction keywords

RUNSPEC section keywords

The RUNSPEC keywords are:

Table 11.1 RUNSPEC keywords

Keyword	Description
REACTION	Number of reactions

PROPS section keywords

The PROPS keywords are:

Table 11.2 PROPS keywords

Simulator options	Keyword	Description
	EQLDKVCR	K-values for equilibrium deviation reactions
	EQLDREAC	Equilibrium deviation reaction rates
	REACACT	Reaction activation energy
Thermal option	REACENTH	Reaction enthalpy
	REACCORD	Reaction rate component order
	REACLIMS	Limits on pressures and temperatures used to calculate rates
	REACOPS	Options and convergence criteria
	REACPHA	Reactant phase
	REACPORD	Reaction rate porosity order
	REACRATE	Reaction rate constant
Solid option	REACSORD	Reaction rate solid order
	REACTRAC	Tracer dependent reaction rate terms
	STOPROD	Stoichiometric coefficients of products
	STOREAC	Stoichiometric coefficients of reactants

SUMMARY section keywords

The SUMMARY section keywords are:

Table 11.3 Chemical reaction SUMMARY keywords

Field	Block	Information
FREAC	BREAC	Reaction rate
FREAT		Reaction total

SCHEDULE section keywords

The SCHEDULE section keywords are:

Table 11.4 SCHEDULE keywords

Simulator options	Keyword	Description
Thermal option	`REACCRIT`	Reaction interpolation criteria

The RPTRST and RPTSCHED controls are:

Table 11.5 RPTRST, and RPTSCHED output controls

Mnemonic	Output
`REAC`	Reaction rate for each cell
`TREACM`	Tracer dependent reaction rate multiplier for each cell

Formulation

The implementation of chemical reactions in ECLIPSE follows [Ref. 54].

The number of chemical reactions is specified by the REACTION keyword. Each reaction is defined by a stoichiometry equation and reaction rate.

Stoichiometry equation

The stoichiometry equation for reaction r is given by

$$\Sigma(S_{Rri} \cdot C_i) \rightarrow \Sigma(S_{Pri} \cdot C_i) \qquad \text{[EQ 11.1]}$$

where

S_{Rri} is the stoichiometric coefficient for the reactant C_i,

S_{Pri} is the stoichiometric coefficient for the product C_i.

The stoichiometric coefficients are defined by the STOREAC and STOPROD keywords. The phase of the reactants can be specified by the REACPHA keyword.

Reaction rate

The reaction rate R_r for reaction r in a grid cell is given by

$$R_r = V_b \cdot A_r \cdot \exp(-E_r/(RT)) \cdot \Pi c_{ri}^{\,n_{ri}} \qquad \text{[EQ 11.2]}$$

where

V_b is the bulk volume (rock volume plus pore volume),

A_r is a reaction rate constant, defined by REACRATE,

E_r is the activation energy, defined by REACACT,

R is the gas constant,

T is the temperature which may be limited using REACLIMS,

n_{ri} is the index or order of the component dependent term, defined by REACCORD.

Component concentration

The terms c_{ri} represent the component concentration in the reacting phase (REACPHA). If the reactant is in the oil phase:

$$c_{ri} = \theta \cdot b_o \cdot S_o \cdot x_i \qquad \text{[EQ 11.3]}$$

If the reactant is in the gas phase:

$$c_{ri} = \theta \cdot b_g \cdot S_g \cdot y_i \qquad \text{[EQ 11.4]}$$

If the reactant is water in the water phase:

$$c_{ri} = \theta \cdot b_w \cdot S_w \qquad \text{[EQ 11.5]}$$

If the reactant is in all phases:

$$c_{ri} = \theta \cdot m_i \qquad \text{[EQ 11.6]}$$

Here θ is the porosity, b_o, b_g and b_w are the molar densities of oil, gas and water; S_o, S_g and S_w are the oil, gas and water saturations; x_i, y_i and m_i are the mole fractions in the oil and gas phases and the molar density of component i.

For reactants in the gas phase, the reaction rate can be made to depend on the partial pressure. Instead of [EQ 11.4]:

$$c_{ri} = P \cdot y_i \qquad \text{[EQ 11.7]}$$

where P is the pressure and can be limited using the `REACLIMS` keyword.

If the reactant is in the solid phase [EQ 11.6] can be used - since at present a solid component cannot appear in any other phase.

Component concentration order

The order n_{ri} of the component term in [EQ 11.2] should be non-negative for all reactants. The order for a reactant can be set to zero, in which case the reaction rate is independent of the concentration of that component; however, this is not recommended, unless keyword `EQLDREAC` is also used. If the order of a component dependent term is non-zero, and that component is not a reactant, then it acts as a catalyst.

Porosity order

In equations [EQ 11.3] - [EQ 11.6], the component terms c_{ri} represent the concentrations per cell volume. In ECLIPSE it is more natural to consider the concentrations per pore volume:

$$c'_{ri} = c_{ri}/\theta$$

The reaction rate [EQ 11.2] can then be rewritten as

$$R_r = V_p \cdot \theta^{n_{rp}} \cdot A_r \cdot \exp(-E_r/(RT)) \cdot \Pi c'^{\,n_{ri}}_{ri} \qquad \text{[EQ 11.8]}$$

where

V_p is the pore volume, $V_p = \theta \cdot V_b$

n_{rp} is the index or order of the porosity term given by

$$n_{rp} = \sum_i n_{ri} - 1$$

Here the sum is over all reaction components **except** those where the rate is defined by the partial pressure [EQ 11.7]. If necessary the porosity order can be modified using the `REACPORD` keyword.

Reactions involving solid components

In simulations with solid components the fluid saturations are modified to include the solid saturation $\hat{S}_s$:

$$\hat{S}_o + \hat{S}_g + \hat{S}_w + \hat{S}_s = 1$$

Normalized fluid saturations can then be defined

$$S_o = \hat{S}_o / (1 - \hat{S}_s)$$

$$S_g = \hat{S}_g / (1 - \hat{S}_s)$$

$$S_w = \hat{S}_w / (1 - \hat{S}_s)$$

Equations [EQ 11.3] - [EQ 11.5] are modified, for example [EQ 11.3] becomes:

$$c_{ri} = \theta \cdot b_o \cdot \hat{S}_o \cdot x_i = \theta \cdot b_o \cdot S_o \cdot x_i \cdot (1 - \hat{S}_s)$$

Equation [EQ 11.8] then becomes

$$R_r = V_p \cdot \theta^{n_{rp}} \cdot (1 - \hat{S}_s)^{n_{rs}} \cdot A_r \cdot \exp(-E_r / (RT)) \cdot \Pi c'_{ri}{}^{n_{ri}}$$

where

$$n_{rs} = \sum_i n_{ri}$$

and the sum is over all reaction components **except** those where the rate is defined by the partial pressure [EQ 11.7] or where the reactant is in all phases [EQ 11.6]. If necessary the values of n_{rs}, can be modified using the `REACSORD` keyword.

Equilibrium deviation reaction rates

The reaction rate can contain extra terms that depend on the deviation from an equilibrium value. [EQ 11.2] is replaced with:

$$R_r = V_b \cdot A_r \cdot \exp(-E_r / (RT)) \cdot \Pi c_{ri}{}^{n_{ri}} \cdot \Pi d_{nijk} \qquad \text{[EQ 11.9]}$$

The equilibrium deviation terms d_{nijk} are given by one of two equations:

$$d_{1ijk} = \theta \cdot b_o \cdot S_o \cdot (x_i - y_j / K_k) \qquad \text{[EQ 11.10]}$$

$$d_{2ijk} = \theta \cdot b_g \cdot S_g \cdot (K_k \cdot x_i - y_j) \qquad \text{[EQ 11.11]}$$

The `EQLDREAC` keyword should be used to define the values of n, i, j and k, where

n determines which of equations [EQ 11.10] - [EQ 11.11] is used,

i is the number of the component used for the oil mole fraction x_i,

j is the number of the component used for the gas mole fraction y_j, and

k is the number of the equilibrium k-value, K_k, defined by the `EQLDKVCR` keyword.

There are a number of special cases that apply to these terms:

- If a deviation term d_{nijk} is negative a reverse reaction occurs.
- If the component number i is set to zero, the oil mole fraction x_i is replaced by a value of 1.

- If the component number j is set to zero, the gas mole fraction y_i is replaced by a value of 1.

The REACTION keyword should be used to specify the total number of equilibrium deviation terms allowed in the simulation and also the number of equilibrium k-values that are defined by the EQLDKVCR keyword.

The equilibrium deviation terms d_{nijk} are used in addition to the component concentration terms c_{ri}, they do not replace them. However, if the component orders n_{ri} are set to their default values of zero (REACCORD), then the component concentration terms disappear and [EQ 11.9] becomes

$$R_r = V_b \cdot A_r \cdot \exp(-E_r/(RT)) \cdot \Pi d_{nijk} \qquad \text{[EQ 11.12]}$$

Tracer dependent reaction rates

In runs with tracers the reaction rate can contain extra terms that depend on the tracer concentrations. The tracers act as a catalyst. [EQ 11.2] is replaced with:

$$R_r = V_b \cdot A_r \cdot \exp(-E_r/(RT)) \cdot \Pi c_{ri}^{n_{ri}} \cdot \Pi T_{rj} \qquad \text{[EQ 11.13]}$$

The tracer multiplier terms T_{rj} are calculated explicitly (that is at the previous time step) as

$$T_{rj} = (\theta \cdot t_j \cdot c_c)^{n_{tj}} \qquad \text{[EQ 11.14]}$$

where t_j is the tracer concentration and c_c is the concentration of the carrier component c in the tracer phase p_j and n_{tj} is the order of the tracer term.

If the tracer phase is oil then:

$$c_c = b_o \cdot S_o \cdot x_c \qquad \text{[EQ 11.15]}$$

If the tracer phase is gas:

$$c_c = b_g \cdot S_g \cdot y_c \qquad \text{[EQ 11.16]}$$

If the carrier is water and the tracer phase is water:

$$c_c = b_w \cdot S_w \qquad \text{[EQ 11.17]}$$

If the tracer is in all phases:

$$c_c = m_c \qquad \text{[EQ 11.18]}$$

The reaction number r tracer t_j, tracer phase p_j and tracer order n_{tj} for each tracer dependent term is defined by the REACTRAC keyword. The REACTION keyword should be used to specify the total number of tracer dependent terms. The tracer names and carrier components must be defined by the TRACER keyword. Values of the tracer multipliers can be output using the mnemonic REAC in keywords RPTRST or RPTSCHED.

Component conservation equations

The residual equations are modified by adding source and sink terms that are proportional to the reaction rates R_r.

The equation for conservation of a fluid component m_{fl} (hydrocarbon or water) in each grid block at each time step becomes:

$$R_{fl} = \frac{d}{dt}(V_p m_{fl}) - \sum_r [(S_{Pri} - S_{Rri}) \cdot R_r] = 0 \qquad \text{[EQ 11.19]}$$

where the flow terms between cells and to and from wells have been ignored for simplicity.

Energy conservation equation

Thermal option only

The equation for conservation of energy e is modified in a similar manner to the component conservation equations. In each grid block at each time step:

$$R_e = \frac{d}{dt}(V_b e) - \sum_r [H_r \cdot R_r] = 0 \qquad \text{[EQ 11.20]}$$

where

V_b is the bulk volume,

H_r is the reaction enthalpy, defined by `REACENTH`,

and the convective, conduction and well terms have been ignored.

Non-uniqueness

The stoichiometric coefficients S_{Rri} and S_{Pri} are not unique - the same reaction can be defined by multiplying all the coefficients by a constant N. However, the reaction rate R_r should then be divided by N, since N reactions take N times longer. This ensures the component balance equation [EQ 11.19] remains unchanged. It is conventional, though not essential, to choose the stoichiometry coefficients so that the coefficient of the first reactant is 1. In this case the rate of reaction R_r represents the rate at which one mole of the first reaction component is reacted.

Thermal option only

In Thermal simulations the reaction enthalpy H_r should also be multiplied by N since N reactions produce N times as much energy. This ensures the energy balance equation [EQ 11.20] remains the same. If the stoichiometric coefficient of the first reactant is 1 then H_r represents the amount of energy released when one mole of the first reaction component is reacted.

Units

The units of the rate constant A_r are not fixed but are chosen to ensure the units of the reaction rate R_r are correct. Therefore they depend on the values entered for the orders of the component terms n_{ri} and may be different for different reactions.

The units of the reaction rate R_r are:

- kg-mol/Day (`METRIC`)
- lb-mol/Day (`FIELD`)
- g-mol/Hour (`LAB`)
- kg-mol/Day (`PVT-M`)

Stability of thermal runs

Thermal option only

To improve the stability when simulating rapid reactions, such as combustion, it may be necessary to interpolate values between the current time step and the previous one. Interpolation parameters can be specified using the `REACCRIT` keyword for the temperature used in the Arrhenius term in [EQ 11.2]:

$$T = \theta_T \cdot T_n + (1 - \theta_T) \cdot T_{n-1}, \; 0 < \theta_T \leq 1$$

and the oil saturation used in [EQ 11.3]:

$$S_o = \theta_S \cdot S_{o,n} + (1 - \theta_S) \cdot S_{o,n-1}, \; 0 < \theta_S \leq 1$$

Output

The reaction rate [EQ 11.2] can be output to the summary file or the RESTART file for graphical display or to the print file.

To output the reaction rate to the summary file the mnemonics BREAC, FREAC and FREAT can be used in the SUMMARY section.

BREAC outputs the reaction rate for specified cells and specified reactions.

FREAC outputs the reaction rate for the whole field for specified reactions.

FREAT outputs the reaction total since the beginning of the simulation for the whole field for specified reactions.

If the reaction number is defaulted in keywords FREAC or FREAT, then data are output for all reactions.

For example:

```
BREAC
--  I  J  K  R
    3  4  5  1 /  -- Rate for reaction 1 in cell (3,4,5)
    3  4  5  2 /  -- Rate for reaction 2 in cell (3,4,5)
    6  7  8  2 /  -- Rate for reaction 2 in cell (6,7,8)
 /

FREAC
1 2/  -- Reaction rates summed over all cells for reactions 1 and 2

FREAT
/  -- Reaction totals summed over all cells for all reactions
```

To output the reaction rate to the RESTART file the REAC mnemonic can be used with the RPTRST keyword. The reaction rate is printed for each cell and each reaction. For example:

```
RPTRST
REAC/
```

To output the reaction rate to the print file the REAC mnemonic can be used with the RPTSCHED keyword. The reaction rate is printed for each cell and each reaction. For example:

```
RPTSCHED
REAC/
```

Examples

Example 1: Combustion

[Ref. 54] gives a thermal example with 5 components plus water:

Table 11.6 Example names for components

Component	Name
C1	C_{32}+C_{47}
C2	C_{20}
C3	C_6+C_8+C_{12}
C4	O_2
C5	N_2-CO_X
W	H_2O

Table 11.7 Example stoichiometry

Reaction	Stoichiometry
1	C1 + 45.17C4 ->37.46 C5 + 29.71 W
2	C2+25.08 C4 -> 20.8 C5 + 16.5 W
3	C3 + 11.66 C4 -> 9.65 C5 + 7.67 W

Table 11.8 Example reaction rate constants, activation energy and reaction enthalpy values

Reaction	Rate constant	Activation Energy (Btu/lbm)	Reaction Enthalpy (Btu/lbm)
1	3.4054×10^{10}	32785	16000
2	0.28164×10^{10}	32785	16000
3	0.4035×10^{10}	32785	16000

The rate of each reaction is proportional to the partial pressure of oxygen and the square of the oil component concentration.

The following keyword needs to be added to the `RUNSPEC` section:

```
REACTION
3 /
```

The following keywords need to be added to the PROPS section:

```
STOREAC
-- C1     C2     C3     C4     C5      W
    1      0      0  45.17      0      0 / reaction 1
    0      1      0  25.08      0      0 / reaction 2
    0      0      1  11.66      0      0 / reaction 3

STOPROD
-- C1     C2     C3     C4     C5      W
    0      0      0      0  37.46  29.71 / reaction 1
    0      0      0      0  20.80  16.51 / reaction 2
    0      0      0      0  9.670   7.67 / reaction 3

REACRATE
   3.4054E10  0.28164E10   0.4035E10     / reaction 1,2,3

REACACT
       32785       32785       32785     / reaction 1,2,3

REACENTH
       16000       16000       16000     / reaction 1,2,3

REACPHA
-- C1     C2     C3     C4     C5      W
  OIL     1*     1*    GPP     1*     1* / reaction 1
   1*    OIL     1*    GPP     1*     1* / reaction 2
   1*     1*    OIL    GPP     1*     1* / reaction 3

REACCORD
-- C1     C2     C3     C4     C5      W
    2      0      0      1      0      0 / reaction 1
    0      2      0      1      0      0 / reaction 2
    0      0      2      1      0      0 / reaction 3
```

Example 2: A dynamic approach to equilibrium

In this example two components are used to model the oil phase and gas phase of a single hydrocarbon. The flash is replaced by an equilibrium-deviation reaction, representing evaporation and condensation.

The following keyword is needed in the RUNSPEC section:

```
REACTION
1 1 1 / one reaction, one deviation term, one EQLD k-value
```

In the PROPS section

```
CNAMES
C_OIL C_GAS /

CVTYPE
DEAD  GAS / Switch off the flash for these components

STOREAC
-- C_OIL   C_GAS  W
    1      0      0 / reaction 1: C_OIL <-> C_GAS

STOPROD
-- C_OIL   C_GAS  W
    0      1      0 / reaction 1: C_OIL <-> C_GAS

EQLDREAC
-- # N I J K
   1 1 1 2 1 / reaction 1: (X1 - Y2/K1)

EQLDKVCR
1.23E6
833.4E6
0
1.6E4
0            / K1 = (A + B/P + C.P) . EXP (D / (T-E))

REACRATE
   3.4054E10 /

REACACT
       32785 /

REACENTH
       -1600 / LATENT HEAT
```

The Coal Bed Methane model

Chapter 12

Introduction

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL

The naturally occurring methane within coal seams can in some cases be economically produced using oil field technology. In these “coal bed methane” projects, wells are drilled into the coal seam to produce gas. Considerable reserves of this “unconventional” natural gas are present in many parts of the world.

The coal beds are naturally fractured systems with the gas adsorbed into the coal matrix. Primary production occurs by initially de-watering the natural fractures and hence reducing the pressure in the fracture system. The reduced pressure in the fractures allows gas desorption from the surface of the coal to the fracture. Gas diffuses from the bulk of the coal towards the fracture surface.

For dual porosity the Coal Bed Methane model uses a modified Warren and Root model to describe the physical processes involved in a typical coal bed methane project.

In ECLIPSE 100 it is possible to introduce a second gas, known as a solvent, for enhanced coal bed methane projects, while in ECLIPSE 300 a full compositional treatment is possible.

Caution You should be aware that certain entities, corporations, individuals or organizations may or do hold applicable patents with regard to enhanced recovery of coal bed gases through injection of both inert and non-inert gases. You are strongly encouraged to investigate if any proposed field applications are subject to these patents.

Single porosity model

In ECLIPSE 300 it is possible to activate an instant sorption model that can be used in single porosity mode. With this model the amount of adsorbed gas is immediately in equilibrium with the free gas phase.

Dual porosity model

The Dual Porosity model consists of two interconnected systems representing the coal matrix and the permeable rock fractures. To model such systems, two simulation cells are associated with each block in the geometric grid, representing the coal matrix and fracture volumes of the cell. In ECLIPSE, the properties of these blocks may be independently defined. In a dual porosity run of ECLIPSE the number of layers in the Z-direction should be doubled. ECLIPSE associates the first half of the grid (the first `NDIVIZ`/2 layers) with the matrix blocks, and the second half with the fractures. In such runs `NDIVIZ` must therefore be even; ECLIPSE checks that this is the case.

Unlike a dual porosity oil reservoir model, in which the matrix has both an associated pressure and an oil saturation, only the gas concentration in the coal is tracked. In the fracture system, however, the standard flow equations are solved.

Note In a coal bed methane model the pore volume of the matrix cells has a different interpretation than for an ordinary dual porosity run; it gives the coal volume of the cell, using the time-dependent sorption model. By default the porosity is set to unity minus the porosity of the fracture. The cell bulk volume times the porosity then equals the coal volume. It is possible to use the keyword `ROCKFRAC` instead of the porosity to set the coal volume in ECLIPSE 300.

Note By default the input permeabilities in the fracture are scaled by the porosities. To turn this off use the keyword `NODPPM`.

Multi porosity model

In ECLIPSE 300 a multi porosity model can be activated, enabling a detailed study of the transient behavior in the matrix. See the "Multi porosity" on page 627 of the option.

Adsorption/diffusion models in ECLIPSE 100

The adsorbed concentration on the surface of the coal is assumed to be a function of pressure only, described by a Langmuir Isotherm. The Langmuir Isotherm is input as a table of pressure versus adsorbed concentrations. Different isotherms can be used in different regions of the field.

The diffusive flow of gas from the coal matrix is given by:

$$F_g = \mathrm{DIFFMF} \cdot D_c \cdot (\mathrm{GC}_b - \mathrm{GC}_s) \qquad \text{[EQ 12.1]}$$

where

F_g = gas flow

DIFFMF = matrix-fracture diffusivity

D_c = diffusion coefficient

GC_b = bulk gas concentration

GC_s = surface gas concentration.

GC_s is a function of fracture pressure given by the LANGMUIR table. The diffusion coefficient D_c is supplied using the DIFFCOAL keyword.

The diffusivity is given by:

$$\text{DIFFMF} = \text{DIFFMMF} \cdot \text{VOL} \cdot \sigma \qquad \text{[EQ 12.2]}$$

where

DIFFMMF is the multiplying factor input using DIFFMMF (default = 1.0)

VOL is the cell coal volume

σ is the factor to account for the matrix-fracture interface area per unit volume.

Sigma is a shape factor which accounts for the matrix fracture interface area per unit volume of coal matrix. For pseudo-steady-state flow in the coal matrix, the product $\sigma \cdot D_c$ is related to the time constant τ by

$$\sigma = \frac{1}{D_c} \cdot \tau$$

where the gas diffusion coefficient (area/time) is D_c

and the gas desorption time (time) is

$$\tau = \frac{1}{\sigma \cdot D_c \cdot DIFFMMF}$$

If the surface gas concentration is greater than the bulk gas concentration, gas may be re-adsorbed into the coal. In this case the flow of gas is treated slightly differently, as shown below. It is also possible to control the rate of re-adsorption and, in the limiting case, to prevent re-adsorption.

$$F_g = \text{DIFFMF} \cdot D_c \cdot S_g \cdot \text{RF} \cdot (\text{GC}_b - \text{GC}_s) \qquad \text{[EQ 12.3]}$$

where

S_g is the gas saturation in the fracture

RF is the re-adsorption factor, input using the DIFFCOAL keyword (default = 1.0).

Gas in the fracture will not contact the entire surface area of the coal if water is also present in the fracture. Hence the flow of gas into the coal is reduced by S_g to represent the reduced contact area.

The RF factor can be used to control the rate of re-adsorption. Setting $\text{RF} = 0$ prevents gas re-adsorption.

Adsorption model in ECLIPSE 300

The Extended Langmuir isotherm, [Ref. 103], is used to describe the coal sorption for the different components; see keyword LANGMEXT. The adsorption capacity is a function of the pressure and the free gas phase composition. For each component two parameters need to be input, the Lagmuir volume constant $\mathbf{V}_i$ and the Langmuir pressure constant $\mathbf{P}_i$. These parameters are typical determined from experiments. Different isotherms can be used in different regions of the field; see keyword COALNUM.

The multicomponent adsorption capacity is calculated by

$$L(p, y_1, y_2, \ldots)_i = \theta \frac{P_s}{RT_s} \left(V_i \frac{y_i \frac{p}{\boldsymbol{P}_i}}{1 + \sum_{j=1}^{nc} y_j \frac{p}{\boldsymbol{P}_j}} \right)$$

where

θ = Scaling factor

P_s = Pressure at standard conditions

R = Universal gas constant

T_s = Temperature at standard conditions

$\mathbf{V}_i$ = Langmuir volume constant comp. i

$\mathbf{P}_i$ =Langmuir pressure constant comp. i

y_i = Hydro carbon mole fraction in gas phase comp. i

p = Pressure

For the special case of a single component, the Extended Langmuir isotherm is identical to the usual Langmuir isotherm giving the storage capacity as a function of pressure only:

$$L(p) = \theta \frac{P_s}{RT_s} \left(V \frac{\frac{p}{\boldsymbol{P}}}{1 + \frac{p}{\boldsymbol{P}}} \right) \qquad \text{[EQ 12.4]}$$

where **V** is the maximum storage capacity for the gas, referred to as the Langmuir volume constant, and **P** is the Langmuir pressure constant. The constants used in the Extended Langmuir formulation can hence be estimated from a serious of single component gas experiments.

Since the Langmuir volume constant V is given as surface volume over coal weight we translate this to moles over coal weight by the ideal gas law (by the term $\frac{P_s}{RT_s}$).

It is possible to scale the adsorption capacity, keyword `LANGMULT`, by a factor θ for each cell in the grid. Typically this can be used to account for differences in ash or moisture contents.

Time dependent diffusion in ECLIPSE 300

The diffusive flow between the matrix and the fracture is given by

$$F_i = DIFFMF \cdot D_{c,i} \cdot S_g \cdot RF_i \cdot (m_i - \rho_c L_i) \qquad \text{[EQ 12.5]}$$

where

m_i = Molar density in the matrix coal.

$DIFFMF$ = Matrix fracture diffusivity.

ρ_c = Rock density (Coal density), input using `ROCKDEN`.

$D_{c,i}$ =Diffusion coefficient (coal) component i, input using `DIFFCBM`.

RF_i =Readsorption factor component i, input using RESORB.

S_g =Gas saturation, for desorption a value of unity is used.

As can be seen by the flow equation the density of the coal needs to be specified, as this is multiplied by the Langmuir adsorption capacity. The term $\rho_c L_i$ gives the equilibrium adsorbed molar density. The coal density ρ_c is entered with the keyword ROCKDEN.

For adsorption, the area of the gas in contact with the coal is accounted for by scaling with the gas saturation. (It is possible to use a value of unity instead by turning on option switch 93, keyword OPTIONS3)

The matrix fracture diffusivity is given by

$$\text{DIFFMF} = \text{DIFFMMF} \cdot \text{VOL} \cdot \sigma \qquad \text{[EQ 12.6]}$$

where

DIFFMMF is the multiplying factor input using DIFFMMF (default = 1.0)

VOL is the cell coal volume

σ is the factor to account for the matrix-fracture interface area per unit volume.

Often the sorption times is a quantity that is easier to obtain than the diffusion coefficients. For desorption we write the flow as

$$F_i = \frac{VOL}{\tau_i} \cdot (m_i - \rho_c L_i) \qquad \text{[EQ 12.7]}$$

where

$$\tau_i = \frac{1}{D_{c,i} \cdot DIFFMMF \cdot \sigma}$$

is called the sorption time. The parameter controls the time lag before the released gas enters the coal fracture system. As can be seen the sorption times are given by the diffusion coefficients, DIFFCBM, and the matrix-fracture interface area, SIGMA, together with the multiplying factor DIFFMMF. If the sorption times are known a value of unity can be assigned to σ and DIFFMMF. The diffusion coefficients can then be assigned to the reciprocal of the sorption times.

The time dependent diffusive flow can be used with

- Dual porosity
- Multi porosity, where coal cells must connect to pore volume cells

Instant sorption in ECLIPSE 300

If the sorption time approaches zero, the adsorbed gas will be in instant equilibrium with the free gas phase. The keyword CBMOPTS is used to activate the instant sorption model.

The instant sorption model can be used with

- Single porosity
- Dual porosity
- Multi porosity

With this model every cell is assumed to contain a pore volume, the remaining fraction of the bulk volume being taken as the coal volume. For dual and multi porosity models this is usually not appropriate as each pore system will add up the rock volume. In this case an average rock volume over the pore systems is calculated and partitioned among the connected pore systems. The default behavior can be changed by using the keyword ROCKFRAC to specify the fraction of the bulk volume of each simulation grid cell making up the coal volume.The sum of the porosities and the rock fractions over the connected pore systems should usually be unity.

Rock compaction

There are several models available in order to model the compression and expansion of the pore volumes and the effect on permability for a coal bed methane reservoir.

A Palmer-Mansoori model, ([Ref. 109] in "ECLIPSE Technical Description"), is available using the ROCKPAMA keyword.

ECLIPSE 300

An extension for multicomponent gases is available by the ROCKPAME keyword enabling the modelling of coal swelling and shrinkage as a function of both composition and pressure.

For more information see "Rock Compaction" on page 859.

It is only the pore volume where the effect of compaction is modeled. The coal volumes of a simulation cell is usually much larger than the pore space. The change in volume due to rock compaction and swelling hence only marginally affect the molar density of the adsorbed gas.

Modeling compositional effects with ECLIPSE 100

Introduction

Improved recovery of methane from coal beds can be achieved by a secondary recovery scheme involving the injection of a second gas, typically nitrogen or carbon dioxide. There are two main mechanisms that aid recovery of the coal's methane content. Firstly, by reducing the partial pressure of the methane present in the fracture, the diffusion from the coal is enhanced. The second mechanism is the competitive adsorption of carbon dioxide into the coal matrix. Molecules of CO_2 preferentially sorb on to the surface adsorption sites, displacing methane.

ECLIPSE allows the modeling of a range of secondary recovery processes by introducing a second gas, known as a solvent, which can be employed to model either nitrogen or CO_2 injection. Two models can be used to describe the adsorption process:

Adsorption model 1: COALADS keyword

The solvent/gas adsorption model is based on three input tables:

1. Langmuir isotherm for pure gas, typically methane.
   The table defines the gas surface concentration versus pressure.
2. Langmuir isotherm for pure solvent, CO_2 say.
   The table defines the solvent surface concentration versus pressure.
3. Table of the relative adsorption of gas and solvent as a function of the composition of the fracture gas.

Then:

Surface concentration of gas = $C_g(P) \cdot F_g(f_g)$

Surface concentration of solvent = $C_s(P) \cdot F_s(f_g)$

Where

$C_g(P)$ is the adsorption of pure gas at pressure P (`LANGMUIR` keyword)

$C_s(P)$ is the adsorption of pure solvent at pressure P (`LANGSOLV` keyword)

F_g is the gas relative adsorption (`COALADS` keyword)

F_s is the solvent relative adsorption

f_g is the fraction of gas in the fracture = $S_g/(S_g + S_{\text{solv}})$

Adsorption model 2: COALPP keyword

The second adsorption model is based on the partial pressures of the component gases. Again, three input tables are required:

1. Langmuir isotherm for pure gas, typically methane.
   The table defines the gas surface concentration versus pressure.

2 Langmuir isotherm for pure solvent, CO_2 for example.
The table defines the solvent surface concentration versus pressure.

3 Table of the relative adsorption of gas and solvent as a function of the composition of the fracture gas.

Then:

Surface concentration of gas = $C_g(P') = C_g(P \cdot F_g(f_g))$

Surface concentration of solvent = $C_s(P') = C_s(P \cdot F_s(f_g))$

Where

$C_g(P')$ is the adsorption of gas at partial pressure P' (`LANGMUIR` keyword)

$C_s(P')$ is the adsorption of solvent at partial pressure P' (`LANGSOLV` keyword)

F_g is the gas relative adsorption (`COALPP` keyword)

F_s is the solvent relative adsorption

fg is the fraction of gas in the fracture = $S_g/(S_g + S_{\text{solv}})$

Using the ECLIPSE 100 coal bed methane model

Single gas mode

The model is activated by specifying the keyword COAL in the RUNSPEC section. The dual porosity model should also be activated with the RUNSPEC keyword DUALPORO, and the program should be run in gas-only or gas-water mode (OIL may not be specified in RUNSPEC).

Grid data

The first NDIVIZ/2 layers of the model represent the coal matrix, and the second NDIVIZ/2 layers represent the fracture system. It is possible to have active fracture cells with no corresponding matrix cell; however, all active matrix cells require a corresponding active fracture cell. The data required for the coal matrix consists of:

1. Geometry

   Either DX / DY / DZ or ZCORN / COORD

2. Porosity

   This is used to calculate the coal volume, and is taken to represent to fraction of the volume occupied by coal. (The porosity can be defaulted, in which case it is set to 1 - fracture porosity.)

3. Net-to-Gross

   Optional, defaults to 1.0

The grid block coal volumes can be output using the 'PORV' mnemonic in the RPTGRID keyword. Note that region and field coal volumes are printed out on the fluid-in-place balance sheets (mnemonic 'FIP' in RPTSOL and RPTSCHED).

The matrix-fracture connection factor, SIGMA, needs to be supplied using either the SIGMA or SIGMAV keywords.

Properties data

Gas and water PVT and relative permeability data need to be supplied in the usual way.

In addition the following keywords must be supplied:

Table 12.1 Properties data

Keywords	Description
LANGMUIR	Inputs tables of surface gas concentration versus pressure.
DIFFCOAL	Inputs the diffusion coefficient and the re-adsorption factor.

It is possible to scale the tables of surface gas concentration versus pressure on a cell-by-cell basis using the MLANG keyword. This keyword is optional, and if it is omitted the raw table data is used.

Equilibration

If the model is equilibrated using the EQUIL keyword, no extra data is required. In this case the initial coal gas concentration is set to the equilibrium concentration at the prevailing fracture pressure. Two optional keywords are available to set the initial coal gas concentration manually:

Table 12.2 Equilibration data

Keywords	Description
GASCONC	Inputs the coal gas concentration on a block-by-block basis.
GCVD	Inputs the coal gas concentration as a function of depth.

Care should be taken when using non-equilibrium starts. ECLIPSE prints a warning if the coal gas concentration exceeds the equilibrium value in any grid block.

For enumerated starts, either one of the above two keywords must be present.

Summary data

Three extra SUMMARY section keywords are available to output the coal gas concentration on a field, region and block basis:

Table 12.3 Summary data

Keywords	Description
FCGC	field coal gas concentration
RCGC	region coal gas concentration
BCGC	block coal gas concentration.

Two gas mode - compositional effects

The second gas component is activated by entering the keyword SOLVENT in the RUNSPEC section.

Solvent properties and adsorption data

The properties of the solvent gas are supplied in an analogous manner to the gas phase, with keywords:

Table 12.4 Solvent Properties data

Keywords	Description
SDENSITY	solvent surface density
PVDS	solvent PVT tables
SSFN	gas/solvent relative permeability data for flow in the fractures.

The adsorption process is described by two extra keywords:

Table 12.5 Adsorption keywords

Keywords	Description
LANGSOLV	Langmuir isotherm for pure solvent
COALADS or COALPP	relative adsorption data for gas and solvent.

The LANGSOLV data can be scaled using the MLANGSLV keyword in a manner analogous to the MLANG keyword.

Either the COALADS keyword or the COALPP keyword can be used to specify the relative adsorption data. If the COALADS keyword is used, the relative adsorption is specified as a table of relative adsorption factors. The alternative COALPP keyword inputs a table of pressure factors for the partial pressure model.

Equilibration

By default the initial solvent fraction is taken to be zero. It is possible to specify an initial solvent fraction using one of two keywords:

Table 12.6 Equilibration data

Keywords	Description
SOLVFRAC	Inputs the initial solvent fraction in the fracture system. This keyword should be used when the initial conditions are specified by equilibration
SSOL	Inputs the initial solvent saturation for enumerated starts.

The initial concentrations of gas and solvent in the coal are then calculated from the LANGMUIR, LANGSOLV and COALADS data. Two optional keywords are available to set the initial coal solvent concentration manually:

Table 12.7 Initial coal solvent concentration manually

Keywords	Description
SOLVCONC	Inputs the coal solvent concentration on a block-by-block basis
SCVD	Inputs the coal solvent concentration as a function of depth.

Notes

1 If initial mobile gas exists in the model, the initial conditions are not quite in equilibrium, as the solvent and gas components have different densities. The equilibration calculation is based on the gas (methane) properties only.

2 An initial gas saturation in the reservoir can be modeled by setting the connate gas saturation greater than zero. This has the advantage of being in initial equilibrium, and allows ECLIPSE to calculate the initial coal concentration based on the SOLVFRAC data alone.

3 The connate gas saturation is specified by starting the gas saturation table (SGFN) at S_{gco}, and ending the water saturation table (SWFN) at 1.0 - S_{gco}.

It is also possible to initialize the model by enumeration and to include a solvent saturation. This is achieved using the SSOL keyword in the SOLUTION section.

Solvent injection

Solvent can be injected by specifying a gas injection well. A separate keyword WSOLVENT can then be used to specify the fraction of the gas flow that is solvent.

Summary data

Three extra SUMMARY section keywords are available to output the coal solvent concentration on a field, region and block basis:

Table 12.8 Summary data

Keywords	Description
FCSC	field coal solvent concentration
RCSC	region coal solvent concentration
BCSC	block coal solvent concentration.

The gas (methane) volume can be extracted from the total gas production using the following field/group/well keywords:

Table 12.9 Field group well keywords

Keywords	Description
FMPR GMPR WMPR	Methane Production Rate
FMPT GMPT WMPT	Methane Production Total
FMIR GMIR WMIR	Methane Injection Rate
FMIT GMIT WMIT	Methane Injection Total.

Summary of keywords ECLIPSE 100

RUNSPEC section

The RUNSPEC keywords are:

Table 12.10 RUNSPEC keywords

Keyword	Description
COAL	Specifies that the Coal Bed Methane model is to be used
DUALPORO	This keyword is obligatory for the Coal Bed Methane model, to specify a dual porosity run. **Do not use** DUALPERM.
SOLVENT	This keyword must be entered to activate the Solvent model if you are using the two gas mode.

GRID section

The GRID keywords are:

Table 12.11 GRID keywords

Keyword	Description
DIFFMMF	Modify the Matrix-Fracture diffusivity.
SIGMA	Input the dual porosity SIGMA factor.
SIGMAV	Input the SIGMA factor on a cell by cell basis.

PROPS section

The PROPS keywords are:

Table 12.12 PROPS keywords

Keyword	Description
COALADS	Input the gas/solvent relative adsorption data
COALPP	Input the gas/solvent partial pressure data.
DIFFCOAL	Input the gas diffusion data.
LANGMUIR	Tables of coal surface gas concentration.
LANGSOLV	Tables of coal surface solvent concentration.
MLANG	Maximum surface gas concentration.

Table 12.12 PROPS keywords

Keyword	Description
MLANGSLV	Maximum surface solvent concentration.
PVDS	PVT properties of solvent.
RPTPROPS	Controls output of the PROPS section: DIFFC or DIFFCOAL outputs the diffusion data ENDPTS outputs the MLANG and MLANGSLV scaling data LANGMUIR, LANGSOLV or COALADS outputs the Langmuir isotherms and relative adsorption data.
SDENSITY	Solvent density data.
SSFN	Gas/Solvent relative permeability data.

SOLUTION section

The SOLUTION keywords are:

Table 12.13 SOLUTION keywords

Keyword	Description
GASCONC	Initial coal gas concentration.
GCVD	Initial coal gas concentration versus depth.
RPTSOL	Controls the output from the SOLUTION section: FIPSOL outputs solvent fluid in place reports GASCONC outputs matrix grid block gas and solvent concentrations SSOL outputs current solvent saturation.
SCVD	Initial solvent concentration versus depth.
SOLVCONC	Initial coal solvent concentration.
SOLVFRAC	Initial solvent fraction in the gas phase
SSOL	Initial solvent saturation.

SUMMARY section

The following keywords control output of data specific to the coal bed methane model in ECLIPSE 100:

Table 12.14 Coal bed methane SUMMARY output controls

Field	Group	Well	Region	Block	Information
FMPR	GMPR	WMPR			Methane Production Rate
FMPT	GMPT	WMPT			Methane Production Total
FMIR	GMIR	WMIR			Methane Injection Rate

Table 12.14 Coal bed methane SUMMARY output controls

Field	Group	Well	Region	Block	Information
FMIT	GMIT	WMIT			Methane Injection Total
FCGC			RCGC	BCGC	Coal Gas Concentration
FCSC			RCSC	BCSC	Coal Solvent Concentration

Note Note that the methane production (FMPR etc.) is identical to the gas production (FGPR etc.) unless the solvent option is active. In that case the gas production rate (FGPR) is the total gas + solvent production rate.

SCHEDULE section

The SCHEDULE keywords are:

Table 12.15 SCHEDULE Keywords

Keyword	Description
RPTSCHED	Control the output from the SCHEDULE section: FIPSOL outputs solvent fluid in place reports GASCONC outputs matrix grid block gas and solvent concentrations SSOL outputs current solvent saturation.
WSOLVENT	Sets the solvent fraction in gas injection wells.

Examples

Basic test ECLIPSE 100

RUNSPEC

```
RUNSPEC    ======================================
TITLE
                  Coal Bed Methane Basic Test
DIMENS
 8    8    2  /
DUALPORO
WATER
GAS
FIELD
COAL
EQLDIMS
 1  100    2    1   20 /
TABDIMS
 1    1   20   20    3    5 /
REGDIMS
 3    1    0    0    0    1 /
WELLDIMS
 2   13    1    2 /
NUPCOL
 4 /
START
 26 'JAN' 1983  /
NSTACK
 20 /
FMTOUT
FMTIN
```

GRID

```
GRID    ============================================================
EQUALS
 'DX'       75    /    PROPERTIES COMMON TO MATRIX AND FRACTURES
 'DY'       75    /
 'DZ'       30    /
 'TOPS'   4000    /
 'PERMZ'     0    /
 'DIFFMMF'  1.0   /
 'PORO'   0.01         4*  2  2  /  FRACTURE PROPERTIES (LAYER TWO)
 'PERMX' 500000    /                MATRIX POROSITY DEFAULTS TO
 'PERMY' 500000    /                1 - PORO(FRACTURE)
-- INACTIVE MATRIX CELLS ARE OK
 'NTG'  0.0  3 3 3 3  1 1 /
 'NTG'  1.0  3 3 3 3  2 2 /
/
RPTGRID
 'DX'  'DY'  'DZ'  'PERMX'  'PERMY'  'PERMZ'  'MULTX'
 'MULTY'  'MULTZ'  'PORO'  'NTG'  'TOPS'  'PORV'
 'DEPTH'  'TRANX'  'TRANY'  'TRANZ'  'KOVERD'  /
--SIGMA FOR 10 BY 10 BY 30 FT BLOCKS
SIGMA
 0.08   /
```

PROPS

```
PROPS   ============================================================
--COAL GAS DIFFUSION COEFFICIENT
DIFFCOAL
 0.2  /
LANGMUIR
 0.0       0.0
 1000.0    0.05
 2000.0    0.065
 3000.0    0.083
 4000.0    0.10
 6000.0    0.11
/
SWFN
  0.0    0.0   0.0
  0.2    0.0   0.0
  1.0    1.0   0.0
/
SGFN
  0.0    0.0   0.0
  0.1    0.0   0.0
  1.0    1.0   0.0
/
```

```
PVTW
  .0000000   1.00000    3.03E-06    .50000    0.00E-01 /
--   PGAS   BGAS   VISGAS
PVDG
      400    5.9   0.013
      800    2.95  0.0135
     1200    1.96  0.014
     1600    1.47  0.0145
     2000    1.18  0.015
     2400    0.98  0.0155
     2800    0.84  0.016
     3200    0.74  0.0165
     3600    0.65  0.017
     4000    0.59  0.0175
     4400    0.54  0.018
     4800    0.49  0.0185
     5200    0.45  0.019
     5600    0.42  0.0195 /
ROCK
 4000.00        .30E-05 /
DENSITY
 52.0000  64.0000   .04400 /
RPTPROPS
 'SOF2'   'SWFN'   'SGFN'   'PVTO'
 'PVTW'   'PVTG'   'DENSITY'  'ROCK'  'DIFFC'
 'LANGMUIR'  /
```

REGIONS

```
REGIONS   ==========================================================
=
EQUALS
 'FIPNUM' 1  1 8 1 8 1 1 /
 'FIPNUM' 2  1 8 1 8 2 2 /
 'FIPNUM' 3  2 2 2 2 2 2 /
/
```

SOLUTION

```
SOLUTION  ==========================================================
EQUIL
4000  3959  4000  0   0   0   0   0   0  /
RPTSOL
 'PRES'  'SWAT'  'SGAS'  'FIP=2'  'EQUIL'  'GASCONC'  /
```

SUMMARY

```
SUMMARY    ==========================================================
=
FPR
FGPR
FWPR
FCGC
RCGC
 1 2 3 /
BCGC
 2 2 1 /
/
RUNSUM
SEPARATE
```

SCHEDULE

```
SCHEDULE   ==========================================================
RPTSCHED
 'PRES'  'SWAT'  'SGAS'  'RESTART=2'  'FIP=2'
 'WELLS=2'  'CPU=2'  'NEWTON=2'  /
WELSPECS
'P'  'G'   8  8  4000  'GAS'  /
/
COMPDAT
'P'  8  8  2  2 'OPEN'  0  .000000  .5000  .00000   .0000  0.000E-01/
/
WCONPROD
'P'  'OPEN'  'WRAT'     1*         100.0      100000.00000
     1*          1*      200.000  0.000000   1*  0.00000000/
/
TSTEP
 1.0 9.0 90.0 9*100 6*1000
/
END
```

CO_2 injection example ECLIPSE 100

RUNSPEC

```
RUNSPEC    ======================================
TITLE
                  CO2 Injection Example
DIMENS
 8    8    2  /
DUALPORO
WATER
GAS
SOLVENT
FIELD
COAL
EQLDIMS
 1  100    2    1   20 /
TABDIMS
 1    1   20   20    2    5 /
REGDIMS
 2    1    0    0    0    1 /
WELLDIMS
 2   13    1    2 /
NUPCOL
 4 /
START
 26 'JAN' 1983  /
NSTACK
 20 /
```

GRID

```
GRID      ============================================================
==
EQUALS
 'DX'       75    /    PROPERTIES COMMON TO MATRIX AND FRACTURES
 'DY'       75    /
 'DIFFMMF'  1.0   /
 'DZ'       30    /
 'PERMZ'     0    /
 'PORO'   0.005      1  8  1  8  2  2  /  FRACTURE PROPERTIES
 'PERMX' 500000    /
 'PERMY' 500000    /
 'NTG'  0.0  3 3 3 3   1 1 /
 'NTG'  1.0  3 3 3 3   2 2 /
-- 10 per cent dip in the X - direction.
 'TOPS'   4000     1 1   1 8  1 2   /
 'TOPS'   4007.5   2 2   1 8  1 2   /
 'TOPS'   4015     3 3   1 8  1 2   /
 'TOPS'   4022.5   4 4   1 8  1 2   /
 'TOPS'   4030     5 5   1 8  1 2   /
 'TOPS'   4037.5   6 6   1 8  1 2   /
 'TOPS'   4045     7 7   1 8  1 2   /
 'TOPS'   4052.5   8 8   1 8  1 2   /
/
SIGMA
 0.08   /
```

PROPS

```
PROPS   ============================================================
DIFFCOAL
 20.0 1.0 /
LANGMUIR
   0.0   0.0
  257.   0.092
  528.   0.14
 1000.   0.20
/
```

```
--
-- Coal quality improves in the Y direction
--
EQUALS
 'MLANG'     5   1 8  1 1  1 1 /
 'MLANG'     7   1 8  2 2  1 1 /
 'MLANG'     9   1 8  3 3  1 1 /
 'MLANG'    12   1 8  4 4  1 1 /
 'MLANG'    15   1 8  5 5  1 1 /
 'MLANG'    20   1 8  6 6  1 1 /
 'MLANG'    30   1 8  7 7  1 1 /
 'MLANG'    35   1 8  8 8  1 1 /
/
--
-- The same trend in the CO2 isotherm.
--
COPY
 'MLANG' 'MLANGSLV'  /
/
MULTIPLY
 'MLANGSLV'  1.4 /
/
LANGSOLV
 0.0        0.080
 279.       0.092
 661.       0.227
1000.       0.260
/
COALADS
 0.0  0.0   1.0
 0.5  0.5   0.2
 1.0  1.0   0.0
/
--
-- Connate gas saturation of 10 per cent
--
SWFN
  0.0    0.0   0.0
  0.2    0.0   0.0
  0.9    1.0   0.0
/
SGFN
  0.0     0.0   0.0
  0.1     0.0   0.0
  1.0     1.0   0.0
/
SSFN
 0     0  0
 0.01  0  0
 1     1  1
/
PVTW
  .0000000   1.00000    3.03E-06    .50000    0.00E-01 /
```

```
--   PGAS   BGAS   VISGAS
PVDG
      400    5.9   0.013
      800    2.95  0.0135
     1200    1.96  0.014
     1600    1.47  0.0145
     2000    1.18  0.015
     2400    0.98  0.0155
     2800    0.84  0.016
     3200    0.74  0.0165
     3600    0.65  0.017
     4000    0.59  0.0175
     4400    0.54  0.018
     4800    0.49  0.0185
     5200    0.45  0.019
     5600    0.42  0.0195 /
--   PGAS   BGAS   VISGAS
PVDS
      400    5.9   0.013
      800    2.95  0.0135
     1200    1.96  0.014
     1600    1.47  0.0145
     2000    1.18  0.015
     2400    0.98  0.0155
     2800    0.84  0.016
     3200    0.74  0.0165
     3600    0.65  0.017
     4000    0.59  0.0175
     4400    0.54  0.018
     4800    0.49  0.0185
     5200    0.45  0.019
     5600    0.42  0.0195 /
ROCK
 4000.00        .30E-05 /
DENSITY
 52.0000  64.0000   .04400 /
SDENSITY
 0.3   /
RPTPROPS
 'SOF2' 'SWFN' 'SGFN' 'PVTO'
 'PVTW' 'PVTG' 'DENSITY'  'ROCK'  'DIFFC'
 'LANGMUIR'  /
REGIONS
EQUALS
 'FIPNUM' 1  1 8 1 8 1 1 /
 'FIPNUM' 2  1 8 1 8 2 2/
/
```

SOLUTION

```
SOLUTION  ==========================================================
EQUIL
 4015 528 3000 0 /
--
-- Initial 40 per cent of the fracture gas is CO2
--
SOLVFRAC
 128*0.4 /
RPTSOL
 'PRES'  'SWAT'  'SGAS'  'FIP=2' 'EQUIL'  'SSOL'
 'FIPSOL' 'GASCONC'  /
```

SUMMARY

```
SUMMARY     ==========================================================
=
FPR
FWGR
FGPR
FWPR
FNPR
FNIR
FCGC
FCSC
RGIP
/
RCGC
 /
RCSC
 /
RNIP
 /
BCGC
 1 1 1 /
/
BCSC
 1 1 1 /
/
WBHP
/
WWGR
/
WGPR
/
WNPR
/
RUNSUM
SEPARATE
```

SCHEDULE

```
SCHEDULE  ==========================================================
RPTSCHED
 'PRES'  'SWAT'  'SGAS'  'RESTART=2' 'FIP=2'
 'WELLS=2' 'CPU=2' 'NEWTON=2' 'SSOL'
 'FIPSOL' 'GASCONC'  /
--
-- Initial period with no production
--
TSTEP
 1 9 90 /
WELSPECS
'P'  'G'   8  8  4000  'GAS'  /
'I'  'G'   1  1  4000  'GAS'  /
/
COMPDAT
'P'    2*   2   2 'OPEN'   0  .000000  .5000  .00000   .0000 0.000E-
01 /
'I'    2*   2   2 'OPEN'   0  .000000  .5000  .00000   .0000 0.000E-
01 /
/
WCONPROD
'P'  'OPEN'  'WRAT'     1*         1000.0      100000.00000
     1*          1*      50.000  0.000000   1*  0.00000000/
/
WCONINJ
 'I'  'GAS'  'SHUT'  'RATE'  1000.0  /
/
--
-- Initial production
--
NEXTSTEP
 0.1 /
TSTEP
 1.0 9.0 90.0 900.0 1000.0
/
--
-- Inject Co2
--
WCONINJ
 'I'  'GAS'  'OPEN'  'RATE'  100000.0  /
/
```

```
WSOLVENT
 'I' 1.0 /
/
NEXTSTEP
 0.1 /
TSTEP
 1.0 9.0 90.0 900.00 3*1000
/
--
-- Blow down
--
WCONPROD
'P'  'OPEN'  'GRAT'     1*         1000.0      100000.00000
     1*          1*      50.000  0.000000   1*  0.00000000/
/
WCONINJ
 'I'  'GAS'  'SHUT'  'RATE'  100000.0  /
/
NEXTSTEP
 0.1 /
TSTEP
 1.0 9.0 90.0 900.00 3*1000
/
END
```

Summary of keywords ECLIPSE 300

RUNSPEC section

The RUNSPEC keywords are:

Table 12.16 RUNSPEC Keywords ECLIPSE 300

Keyword	Description
COAL	Specifies that the Coal Bed Methane model is to be used.
CBMOPTS	Options for Coal Bed Methane.
DUALPORO	Specify a dual porosity run. **Do not use** DUALPERM
NMATRIX	Input number of matrix cells (multi porosity option)

GRID section

The GRID keywords are:

Table 12.17 GRID Keywords ECLIPSE 300

Keyword	Description
DIFFMMF	Modify the Matrix-Fracture diffusivity.
SIGMA	Input the dual porosity SIGMA factor.
SIGMAV	Input the SIGMA factor on a cell by cell basis.
COALNUM	Assign coal regions
COALNUMR	Assign coal regions for matrix sub cells
CRNDENS	Input rock (coal) density by coal region number
ROCKDEN	Input rock (coal) density.
ROCKFRAC	Input rock (coal) fractions of the bulk volume on a cell by cell basis.

PROPS section

The PROPS keywords are:

Table 12.18 PROPS Keywords for ECLIPSE 300

Keyword	Description
LANGMEXT	Input the Langmuir volume and pressure constant for each component.
DIFFCBM	Input gas diffusion data.
RESORB	Input gas readsorption factors.
LANGMULT	Inputs a scaling factor for the adsorption capacity for each grid cell.
ROCKPAMA	Palmer-Mansoori rock compaction model.
ROCKPAME	Modified Palmer-Mansoori model made dependent on composition for coal swelling and shrinkage.
RPTPROPS	Controls output from the PROPS section. DIFFCBM outputs diffusion data. LANGMEXT outputs the extended Langmuir isotherm data.

SOLUTION section

The SOLUTION keywords are

Table 12.19 SOLUTION Keywords for ECLIPSE 300

Keyword	Description
SORBFRAC	Scale initial adsorbed gas.
SORBPRES	Specify Langmuir sorption pressure.

SUMMARY section

The following keywords control output of data typically used with the coal bed methane model in ECLIPSE 300

Table 12.20 SUMMARY data

Field	Group	Well	Region	Block	Information
FCWGPR	GCWGPR	WCWGPR			Component Gas Production Rate
FCWGPT	GCWGPT	WCWGPT			Component Gas Production Total
FCWGIR	GCWGIR	WCWGIR			Component Gas Injection Rate
FCWGIT	GCWGIT	WCWGIT			Component Gas Injection Total

If methane is component number one, the methane production rate can be output by specifying:

```
FCWGPR
1 /
```

SCHEDULE section

The SCHEDULE keywords are:

Table 12.21 SCHEDULE Keywords for ECLIPSE 300

Keyword	Description
WINJGAS	Specify nature of injection gas.
RPRKPAME	Report rock compaction multipliers (ROCKPAME) versus pressure for a given cell.

Using the ECLIPSE 300 coal bed methane model

The model is activated by specifying the keyword COAL in the RUNSPEC section. The dual porosity model should also be activated with the RUNSPEC keyword DUALPORO, and the number of coal regions should be set by item 6 of REGDIMS (default =1). For multi porosity cases, the keyword NMATRIX specifies the number of matrix cells per cell (see "Multi porosity" on page 627). It is possible to choose between two types of adsorption model: instant and time dependent. This is done by using the CBMOPTS keyword. For the instant adsorption model it is also possible to specify that no water flow is allowed in and out of a coal cell by using the CBMOPTS keyword.

Note The coal bed methane option is currently not compatible with the Thermal option or the ECLIPSE 300 Black oil option.

Grid data

Using the time dependent adsorption model and dual porosity, the first NDIVIZ/2 layers of the model represent the coal matrix and the second NDIVIZ/2 layers represent the fracture system. The input follows what is described for ECLIPSE 100 above, except:

- It is strongly recommended to use the keyword ROCKFRAC to set the coal volumes instead of specifying porosities when using the time dependent sorption model.
- The density of the coal must be supplied by the ROCKDEN keyword.
- It is possible to set coal regions with the COALNUM keyword; the default is that all matrix cells are assigned to coal region number one.
- Using the multi porosity option, the NMATOPTS keyword can be used to generate the pore volumes and transmissibilities for the matrix subcells.

Properties data

Gas and water PVT, EOS and relative permeability data need to be supplied in the usual way.

In addition the following keywords must be supplied:

Table 12.22 Properties

Keyword	Description
LANGMEXT	Defines the Langmuir isotherm for each component and each coal region
DIFFCBM	Inputs the diffusion coefficients (not used by the instant sorption model)

If the GASWAT option is used water and gas relative permeability curves are entered by WSF and GSF. It is also possible to use the three phase relative permeability curves SWFN, SGFN and SOF3. However in the case of no oil present you must still enter a dummy oil relative permeability curve. Optionally the FACTLI keyword can also be used in order to make sure that no oil phase is predicted during the simulation. Note that components predicted to be in an oil phase are not adsorbing on the coal surface.

Optional keywords are

Table 12.23 Optional keywords

Keyword	Description
RESORB	Inputs the re-adsorption factors (default=1.0)
LANGMULT	Inputs a scaling factor for the adsorption capacity for each grid cell.
ROCKPAMA	Palmer-Mansoori rock compaction model.
ROCKPAME	Modified Palmer-Mansoori model made dependent on composition for coal swelling and shrinkage.

Equilibration

Time-dependent sorption

The matrix coal gas is always initialized according to the input matrix properties. This means that the input mole fractions and pressure are used to set the gas contents according to the Langmuir adsorption capacity.

An equilibrium is obtained if the initial specified composition of the fractures equals the input matrix composition and the system has free gas. In addition the pressure of the fracture must be level or higher than the input matrix pressure. If the fractures are fully water filled, an equilibrium mole fraction and sorption pressure is calculated internally.

Please note that the resulting adsorbed mole fractions generally are not equal to the input mole fractions.

When equilibration is done by EQUIL the fractures are equilibrated in the usual way. The initial coal gas molar densities are calculated from the prevailing matrix pressure and composition as specified by COMPVD or ZMFVD.

A different sorption pressure can be specified by the keyword SORBPRES. Entering lower pressure values than what is initially computed will give a lower gas content. It is also possible to model undersaturated coal by scaling the calculated coal gas by a factor, see the keyword SORBFRAC.

Enumerated start is possible in the usual way. The input needed is:

1. Mole fractions for the gas phase, YMF values and oil phase mole fraction XMF. If no oil phase is present zero (0) XMF values should be entered.
2. Pressure
3. Gas and water saturations

The matrix gas mole fractions and the pressure are then used to initialize the matrix coal gas.

Several equilibrium regions can be used in order to specify different conditions for the matrix and the fractures.

Note If the multi porosity option is used together with time dependent sorption, a coal cell needs to connect to a pore volume cell. Adjacent coal subcells are not permitted.

Instant sorption

For the instant sorption model the adsorbed gas content is calculated from the free gas phase composition and pressure for each simulation cell. If water is present the pressure in the pore space and the composition given by COMPVD or ZMFVD is used to determine the amount of adsorbed gas. SORBPRES and SORBFRAC can currently not be used with the instant sorption model.

Output and restart

For the time-dependent model the adsorbed molar densities are output in the usual way as molar densities in the coal volumes.

Using the instant desorption model the adsorbed molar density AMSC is also output, where MLSC are the molar densities in the pore volumes (AMSC is only output when a flexible restart file has been requested)

Summary data

The SUMMARY data are:

Table 12.24 Coal bed methane example files

Keyword	Description
FCWGPR	Production rate for a specific component (given in surface volume by assuming an ideal gas)
FCWGPT	Production total for a specific component (given in surface volume by assuming an ideal gas)
WYMF	Well mole fraction for a specific component

Examples

The following data sets come with the ECLIPSE installation:

Table 12.25 Coal bed methane example files

File	Description
ECBM2.DATA	Enhanced Coal bed methane example
ECBM2I.DATA	Instant sorption and single porosity
ECBM4.DATA	ECBM with coal shrinkage and swelling effects

Compatibility Between ECLIPSE 100 Special Options

Chapter 13

Introduction

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL

ECLIPSE 100 contains of a collection of specialist extensions to the simulator. Many of these extensions are normally used alone, without the need to apply other extensions in the same run. However, it is often desirable to combine two or more special options in a single run, for example using the Gas Field Operations model with Networks. Some combinations of options are naturally incompatible (for example, Coal Bed Methane and the Foam model), while other combinations do not yet work together because these have been assigned a low priority for development.

This chapter summarizes the combinations of ECLIPSE 100 special options that are currently incompatible. For a more detailed account of the restrictions on the use of each option, please refer to the appropriate chapter in this manual.

Coal bed methane

The following options should not be required to run in combination with the Coal Bed Methane model, and are thus naturally incompatible:

- The Foam model
- The Gi Pseudo-compositional model
- The Polymer Flood model
- The Surfactant model.

Environmental tracers

Environmental Tracers are incompatible with:

- Flux Boundary Conditions
- Local Grid Refinement (if the tracer is diffusing).

Flux boundary conditions

When using flow boundary conditions, the `FLUX` file only contains oil/water/gas flows, and thus it is incompatible with any option including another fluid or conservation equation. However, the option may still be useful in cases where the other fluid is contained within the `USEFLUX` region. When pressure boundary conditions are used, the `FLUX` file also contains solution data for API or Brine Tracking, Foam and Surfactant.

Flux Boundary Conditions are incompatible with:

- Environmental Tracers
- The Foam model
- The Gas Field Operations model
- Gas Lift Optimization
- The Gi Pseudo-compositional model
- The Polymer Flood model
- The Solvent model
- The Surfactant model.

Foam model

The Foam model is incompatible with:

- The Coal Bed Methane model
- Flux Boundary Conditions.

It is also not fully integrated with the Surfactant model.

Gas calorific value control

There are limitations when using Reservoir Coupling.

Gas field operations model

The Gas Field Operations model is incompatible with:

- Flux Boundary Conditions
- Parallel options.

There are also limitations when using Reservoir Coupling.

Gas lift optimization

Gas Lift Optimization is incompatible with:

- Flux Boundary Conditions

- Reservoir Coupling.

Gi pseudo-compositional model

The Gi Pseudo-compositional model is incompatible with

- The Coal Bed Methane model
- Flux Boundary Conditions
- Local Grid Refinement
- Multi-segment Wells
- The Wellbore Friction option
- The Polymer model
- The Solvent model

Gradient option

The Gradient option is incompatible with:

- Reservoir Coupling.

When run with Local Grid Refinement, only in-place refinements are allowed.

Local grid refinement and coarsening

The Local Grid Refinement and Coarsening option is incompatible with:

- Environmental Tracers (if the tracer is diffusing)
- The Gi Pseudo-compositional model.

In addition, the Autorefinement option cannot be run with:

- Multi-segment Wells
- The Wellbore Friction option.

Multi-segment wells

Multi-segment Wells are incompatible with:

- The Gi Pseudo-compositional model
- The Autorefinement option in local grid refinement
- The Polymer Flood model
- The Solvent model
- The Wellbore Friction option.

Note also that Multi-segment Wells cannot be used with the Temperature option in ECLIPSE 100.

Network option

The Network option has no restrictions on its use with other options.

Parallel options

Parallel options cannot be used with:

- Gas Field Operations model
- `VISCD` keyword.

Polymer flood model

The Polymer Flood model is incompatible with:

- The Coal Bed Methane model
- Flux Boundary conditions
- Multi-segment wells
- The Wellbore Friction option
- The Solvent model
- The Gi pseudo-compositional model

Note also that wells are not allowed to crossflow when this model is in use.

Reservoir coupling

Reservoir Coupling is incompatible with:

- Gas Lift Optimization
- The Gradient option.

It also has limited capabilities with:

- Gas Calorific Value Control
- The Gas Field Operations model.

Solvent model

The Solvent model is incompatible with:

- Flux Boundary Conditions
- Multi-segment Wells
- The Wellbore Friction option
- The Polymer model
- The Gi pseudo-compositional model

Note also that wells are not allowed to crossflow when this model is in use.

Surfactant model

The Surfactant model is incompatible with:

- Flux Boundary Conditions.
- Relative permeability hysteresis.

Wellbore friction option

The Wellbore Friction option is incompatible with:

- The Gi Pseudo-Compositional model
- The Autorefinement option in local grid refinement
- Multi-segment Wells
- The Polymer Flood model
- The Solvent model.

Note also that the Wellbore Friction option cannot be used with the Miscible Flood and Brine Tracking options in ECLIPSE 100.

Convergence Reports

Chapter 14

Introduction

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

Both ECLIPSE 100 and ECLIPSE 300 can be made to produce reports showing how both the linear and non-linear iterations are proceeding and the methods by which time steps are selected. The form of this output varies significantly between the two simulators, and is detailed below.

ECLIPSE 100

Mnemonics SUMMARY and NEWTON in the RPTSCHED keyword control output of reports showing the convergence of the non-linear iteration and a brief summary of the state of the reservoir after each time step (including non-report times). A sample of the sort of output produced is shown below. The report is copied to the debug file if switch 1of DEBUG is greater than zero.

```
IT= 0   CNV    CELL        MAT BAL   DPRESS    DSWAT     DSGAS
OIL   .04562(  6, 10, 1)   1.0-002      .00   .00000    .00000
WAT   .00000( 10, 10, 1)   1.6-010      .00   .00000    .00000
GAS   .04780(  1,  1, 1)   6.6-003      .00   .00000    .00000
  LINIT=11 NSCHP=    7 NCHOP=     0 NSTAT1,2,3=    0  219    81 NTRA
N=   67
IT= 1   CNV    CELL        MAT BAL   DPRESS    DSWAT     DSGAS
OIL  -.09866( 10, 10, 2)   1.4-004  -170.43   .00013    .01945
WAT  -.00011( 10,  9, 3)  -2.5-006  -233.55   .00006    .00000
GAS  3.80303( 10, 10, 1)  -6.8-004  -170.60   .00013    .01982
  LINIT= 8 NSCHP=    0 NCHOP=     0 NSTAT1,2,3=    0  236    64 NTRA
N=   85
```

```
IT= 2   CNV    CELL        MAT BAL   DPRESS    DSWAT     DSGAS
OIL  -.00744( 10, 10, 2)  -7.4-008  -160.91   .00012    .01986
WAT   .00000( 10,  9, 1)   6.2-011  -222.30   .00016    .02714
GAS -1.28249( 10, 10, 1)  -1.3-008  -162.22   .00012    .03981
  LINIT= 6 NSCHP=    0 NCHOP=     0 NSTAT1,2,3=    0  243    57 NTRA
N=   51
IT= 3   CNV    CELL        MAT BAL   DPRESS    DSWAT     DSGAS
OIL  -.00011(  1,  6, 1)   3.8-007  -323.90   .00024    .00579
WAT  -.00000( 10, 10, 1)  -5.2-013  -161.05   .00012    .04081
GAS   .00658( 10, 10, 1)  -7.5-007  -161.05   .00012    .04081
  LINIT= 4 NSCHP=    0 NCHOP=     0 NSTAT1,2,3=    0  261    39 NTRA
N=   74
IT= 4   CNV    CELL        MAT BAL   DPRESS    DSWAT     DSGAS
OIL   .00001(  8,  2, 1)   2.0-008  -322.26   .00024    .00981
WAT  -.00000( 10, 10, 1)  -1.2-016  -161.03   .00012    .04079
GAS   .00004(  2,  4, 1)  -4.1-008  -324.95   .00024    .00508
STEP  19 TIME= 1277.50  DAYS (+182.5  DAYS MAXS  4 ITS) PAV=  5468.8
  PSIA
  WCT= .000 GOR=     7.53 MSCF/STB WGR=    .0000 STB/MSCF
    PR.RATE PR.TOTAL IN.RATE  IN.TOTAL   MAX SATURN CHANGE  MAX TIME
 TRNC ERR
OIL  15870. 24540186.      0.        0.  -
.041( 10, 10, 1)   .016(  9,  9, 1)
WAT      0.        0.      0.        0.   .000(  1,  1, 3)  -
.000( 10, 10, 3)
GAS 119569. 81159402. 100000.127750000.   .041( 10, 10, 1)  -
.016(  9,  9, 1)
MAX DP = -330.24( 1, 1, 3) MAX DRS= -.0000( 5, 2, 3)
```

The lines beginning ‘`IT=`’ show the progress of the non-linear iteration. The convergence errors for each phase (under `CNV`) are normalized to appear as saturation errors. The value quoted is the largest in the entire reservoir and the grid block in which it occurs is also reported. The material balance errors for each phase (under `MAT BAL`) are quoted as a fraction of the total reservoir pore volume. Pressure and saturation changes during the time step are also reported for the cell with the greatest convergence error. By default, the program continues the iteration until the largest convergence error for any phase in any grid block is less than 0.001, and the largest material balance error is less than 1E-7. These targets may be reset using the `TUNING` keyword. However such changes are **not** recommended, and can have odd results. It is, for example, quite possible for a loosening of convergence targets to result in **longer** run times. See "Formulation of the Equations" on page 341 for a precise definition of `CNV` and MAT BAL.

The lines beginning 'LINIT=' give further information about the iteration. LINIT is the number of nested factorization iterations required to solve the linearized equations for the current non-linear iteration. NSCHP is the number of saturation changes which were altered to suppress possible oscillations. NCHOP is the number of times the changes in P, R_s or R_v were reduced (again to increase stability). Any non-zero value of NSCHP or NCHOP has a detrimental effect on material balance errors for the subsequent non-linear iteration. NSTAT1, 2 and 3 are the number of grid blocks in solution states 1, 2 and 3, and NTRAN is the number of inter-state transitions. The significance of the solution states is as follows:

Table 14.1 Solution states

State	Description
State 1	there is no liquid hydrocarbon phase. R_v, the oil-gas ratio in the gas phase, is a variable.
State 2	there are both liquid and gaseous hydrocarbon phases. S_{gas}, the saturation of the gaseous phase, is a variable.
State 3	there is no gaseous hydrocarbon phase. R_s, the gas-oil ratio in the liquid phase, is a variable.

Once the non-linear equations have been solved, ECLIPSE prints a brief summary of the state of the reservoir at the end of the time step. On the first line (beginning STEP) are the time step number, the current time, the length of the time step just finished, and a mnemonic which indicates the reason why that time step length was selected. Possible values of this mnemonic are:

Table 14.2 Mnemonic values

Mnemonic	Description
INIT	Either the 1st time step in the run, or a maximum value for the next time step was set using keyword TUNING or NEXTSTEP.
TRNC	Time step was set in order to keep estimate of time truncation errors within specified targets.
MINF	Time step was determined by the minimum permitted ratio between one time step and the next (default 0.3 - see TUNING).
MAXF	Time step was determined by the maximum permitted ratio between one time step and the next (default 3.0 - see TUNING).
MINS	Time step is the minimum permitted length (default 0.1 DAYS, see TUNING).
MAXS	Time step is the maximum permitted length (default 365 DAYS, see TUNING).
REPT	Time step was set so as to end at the next report time.
HALF	Time step is half the interval to the next report time (if time step would otherwise be larger, but not big enough to reach report time, ECLIPSE reduces the time step to half the remaining interval).
CHOP	If the non-linear equations do not converge, a new time step a tenth the size (see TUNING) is tried (regardless of MINS or MINF limits).

Table 14.2 Mnemonic values

Mnemonic	Description
DIFF	Following a CHOP the time step is expanded cautiously to avoid immediate subsequent chops as the simulation proceeds through a difficult patch. See TUNING (TFDIFF default = 1.25).
THRP	Time step was determined by the maximum throughput ratio THRUPT (see TUNING).
SATM	Time step chosen so that the maximum expected saturation change in any cell is DSTARG, set in the IMPES keyword (default = 0.05). This applies only if the IMPES solution method is being used.
MAXP	Time step chosen so that the maximum expected pressure change in any cell is TRGDPR, set in the TUNING keyword.
PCHP	Time step chopped because the pressure change was greater than DPMAX, set in the IMPES keyword (default = 200 psi). This applies only if the IMPES solution method is being used.
NETW	Time step was determined by the network balancing error controls set in items 6 - 8 in keyword NETBALAN. This applies only to the Network Option.
EMTH	Time Step was set so as to end at the end of the month. This applies only to the GOR Penalty option (see keyword WGORPEN) and the Gas Field Operations model.
LGRC	Time step was set in order to keep the estimated fluid-in-place error at the target value TRGFIP set in keyword TUNING. This applies only to the Local Grid Refinement Option.
SURF	Time step was set in order to keep the estimated surfactant concentration change at the target value TRGSFT set in keyword TUNING. This applies only to the Surfactant model.
WCYC	Time step was set to coincide with a well cycling on/off time, or was restricted when a cycling well turned on (see keyword WCYCLE).
SLVR	Time step in a Reservoir Coupling master run is limited by a report date in a slave reservoir.
SLVC	Time step in a Reservoir Coupling master run is limited by the allowed flow rate change in a slave.
MAST	Time step in a Reservoir Coupling slave run is limited to reach the end of the master's time step, or the time step in either a slave or master run is limited by the master time step on the Reservoir Coupling file.
NLTR	Time step has been chopped due to problems converging non-linear tracer equations. For example, when adsorption is used.
EFF+	Time step has been selected due to efficiency.
EFF-	Time step has been selected due to efficiency, but altered slightly to provide new data for the optimization algorithm.
EFFT	Time step has been selected due to efficiency, but selection has been determined by a non-linear tracer.
ACTN	Time step has been selected to try and reduce the amount by which an ACTION is violated.

Table 14.2 Mnemonic values

Mnemonic	Description
DLYA	Time step has been selected to enact a delayed action at the appropriate time.
RAIN	Time step has been selected to coincide with the change of rate at a constant flux aquifer.
MAXW	Time step is the maximum permitted length following a well modification (see TMAXWC in the TUNING keyword).

On the remainder of the line are the number of Newton iterations required to solve the non-linear equations, the hydrocarbon pore volume weighted field average pressure, and the total water cut, gas-oil ratio and water-gas ratio for field production.

This initial line is followed by a table showing the production and injection rates and totals for each phase, together with the maximum saturation changes and the locations of the grid blocks in which they occur. Also shown are maximum values of the estimated time truncation errors and the locations of the grid blocks in which they occur. Time truncation errors are normalized to appear as saturation errors. This has the side effect of making them appear much more important than they are in fact. In practice, time truncation errors that occur early in a run have a small and rapidly decreasing effect on the later results.

After the table is a single line showing the maximum changes in pressure, R_s and R_v, together with the locations of the grid blocks in which they occur.

Time truncation errors are calculated as

$$\text{TIME TRNC ERR} = dS - \frac{dS_p}{dt_p} dt \qquad \text{[EQ 14.1]}$$

where

dS is the saturation change during the current time step, dt, and

dS_p is the saturation change during the previous time step, dt_p.

The Maximum value is taken over all cells in the reservoir. A large time truncation error arises when a saturation change is significantly different from that expected.

Output of cells with convergence problems

It is possible to output information to the RESTART file providing details of cells that are causing convergence problems. At every non-linear iteration the worst cells have their counter increased by one. ECLIPSE 100 outputs two sets of mnemonics to the RESTART file. The first set provides the worst convergence cells as measured by the magnitude of the residual of every equation, whereas the second set provides the worst cells based on the magnitude of the Newton increments for every variable.

This option is activated by specifying the mnemonic 'CONV=integer' in keyword RPTRST. By default, this 'integer' value is set to 1, so that the single worst cell in the run is focused on. Modifying this 'integer' value alters the number of worst cells.

Depending on the phases present in the model and the special options used, different mnemonics are output to the RESTART file. Specifying 'CONV=integer' in the RPTRST outputs all relevant mnemonics for the particular model. The possible mnemonics are:

Table 14.3 Possible mnemonics

Mnemonic	Description
CNV_OIL	Residual of oil equation
CNV_WAT	Residual of water equation
CNV_GAS	Residual of gas equation
CNV_PLY	Residual of polymer equation (POLYMER option)
CNV_SAL	Residual of brine equation (BRINE option when used with polymer)
CNV_SOL	Residual of solvent equation (SOLVENT option)
CNV_GGI	Residual of Gi Pseudo-Compositional model (GIMODEL option)
CNV_DPRE	Pressure increment
CNV_DWAT	Water saturation increment
CNV_DGAS	Gas saturation increment / R_v increment / R_s increment
CNV_DPLY	Polymer concentration increment
CNV_DSAL	Brine concentration increment
CNV_DSOL	Solvent concentration increment
CNV_DGGI	Dry injection gas parameter (Gi) increment

Note The active variables for each grid block depend on the solution state (see Table 14.1). CNV_DGAS therefore denotes a composite mnemonic reporting the cells with the largest changes in gas saturation, oil-gas ratio in the gas phase, or gas-oil ratio in the liquid phase.

For models with local grid refinements the worst converging cells are evaluated separately in the global grid and in each local grid. The reason is that ECLIPSE 100 by default solves local grids separately using local time stepping. For each grid (global and local) the worst converging cells have their counter increased by 1 after each iteration. The number of cells to focus on in a local grid is taken as the number entered in the CONV mnemonic in RPTRST weighted by the ratio of active cells in the local grid to the total number of active cells including global host cells.

ECLIPSE 300

The simulator outputs information concerning convergence and time steps to the log file (and echoes this to the `PRINT` file). A section of this might take the following form:

```
       Time Step GOR   WCT   OPR   WPR  GPR    FPR   WIR   GIR  It
-----  ---- ---- ---- ----  ----- ----- ---- ------ ----- ----- --
Init;  10.0 10.0 6.46 .026  959.3 26.57 6200  3530.3 340.0 4700 2
SCT ;  25.0 15.0 6.48 .027  955.7 26.76 6200  3524.1 340.0 4700 1
SCT ;  41.0 16.0 6.51 .027  950.9 26.98 6200  3518.1 340.0 4700 1
SCT ;  59.0 18.0 6.54 .027  946.8 27.23 6200  3511.7 340.0 4700 1
SCT ;  80.0 21.0 6.57 .028  942.5 27.51 6200  3504.4 340.0 4700 1
SCT ; 105.0 25.0 6.61 .028  937.6 27.79 6200  3495.8 340.0 4700 1
SCT ; 136.0 31.0 6.65 .029  932.0 28.19 6200  3485.1 340.0 4700 1
SCT ; 175.0 39.0 6.70 .030  925.2 28.73 6200  3471.9 340.0 4700 1
SCT ; 225.0 50.0 6.76 .031  917.0 29.42 6200  3455.2 340.0 4700 1
Max ; 285.0 60.0 6.83 .032  907.0 30.18 6200  3435.6 340.0 4700 1
HRep; 325.0 40.0 6.92 .032  895.8 30.25 6200  3422.9 340.0 4700 1
Rep ; 365.0 40.0 6.97 .033  888.3 30.40 6200  3410.1 340.0 4700 1
```

The step codes at the left of each line contain the reason for each step. The full set is:

```
ADA  : Maximum increase factor for Appleyard time-step selector.
       The number of Newton iterations will be appended to this step
       code. If this number is greater than 9 a '+' wil be used.
CFL  : Stability limited
EFF+ : Timestep chosen by efficient timestep selection method (ZIPPY2)
EnC  : Energy Change (thermal)
HRep : Half step to report
Init : Initial step
Max  : Maximum step
MaxW : Maximum step after well modification
MDF  : Max decrease factor
MIF  : Max increase factor
Min  : Minimum step
NIte : Number of Iterations
NVLP : Appleyard time-step selector based on envelope function
PCT  : Pressure change target
Redo : Redoing the timestep due to a chop
Redu : Reduced step
Rep  : Step to report
SAC  : Saturation Change (thermal)
SCT  : Solution change target
SST  : Solid solution change target (CO2STORE option)
Spec : Specified next step
TEC  : Temperature Change (thermal)
TPT  : Throughput target
TTE  : Time Trunction Error
WAG  : Step to WAG change
```

Solution change target (SCT)

Since our primary solution variables are molar densities, it is natural to use these to control the convergence. However, large saturation changes may occur for very small changes in composition, due to the problems of near critical states mentioned in "Saturation Functions" on page 869, so the use of an effective saturation is preferred. Consider the expression:

$$\Delta S' = \frac{dU^T}{dm_c} \cdot \Delta m_c \qquad \text{[EQ 14.2]}$$

where

$$u^T = \frac{m_t L}{b_o^m} + \frac{m_t V}{b_g^m} + \frac{m_w}{b_w^m} \qquad \text{[EQ 14.3]}$$

where

u^T is the fluid volume per unit pore volume

$\Delta S'$ is thus the change in fluid volume per unit pore volume for a given molar density change, that is the effective saturation change for the molar density change.

In all cases the maximum solution change over a time step is monitored and compared with the specified target. In the `IMPES` and `AIM` cases this is applied in addition to the throughput targets. The simulator will attempt to increase the time step as long as it is within its current targets, up to the maximum time step allowed. All these elements of the time stepping algorithm can be modified using the `TSCRIT` keyword, and some can be modified using the `TUNING` keyword.

Throughput target (TPT)

The total flow of a fluid (or component) out of a block during a time step, divided by the amount of that fluid (or component) in the grid block at the start of the time step defines a throughput ratio.

These throughput ratios usually act as the main time stepping control in `IMPES` and `AIM` modes. (Throughput ratios are closely related to the onset of instabilities in explicit algorithms.) The time steps are increased or decreased to match a target maximum throughput ratio. This is a maximum taken over all cells and fluids (or components), and is quite conservative.

Both maximum solution change and throughput ratios are subject to limits. The simulator chooses the time step to attempt to match the target. If the actual value is above this, the step is still accepted, unless the target is exceeded, in which case the time step is reduced.

Time truncation error (TTE)

A further control on the time step is provided by the time truncation error (TTE) control. A comparison is made in the change in the solution over a step,

$$\Delta X_a = X^{T+\Delta T} - X^T \qquad \text{[EQ 14.4]}$$

with the change estimated from the rate of change at the previous step:

$$\Delta X_e = \frac{dX}{dT} \cdot \Delta T \qquad \text{[EQ 14.5]}$$

where $\frac{dX}{dT}$ is the change per unit time for the previous step.

The time truncation error (TTE) is then estimated from the value of

$$\left|\Delta X_a - \Delta X_e\right| \cdot \frac{dU^T}{dX} \qquad \text{[EQ 14.6]}$$

where $\frac{dU^T}{dX}$ is used to normalize into dimensionless units of order 1.

Note that if the solution is changing at a constant rate, ΔX_e equals ΔX_a, and there is no TTE limitation.

Minimum step codes

Minimum step codes may start with the two letters `Mn` followed by an abbreviated version of one of the above - for example `MnTP` - where the second part indicates what constraint would have applied if the minimum step had not been imposed.

```
MnNC : Minimum time step, No Convergence
MnSC : Minimum time step due to Solution change
MnTP : Minimum time step due to Throughput change
MnPC : Minimum time step due to Pressure change
MnTT : Minimum time step due to Time Truncation error
MnTE : Minimum time step due to Temperature change (thermal)
MnEn : Minimum time step due to Energy change (thermal)
MnSA : Minimum time step due to Saturation change (thermal)
```

Maximum decrease factors

Similarly, time step sizes may be decreased by the maximum decrease allowed for a certain target.

```
MdSC : Maximum decrease due to Solution change
MdTP : Maximum decrease due to Throughput change
MdPC : Maximum decrease due to Pressure change
MdTT : Maximum decrease due to Time Truncation error
MdTE : Maximum decrease due to Temperature change (thermal)
MdEn : Maximum decrease due to Energy change (thermal)
MdSA : Maximum decrease due to Saturation change (thermal)
```

Only lines containing the step code contain the “;” character. This may therefore be used to find such lines, or extract them from the `PRT` file using a utility such as `grep` on UNIX systems.

Reduction of time step size may be necessary when the simulator has difficulty converging. The format in the output file when such a reduction is in force is as follows:

```
@--Message at   250.0000 Days 8 Sep 1990
@  Reducing SCT  time step to  15.5000
@  Throughput 1.66718 > limit 1.00000
```

This states that the current time step, whose size was set to attain a solution change target, was reduced because the cell throughput was greater than the prescribed limit. Reducing a time step allows the simulator to accurately calculate events that are occurring on a smaller time scale than the original time step size that was estimated based on the solution change target.

If RPTPRINT argument 9 is set to 1, information is also output on the number of cells in each phase/implicitness state. A line is provided for each non-linear iteration, for example:

```
3/OW/GW/W 253 0 63 0 3/OW/GW/W/OG/O/G 5 0 3 0 0 0 0
3/OW/GW/W 240 0 76 0 3/OW/GW/W/OG/O/G 5 0 3 0 0 0 0
```

The first set of numbers are for explicit mobility cells in three-phase, two-phase oil-water, two-phase oil-gas, and water filled states. The second set of numbers are for implicit mobility cells in a similar set of phase states.

Output of cells with convergence problems

It is possible to output information to the RESTART file providing details of cells that are causing convergence problems. At every non-linear iteration, and for the volume balance residual equation as well as the pressure update, the worst cells have their counter increased by one.

This can be done using the mnemonic 'CONV=integer' with keyword RPTRST. By default, this 'integer' value is set to 10, so that the worst 10 cells in the run are focused on. Modifying this 'integer' value alters the number of worst cells. This could prove to be useful when using the external program, FloViz, for instance. Loading in the RESTART files, and animating the run, could enable problem cells in the run to be pin-pointed.

Example

If 'CONV=40' is used with the RPTRST keyword, then two separate records are output to the formatted RESTART files (for t>0):

```
'CONV_VBR'  40 'REAL'
.....

'CONV_PRU'  40 'REAL'
.....
```

Here, 'CONV_VBR' focuses on the worst 40 cells based on the volume balance residual equation' and CONV_PRU' focuses on the worst 40 cells based on the pressure updates.

Diffusion

Chapter 15

Introduction

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

Molecular diffusion can be modeled in both ECLIPSE 100 and ECLIPSE 300. This chapter describes the keywords that can be used; the diffusion coefficients; and the diffusivity calculations.

Keywords

RUNSPEC section

The RUNSPEC keywords are:

Table 15.1 RUNSPEC keywords

Keyword	Description
DIFFUSE	Enables molecular diffusion

GRID section

The GRID section keywords are:

Table 15.2 GRID section keywords

Simulator	Keyword	Description
	DIFFMMF	Matrix-fracture diffusivity multipliers
	DIFFMR	Diffusivity multipliers in the radial direction
ECLIPSE 100	DIFFMR-	Diffusivity multipliers in the negative radial direction
	DIFFMTHT	Diffusivity multipliers in the theta direction
ECLIPSE 100	DIFFMTH-	Diffusivity multipliers in the negative theta direction
	DIFFMX	Diffusivity multipliers in the X-direction
ECLIPSE 100	DIFFMX-	Diffusivity multipliers in the negative X-direction
	DIFFMY	Diffusivity multipliers in the Y-direction
ECLIPSE 100	DIFFMY-	Diffusivity multipliers in the negative Y-direction
	DIFFMZ	Diffusivity multipliers in the Z-direction
ECLIPSE 100	DIFFMZ-	Diffusivity multipliers in the negative Z-direction
ECLIPSE 100	MULTREGD	Multiplies the diffusivity between flux or MULTNUM regions

EDIT section

The EDIT section keywords are:

Table 15.3 EDIT section keywords

Simulator	Keyword	Description
	DIFFR	Diffusivities in the radial direction
	DIFFTHT	Diffusivities in the theta direction
	DIFFX	Diffusivities in the X-direction
	DIFFY	Diffusivities in the Y-direction
	DIFFZ	Diffusivities in the Z-direction
ECLIPSE 100	MULTREGD	Multiplies the diffusivity between flux or MULTNUM regions

PROPS section

The PROPS section keywords are:

Table 15.4 PROPS keywords

Simulator	Keyword	Description
ECLIPSE 300	DIFFAGAS	Activity corrected gas diffusion coefficients
ECLIPSE 300	DIFFAOIL	Activity corrected oil diffusion coefficients
ECLIPSE 100	DIFFC	Molecular diffusion data for each PVT region
ECLIPSE 300	DIFFCGAS	Gas diffusion coefficients
ECLIPSE 300	DIFFCOIL	Oil diffusion coefficients
ECLIPSE 300	DIFFCWAT	Water diffusion coefficients (GASWAT or CO2STORE option)
ECLIPSE 300	DIFFCWG	Water-Gas cross phase diffusion coefficients (GASWAT or CO2STORE option)
ECLIPSE 300	DIFFCGW	Gas-Water cross phase diffusion coefficients (GASWAT or CO2STORE option)
ECLIPSE 300	DIFFCOG	Oil-Gas cross phase diffusion coefficients
ECLIPSE 300	DIFFCGO	Gas-Oil cross phase diffusion coefficients
ECLIPSE 100	DIFFDP	Restricts molecular diffusion in dual porosity runs
ECLIPSE 300	DIFFTGAS	Thermal diffusion coefficients for gas
ECLIPSE 300	DIFFTOIL	Thermal diffusion coefficients for oil

SCHEDULE section

The SCHEDULE keywords are:

Table 15.5 SCHEDULE keywords

Keyword	Description
DIFFMMF	Matrix-fracture diffusivity multipliers

Diffusion coefficients: ECLIPSE 100

ECLIPSE 100 allows diffusion of both gas and oil components within both the oil and gas phases. For example, in a gas condensate model with no oil phase present, gas diffuses from a low R_v region to a high R_v region, and vaporized oil flows in the opposite direction from the high R_v to the low R_v region (R_v is the vaporized oil-gas ratio). Consider a gas component in the gas phase. The moles of gas and oil within the gas phase are given by:

$$M_{\text{gas}} = \frac{S_g \cdot \rho_g}{B_g \cdot \text{MW}_g} \qquad M_{\text{oil}} = \frac{R_v \cdot S_g \cdot \rho_o}{B_g \cdot \text{MW}_o} \qquad \text{[EQ 15.1]}$$

hence the gas total molar concentration c is

$$c_g = \frac{S_g}{B_g}\left(\frac{\rho_g}{\text{MW}_g} + \frac{R_v \cdot \rho_o}{\text{MW}_o}\right) = \frac{S_g \cdot \rho_g}{B_g \cdot \text{MW}_g}(1 + R_v F) \qquad \text{[EQ 15.2]}$$

The gas mole fraction is:

$$x_g = \frac{\rho_g / \text{MW}_g}{\rho_g / \text{MW}_g + R_v \rho_o / \text{MW}_o} = \frac{1}{1 + R_v F} \qquad \text{[EQ 15.3]}$$

with

$$F = \frac{\rho_o \cdot \text{MW}_g}{\rho_g \cdot \text{MW}_o} \qquad \text{[EQ 15.4]}$$

where

S_g is the gas saturation

B_g is the gas formation volume factor

ρ_g is the surface density of gas

ρ_o is the surface density of oil

MW_g is the molecular weight of gas

MW_o is the molecular weight of oil

The diffusive flow of gas at surface conditions F_{gg} is given by (for more information see [Ref. 7]):

$$F_{gg} = \frac{\text{MW}_g \cdot J_g \cdot A \cdot \phi}{\rho_g} \qquad \text{[EQ 15.5]}$$

and the gas-in-gas diffusion coefficient D_{gg} is given by:

$$J_g = -c_g \cdot D_{gg} \cdot \frac{dx_g}{dl} \qquad \text{[EQ 15.6]}$$

with

$$dx_g = -\frac{F}{(1 + R_v F)^2} dR_v$$

Substituting gives the diffusive flow of gas:

$$F_{gg} = \frac{S_g}{B_g} \cdot \frac{F}{1 + R_v F} \cdot D_{gg} \cdot \frac{dR_v}{dl} \cdot A \cdot \phi \qquad \text{[EQ 15.7]}$$

where

J_g is the molar flux of gas per unit area

$\frac{dx_g}{dl}$ is the mole fraction gradient of gas

A is the area of flow

ϕ is the porosity

In the simulator the diffusive flow of gas from cell i to j is:

$$F_{gg} = \frac{S_g}{B_g} \cdot \frac{F}{(1 + R_v F)} \times D_{gg} \times \frac{(R_{vj} - R_{vi})}{d} \times A \times \phi$$

or

$$F_{gg} = -\frac{S_g}{B_g} \cdot \frac{F}{1 + R_v F} \cdot D_{gg} \cdot (R_{vi} - R_{vj}) \cdot T_d \qquad \text{[EQ 15.8]}$$

where

T_d is the diffusivity = $A\phi/d$

d is the distance between cell centers

R_{vi} is the vapor oil-gas ratio for cell i

R_{vj} is the vapor oil-gas ratio for cell j

S_g is taken as $\mathrm{MIN}(S_{gi}, S_{gj})$

B_g is taken as $(B_{gi} + B_{gj})/2$

R_v is taken as $(R_{vi} + R_{vj})/2$

For the oil-in-gas diffusive flow relationship F_{og}:

$$F_{og} = \frac{\mathrm{MW}_o \cdot J_o \cdot A \cdot \phi}{\rho_o}$$

and the oil-in-gas diffusion coefficient D_{og} is given by:

$$J_o = -c_g \cdot D_{og} \cdot \frac{dx_o}{dl}$$

$$c_g = \frac{S_g}{B_g} \cdot \frac{\rho_g}{\mathrm{MW}_g}(1 + R_v F)$$

$$x_o = \frac{R_v F}{1 + R_v F}$$

$$dx_o = \frac{F}{(1 + R_v F)^2} dR_v$$

$$F_{og} = -\frac{S_g}{B_g} \cdot \frac{F}{(1+R_v F)} \times D_{og} \times \frac{(R_{vj}-R_{vi})}{d} \times A \times \phi$$

$$F_{og} = \frac{S_g}{B_g} \times \frac{F}{1+R_v F} \times D_{og} \times (R_{vi}-R_{vj}) \times T_d$$

where

J_o is the molar flux of oil per unit area

$\frac{dx_o}{dl}$ is the mole fraction gradient of oil

For the oil-in-oil diffusive flow relationship F_{oo}:

$$F_{oo} = \frac{\mathrm{MW}_o \cdot J_o \cdot A \cdot \phi}{\rho_o}$$

and the oil-in-oil diffusion coefficient D_{oo} is given by:

$$J_o = -c \cdot D_{oo} \cdot \frac{dx_o}{dl}$$

$$c = \frac{S_o}{B_o} \cdot \frac{\rho_o}{\mathrm{MW}_o}\left(\frac{R_s+F}{F}\right)$$

$$x_o = \frac{F}{F+R_s}$$

$$dx_o = -\frac{F}{(F+R_s)^2} dR_s$$

$$F_{oo} = \frac{S_o}{B_o} \cdot \frac{1}{(F+R_s)} \times D_{oo} \times \frac{(R_{sj}-R_{si})}{d} \times A \times \phi$$

$$F_{oo} = -\frac{S_o}{B_o} \times \frac{1}{(F+R_s)} \times D_{oo} \times (R_{si}-R_{sj}) \times T_d$$

For the gas-in-oil diffusive flow relationship F_{go}:

$$F_{go} = \frac{\mathrm{MW}_g \cdot J_g \cdot A \cdot \phi}{\rho_g}$$

and the gas-in-oil diffusion coefficient D_{go} is given by:

$$J_g = -c_o \cdot D_{go} \cdot \frac{dx_g}{dl}$$

$$c_o = \frac{S_o}{B_o} \cdot \frac{\rho_o}{\mathrm{MW}_o}\left(\frac{F+R_s}{F}\right)$$

$$x_g = \frac{R_s}{F+R_s}$$

$$dx_g = \frac{F}{(F+R_s)^2} dR_s$$

$$F_{go} = -\frac{S_o}{B_o} \cdot \frac{1}{(F+R_s)} \times D_{go} \times \frac{(R_{sj}-R_{si})}{d} \times A \times \phi$$

$$F_{go} = \frac{S_o}{B_o} \times \frac{1}{(F+R_s)} \times D_{go} \times (R_{si}-R_{sj}) \times T_d$$

The diffusive flow of gas at surface conditions is then given by:

$$F_{gg} = \frac{\mathrm{MW}_g \cdot J_g \cdot A \cdot \phi}{\rho_g} \qquad \text{[EQ 15.9]}$$

where

J_g is the molar flux of gas per unit area

c is the total molar concentration of the gas phase

D_{gg} is the gas-in-gas diffusion coefficient

$\frac{dx_g}{dl}$ is the mole fraction gradient of gas

MW_g is the molecular weight of the gas

A is the area of flow

ϕ is the porosity

ρ_g is the surface density of the gas.

The moles of gas and oil within the gas phase is given by:

$$M_{\text{gas}} = \frac{S_g \cdot \rho_g}{B_g \cdot \mathrm{MW}_g} \qquad M_{\text{oil}} = \frac{R_v \cdot S_g \cdot \rho_o}{B_g \cdot \mathrm{MW}_o} \qquad \text{[EQ 15.10]}$$

hence

$$c = \frac{S_g}{B_g}\left(\frac{\rho_g}{\mathrm{MW}_g} + \frac{R_v \cdot \rho_o}{\mathrm{MW}_o}\right) = \frac{S_g \cdot \rho_g}{B_g \cdot \mathrm{MW}_g}(1 + R_v F) \qquad \text{[EQ 15.11]}$$

$$x_g = \frac{\rho_g / \mathrm{MW}_g}{\rho_g / \mathrm{MW}_g + R_v \rho_o / \mathrm{MW}_o} = \frac{1}{1 + R_v F} \qquad \text{[EQ 15.12]}$$

with

$$F = \frac{\rho_o \cdot \mathrm{MW}_g}{\rho_g \cdot \mathrm{MW}_o} \qquad \text{[EQ 15.13]}$$

where

S_g is the gas saturation

B_g is the gas formation volume factor

ρ_g is the surface density of gas

ρ_o is the surface density of oil

MW_g is the molecular weight of gas

MW_o is the molecular weight of oil

R_v is the vaporized oil-gas ratio.

Substituting equations [EQ 15.1], [EQ 15.12], [EQ 15.13] into equation [EQ 15.9] gives the diffusive flow of gas:

$$F_{gg} = \frac{S_g}{B_g} \cdot \frac{F}{1 + R_v F} \cdot D_{gg} \cdot \frac{dR_v}{dl} \cdot A \cdot \phi \qquad \text{[EQ 15.14]}$$

In the simulator the diffusive flow of gas from cell i to j is:

$$F_{gg} = -\frac{S_g}{B_g} \cdot \frac{F}{1 + R_v F} \cdot D_{gg} \cdot (R_{vi} - R_{vj}) \cdot T_d \qquad \text{[EQ 15.15]}$$

where

T_d is the diffusivity = $A\phi / d$

d is the distance between cell centers

R_{vi} is the vapor oil-gas ratio for cell i

R_{vj} is the vapor oil-gas ratio for cell j

S_g is taken as $\mathrm{MIN}(S_{gi}, S_{gj})$

B_g is taken as $(B_{gi} + B_{gj})/2$

R_v is taken as $(R_{vi} + R_{vj})/2$

D_{gg} is the gas-in-gas diffusion coefficient.

"Diffusivity calculations" on page 227 describes the calculation of the diffusivities in more detail.

Similar expressions can be derived for the oil-in-gas, gas-in-oil and oil-in-oil diffusive flows:

$$F_{og} = \frac{S_g}{B_g} \cdot \frac{1}{1 + F R_v} \cdot D_{og} \cdot (R_{vi} - R_{vj}) \cdot T_d$$

$$F_{oo} = -\frac{S_o}{B_o} \cdot \frac{1}{F + R_s} \cdot D_{oo} \cdot (R_{si} - R_{sj}) \cdot T_d$$

$$F_{go} = \frac{S_o}{B_o} \cdot \frac{F}{F + R_s} \cdot D_{go} \cdot (R_{si} - R_{sj}) \cdot T_d \qquad \text{[EQ 15.16]}$$

Diffusion coefficients: ECLIPSE 300

There are two diffusion models that can be used in ECLIPSE 300. In the first model diffusion is driven by the concentration gradient [Ref. 7]:

$$J_i = -cD_i \frac{\partial x_i}{\partial d}. \qquad \text{[EQ 15.17]}$$

In the second model diffusion is driven by the gradient of the chemical potential:

$$J_i = -cD_i^a x_i \frac{1}{RT}\frac{\partial}{\partial d}[\mu_i - M_i G(h - h_o) + M_i D_i^T \ln(T)] \qquad \text{[EQ 15.18]}$$

where:

J_i is the molar flux of component i per unit area,

c is the total molar concentration given by $c = 1/v_m$,

v_m is the molar volume of the mixture,

D_i is the diffusion coefficient of component i,

D_i^a is the activity-corrected diffusion coefficient of component i,

D_i^T is the thermal diffusion coefficient of component i,

x_i is the mole fraction of component i,

$\frac{\partial}{\partial d}$ is the gradient in the direction of flow,

M_i is the molecular weight of component i,

G is the acceleration due to gravity,

h is the height,

h_0 is the reference height,

T is the temperature,

R is the gas constant, and

μ_i is the chemical potential of component i, given by

$$\mu_i = \mu_{i0} + \mathrm{RT}\ln(f_i) \qquad \text{[EQ 15.19]}$$

where:

μ_{i0} is the reference chemical potential, and

f_i is the component fugacity.

Therefore, there are two possible ways of specifying diffusion coefficients:

- Use normal diffusion coefficients D_i defined by keywords `DIFFCOIL` (`DIFFCWAT` for `GASWAT` or `CO2STORE` option) and `DIFFCGAS`.

- Use activity corrected diffusion coefficients D_i^a defined by keywords DIFFAOIL and DIFFAGAS. Thermal diffusion coefficients D_i^T can optionally be defined by keywords DIFFTOIL and DIFFTGAS.

Relationship between the two diffusion models

[EQ 15.17] is written in terms of concentrations, which are not ideal for diffusion in liquids and gases at high pressure. The true equilibrium is obtained when the component chemical potential is equal to the gravity potential, which are the first two terms in [EQ 15.18]. The third term is a non-equilibrium term and represents diffusion driven by a temperature gradient.

Relationship between the diffusion coefficients

For horizontal flow in isothermal systems, [EQ 15.18] can be written as:

$$J_i = -cD_i^a x_i \frac{\partial}{\partial d}\ln(f_i). \qquad \text{[EQ 15.20]}$$

Where $\partial\ln(f_i)/\partial d$ can be written as:

$$\frac{\partial}{\partial d}\ln(f_i) = \frac{\partial}{\partial\ln(x_i)}\ln(f_i)\cdot\frac{\partial}{\partial x_i}\ln(x_i)\cdot\frac{\partial x_i}{\partial d} = \frac{\partial}{\partial\ln(x_i)}\ln(f_i)\cdot\frac{1}{x_i}\cdot\frac{\partial x_i}{\partial d} \qquad \text{[EQ 15.21]}$$

and we have ignored terms $\partial\ln(f_i)/\partial x_j$. [EQ 15.20] can then be written as:

$$J_i = -\left(c\cdot D_i^a\cdot\frac{\partial}{\partial\ln(x_i)}\ln(f_i)\cdot\frac{\partial x_i}{\partial d}\right). \qquad \text{[EQ 15.22]}$$

Comparing [EQ 15.17] and [EQ 15.22], D_i^a is given by:

$$D_i^a = \frac{D_i}{\partial\ln(f_i)/\partial\ln(x_i)}. \qquad \text{[EQ 15.23]}$$

D_i^a is therefore the activity-corrected diffusion coefficient of [Ref. 7].

Equivalence at low pressures

At low pressures, the two coefficients are equal, as $f_i \to x_i\mathrm{P}$, and the predicted flows become the same, as $\partial\ln(f_i)/\partial d \to \partial\ln(x_i)/\partial d \to (1/x_i)\cdot(\partial x_i/\partial d)$.

Component flow

In a porous medium, with both oil and gas present, the molar concentration includes the saturation, S, and porosity, ϕ, so that

$$J_i = J_{io} + J_{ig} \qquad \text{[EQ 15.24]}$$

with J_{io}, the oil phase molar flux of component i given in terms of activity-corrected coefficients by:

$$J_{io} = -\left(\phi \cdot S_o \cdot b_o^m \cdot D_{io}^a \cdot x_i \cdot \frac{1}{RT} \cdot \frac{\partial}{\partial d}(RT\ln f_i - M_i Gh + M_i D_{io}^T \ln T)\right) \quad \text{[EQ 15.25]}$$

or in terms of normal diffusion coefficients:

$$J_{io} = -\left(\phi \cdot S_o \cdot b_o^m \cdot D_{io} \cdot \frac{\partial x_i}{\partial d}\right) \quad \text{[EQ 15.26]}$$

Similar formulae apply for J_{ig}, the gas phase molar flux of component i.

Note that separate oil and gas diffusion coefficients may be defined, D_{io} and D_{ig}.

For any two cells, the diffusive flow is proportional to the cross-sectional area between the cells, and inversely proportional to the distance between them. Integrating the flux and approximating the gradients between two grid blocks using differences, the diffusive flow between blocks is:

$$F_i^{\text{diff}} = F_{io}^{\text{diff}} + F_{ig}^{\text{diff}} \quad \text{[EQ 15.27]}$$

where:

$$F_{io}^{\text{diff}} = T_D D_{io}^a \left(\frac{x_i S_o b_o^m}{RT}\right)_u \Delta\,(RT\ln f_i - M_i \mathrm{G}h + M_i D_{io}^T \ln T) \quad \text{[EQ 15.28]}$$

$$F_{ig}^{\text{diff}} = T_D D_{ig}^a \left(\frac{y_i S_g b_g^m}{RT}\right)_u \Delta\,(RT\ln f_i - M_i \mathrm{G}h + M_i D_{io}^T \ln T). \quad \text{[EQ 15.29]}$$

where (x_i, y_i) are the liquid and vapor mole fractions of component i, respectively.

In terms of normal diffusion coefficients:

$$F_{io}^{\text{diff}} = T_D D_{io} (S_o b_o^m)_u \Delta x_i \quad \text{[EQ 15.30]}$$

$$F_{ig}^{\text{diff}} = T_D D_{ig} (S_g b_g^m)_u \Delta y_i. \quad \text{[EQ 15.31]}$$

Combinations such as $(x_i S_o b_o^m)_u$ are defined on the cell interface, and treated using upstreaming in the program, indicated by the subscript ‘u’.

In this simple case, T_D is given by:

$$T_D = \frac{\phi A}{d} \quad \text{[EQ 15.32]}$$

where

A is the cross-sectional area

ϕ is the porosity

d is the distance between cell centers.

T_D is a diffusive analogue of transmissibility, called the *diffusivity*, with porosities replacing permeabilities. In "Diffusivity calculations" on page 227, the diffusivities for more complex cases are considered.

Thermal diffusion and hydrostatic equilibrium calculations

The term “thermal diffusion” is used here to describe the diffusion of components caused by a temperature gradient, and should not be confused with diffusion in a thermal simulation. The thermal diffusion term is usually small, but has been included for simulating the initial conditions in a reservoir where there is a temperature gradient. In the absence of any wells and aquifers, a simulation run reaches a steady state when there is no diffusive flow. From equation [EQ 15.18], the diffusive flow will be zero when

$$\frac{\partial}{\partial d}[\mu_i - M_i g h + M_i D_i^T \ln(T)] = 0 \qquad \text{[EQ 15.33]}$$

for each phase. The second term in this equation drives the heavy components to the bottom of the reservoir. The third term drives those components with a low enthalpy / high entropy to the hottest parts of the reservoir. When this last term is included the reservoir will not be in thermodynamic equilibrium (since energy is flowing) but it is in hydrostatic equilibrium. The reservoir temperature can be set with either the `TEMPVD` keyword or the `TEMPI` keyword.

The steady state solution of [EQ 15.33] is independent of the diffusion coefficients D_i^a .

However, these coefficients control the time it takes the reservoir to reach a steady state. Large time steps can be used if the simulation is run using the full implicit method `FULLIMP`.

The cross phase diffusion option (ECLIPSE 300)

As well as diffusion of oil and gas components within the oil and gas phases, diffusion of components between the phases may be important.

In ECLIPSE 300 two models are available:

1 The activity driven diffusion. The diffusion between cells are driven by the chemical potential, thus resulting in diffusion over phase boundaries. The diffusion coefficients for the cross phase diffusion are set by combining the diffusion coefficients given by DIFFAOIL and DIFFAGAS so $Dog_i^a = \sqrt{Do_i^a \times Dg_i^a}$. The activity driven diffusion option is not compatible with the CO2STORE option

2 The molecular driven diffusion. The cross phase diffusion coefficients are set by the keywords DIFFCOG and DIFFCGO. The fluids are mixed on the border of each cell and the resulting composition is used to set up the potential for molecular diffusion of each component This model can be used by EOS runs and the CO2STORE option. For GASWAT or the CO2STORE option the water-gas and gas-water diffusion coefficients are set by the keywords DIFFCWG and DIFFCGW. The molecular diffusion option will construct a border composition by mixing the composition of neighbor cells. This means that the option generally will require more computational time than the activity driven diffusion option.

Note The activity driven diffusion option can be used with the GASWAT option.

The molecular cross phase diffusion option

As an example we consider a cell I containing oil and a cell J containing gas. An artificial border composition is constructed by

$$\tilde{m}_i = (bo \cdot x_i)_I + (bg \cdot y_i)_J$$

where bo is the oil molar density and x_i is the oil mole fraction of component i, bg is the gas molar density and y_i is the gas mole fraction of component i

The border pressure and temperature is set to

$$\tilde{P} = \frac{P_I + P_J}{2}, \tilde{T} = \frac{T_I + T_J}{2}$$

The corresponding oil and gas mole fractions and densities of the artificial border cell is denoted $\tilde{x}_i$, $\tilde{y}_i$, bo and $\tilde{bg}$. The cross phase diffusive flow between the oil in cell I and the gas in cell J for component i is given by

$$Fcross = FX_{og} + FX_{go} + FY_{og} + FY_{go}$$

where

$$FX_{og} = Diff \times Dog_i \times min(So_I, Sg_J) \times bo \times max(x_i - \tilde{x}_i, 0)$$

$$FXgo = Diff \times Dgo_i \times min(So_I, Sg_J) \times \tilde{bo} \times min(x_i - \tilde{x}_i, 0)$$

$$FY_{og} = Diff \times Dog_i \times min(So_I, Sg_J) \times \tilde{bg} \times max(\tilde{y}_i - y_i, 0)$$

$$FYgo = Diff \times Dgo_i \times min(So_I, Sg_J) \times bg \times min(\tilde{y}_i - y_i, 0)$$

So_I and Sg_J is the oil and gas saturation in cell I and J, $Diff$ is the diffusivity and Dog_i , Dgo_i is the cross phase oil-gas and gas-oil diffusion coefficients.

It is possible to activate the molecular cross phase diffusion option without also specifying phase specific diffusion coefficients.

The cross phase diffusion option (ECLIPSE 100)

As well as diffusion of oil and gas components within the oil and gas phases, diffusion of components directly from the gas phase to the oil phase may be important.

For example, if in a dual porosity system the fracture contains gas and the matrix mainly oil, then gas is expected to diffuse from the fracture to the matrix and oil from the matrix to the fracture. This process can be modeled using the cross phase diffusion option.

The cross phase diffusion process is assumed to be limited by the slower diffusion within the oil. Hence the diffusive flow takes the form of the gas-in-oil and oil-in-oil diffusion:

$$F_{gx} = \frac{S_c}{B_o} \cdot \frac{F}{(F + R_{sj})} \cdot D_{gx} \cdot (R_{si}' - R_{sj}) \cdot T_d$$

$$F_{ox} = -\frac{S_c}{B_o} \cdot \frac{1}{F + R_{sj}} \cdot D_{ox} \cdot (R_{si}' - R_{sj}) \cdot T_d$$

[EQ 15.34]

where

cell j contains mainly oil

cell i contains mainly gas

R_{si}' is the saturated R_s corresponding to the pressure of cell i

R_{sj} is the R_s of cell j

S_c is the contact saturation $\mathrm{MIN}(S_{oi} + S_{gi}, S_{oj} + S_{gj}) - \mathrm{MIN}(S_{oi}, S_{oj}) - \mathrm{MIN}(S_{gi}, S_{gj})$

D_{gx} is the gas diffusion coefficient

and D_{ox} is the oil diffusion coefficient.

Diffusivity calculations

The calculation of the diffusivity is analogous to the calculation of transmissibility, with the permeability replaced by the porosity.

ECLIPSE 100

The method of calculating the diffusivity varies depending on the type of model, and can be modified in line with the transmissibility calculation by using the `OLDTRAN` or `NEWTRAN` keyword.

Block center diffusivity calculations

In this case, X and Y direction diffusivities values are obtained using cell center separations and cross sectional areas obtained from D_x, D_y and D_z, with a dip correction.

ECLIPSE 100

This type of calculation is also referred to as `OLDTRAN`.

The expressions for the cartesian case are given below:

X-diffusivity

$$T_{D_{x_I}} = \frac{M_{D_{x_I}} A F_{\text{dip}}}{B} \qquad \text{[EQ 15.35]}$$

with:

$T_{D_{x_I}}$ is diffusivity between cell I and cell J, its neighbor in the positive X-direction

$M_{D_{x_I}}$ represents the diffusivity multiplier for cell I

A is the interface area between cell I and J

F_{dip} represents the dip correction.

A, F_{dip} and B are given by the expressions:

$$A = \frac{(D_{x_J} D_{y_I} D_{z_I} R_{ntg_I} + D_{x_I} D_{y_J} D_{z_J} R_{ntg_J})}{(D_{x_I} + D_{x_J})} \qquad \text{[EQ 15.36]}$$

$$B = \frac{\left(\frac{D_{x_I}}{\phi_I} + \frac{D_{x_J}}{\phi_J}\right)}{2} \qquad \text{[EQ 15.37]}$$

$$F_{\text{dip}} = \frac{D_h^2}{D_h^2 + D_v^2} \qquad \text{[EQ 15.38]}$$

with:

$$D_h^2 = \left(\frac{D_{x_I} + D_{x_J}}{2}\right)^2 \qquad \text{[EQ 15.39]}$$

$$D_v^2 = (d_I - d_J)^2 \text{ (where } d \text{ denotes depth)} \qquad \text{[EQ 15.40]}$$

and R_{ntg} is the net to gross ratio.

Y-diffusivity

The expression for the Y-diffusivity value is entirely analogous to the above, with the appropriate permutations of X,Y and Z.

Z-diffusivity

The expression for the Z-diffusivity value is analogous to the above, with the appropriate permutations of X, Y and Z, but differs in the following ways:

- R_{ntg} does not appear
- There is no dip correction.

Corner point diffusivity calculations

In this case, the diffusivity values are calculated from the X-, Y- and Z-projections of the mutual interface area of the two cells. An inner product is then taken with the vector distance from the cell center to the center of the cell face, so that a dip correction is automatically incorporated.

ECLIPSE 100 This type of calculation is also referred to as `NEWTRAN`.

X-diffusivity

The X-diffusivity is given by the expression:

$$T_{D_{x_I}} = \frac{M_{D_{x_I}}}{\frac{1}{T_I} + \frac{1}{T_J}} \qquad \text{[EQ 15.41]}$$

with:

$$T_I = \frac{\phi_I R_{ntg_I} \boldsymbol{A} \cdot \boldsymbol{D_I}}{\boldsymbol{D_I} \cdot \boldsymbol{D_I}} \qquad \text{[EQ 15.42]}$$

where

$$\boldsymbol{A} \cdot \boldsymbol{D_I} = A_x D_{x_I} + A_y D_{y_I} + A_z D_{z_I} \qquad \text{[EQ 15.43]}$$

and

$$\boldsymbol{D_I} \cdot \boldsymbol{D_I} = D_{x_I}^2 + D_{y_I}^2 + D_{z_I}^2 . \qquad \text{[EQ 15.44]}$$

A_x, A_y and A_z are the X-,Y- and Z- projections of the mutual interface area of cell I and cell J (which need not be neighbors in the cartesian indexing grid), and D_{x_I}, D_{y_I} and D_{z_I} are the X-, Y- and Z-components of the distance between the centricities of cell I and the center of the relevant face of cell I, these centers being obtained as the appropriate average. The expression for T_J is analogous.

Y- and Z-diffusivity

The Y-and Z- diffusivity expressions are similar, the net to gross ratio being absent from the Z expression.

Radial diffusivity calculations

In this case, the expressions used are based upon true radial diffusive flow between the pressure equivalent radii, with a dip correction.

Radial diffusivity

The radial diffusivity is:

$$T_{D_{r_I}} = \frac{M_{D_{r_I}} F_{\text{dip}}}{\frac{1}{T_I} + \frac{1}{T_J}} \qquad \text{[EQ 15.45]}$$

with:

$$T_I = \frac{\phi_I R_{ntg_I} D_{\theta_I} D_{z_I}}{D_{1p}} \qquad \text{[EQ 15.46]}$$

$$D_{1p} = \frac{R_1^2}{(R_2^2 - R_1^2)} \ln(R_1 / R_2) + \frac{1}{2} \qquad \text{[EQ 15.47]}$$

and

$$T_J = \frac{\phi_J R_{ntg_J} D_{\theta_J} D_{z_J}}{D_{2m}} \qquad \text{[EQ 15.48]}$$

$$D_{2m} = \frac{R_3^2}{(R_3^2 - R_2^2)} \ln\left(\frac{R_3}{R_2}\right) - \frac{1}{2}, \qquad \text{[EQ 15.49]}$$

where R_1 is the inner radius of cell I, R_2 the outer radius of cell I, and R_3 the outer radius of cell J.

Azimuthal diffusivities

The theta-direction diffusivity is given by:

$$T_{D_{\theta_I}} = \frac{M_{D_{\theta_I}} F_{\text{dip}}}{\frac{1}{T_I} + \frac{1}{T_J}}$$

with

$$T_I = \frac{2\phi_I R_{ntg_I} D_{z_I} \ln\left(\frac{R_2}{R_1}\right)}{D_{\theta_I}}$$

where R_1 is the inner radius, and R_2 is the outer radius. The expression for T_j is defined similarly.

ln() implies the natural logarithm throughout. This expression is also used in completing the circle in radial geometry.

Vertical diffusivities

The Z-direction diffusivity is given by:

$$T_{D_{z_I}} = \frac{M_{D_{z_I}}}{\frac{1}{T_I} + \frac{1}{T_J}}$$

with

$$T_I = \frac{\phi_I D_{\theta_I}(R_2{}^2 - R_1{}^2)}{D_{z_I}}$$

where R_1 is the inner radius, and R_2 is the outer radius. The expression for T_j is defined similarly.

Matrix-fracture diffusivity calculations

The matrix-fracture coupling diffusivities are proportional to the overall cell volume and to the matrix cell bulk volume:

$$T_D = \text{DIFFMMF} \cdot \phi_m \cdot V \cdot \sigma \qquad \text{[EQ 15.50]}$$

where

ϕ_m is the matrix cell porosity

V is the matrix cell bulk volume

and

σ is a factor to account for the matrix-fracture interface area per unit bulk volume.

Kazemi [Ref. 5] has proposed the following form for σ:

$$\sigma = 4\left(\frac{1}{l_x^2} + \frac{1}{l_y^2} + \frac{1}{l_z^2}\right) \qquad \text{[EQ 15.51]}$$

where l_x, l_y and l_z are typical X,Y and Z dimensions of the blocks of material making up the matrix volume.

In the above equation, l_x, l_y and l_z are thus not related to the simulation grid dimensions.

Alternatively, as σ acts as a multiplier on the matrix-fracture coupling, it may simply be treated as a history matching parameter.

σ can be specified as a single value for the whole field, or on a cell by cell basis using the keyword `SIGMAV`. If σ is defined on a cell by cell basis, the values corresponding to the first `NDIVIZ`/2 layers are used.

Using the molecular diffusion option

The Molecular Diffusion option is initiated by the keyword DIFFUSE in the RUNSPEC section.

The diffusivity will be calculated automatically as described above. The default diffusivities can be modified in two ways:

either

- by multiplying the calculated value by an input number

or

- by inputting the diffusivities explicitly.

The multipliers are input using the DIFFMX, DIFFMY, DIFFMZ keywords in much the same way as MULTX, MULTY, MULTZ in the GRID section; and the diffusivities can be over-written in the EDIT section using DIFFX, DIFFY, DIFFZ analogously to the TRANX, TRANY, TRANZ keywords. The diffusivities associated with the NNCs representing the matrix-to-fracture flows in dual porosity runs can be modified using the DIFFMMF keyword.

ECLIPSE 100 In addition, inter-region diffusivity multipliers may be specified using MULTREGD and MULTNUM in the GRID section.

ECLIPSE 100 The diffusion coefficients and average phase molecular weights have to be supplied with the DIFFC keyword in the PROPS section. Separate gas-in-oil and oil-in-oil diffusion coefficients need to be specified for the cross phase diffusion model. If these are not supplied, no cross phase diffuse will occur.

ECLIPSE 300 Diffusion coefficients can be entered using DIFFCOIL, DIFFCGAS, DIFFCWAT, DIFFCOG, DIFFCGO, DIFFCWG, DIFFCGW or DIFFAOIL and DIFFAGAS.

ECLIPSE 100 In dual porosity runs, the fracture-to-fracture diffusion calculation can be turned off using the DIFFDP keyword.

Dual Porosity

Chapter 16

Introduction

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

In a dual porosity reservoir, fluids exist in two interconnected systems:

- The rock matrix, which usually provides the bulk of the reservoir volume
- The highly permeable rock fractures.

If the matrix blocks are linked only through the fracture system, this conventionally could be regarded as a dual porosity single permeability system, since fluid flow through the reservoir takes place only in the fracture network with the matrix blocks acting as sources. If there is the possibility of flow directly between neighboring matrix blocks, this is conventionally considered to be a dual porosity dual permeability system. Dual porosity runs are specified by the keyword DUALPORO in RUNSPEC, while dual permeability requires the DUALPERM keyword. It should be noted that dual porosity dual permeability runs are computationally more expensive than dual porosity single permeability runs.

To model such systems, two simulation cells are associated with each block in the geometric grid, representing the matrix and fracture volumes of the cell. In ECLIPSE the porosity, permeability, depth etc. of these may be independently defined. A matrix-fracture coupling transmissibility is constructed automatically by ECLIPSE to simulate flow between the two systems due to fluid expansion, gravity drainage, capillary pressure etc.

In a dual porosity run of ECLIPSE the number of layers in the Z-direction should be doubled. ECLIPSE associates the first half of the grid (the first NDIVIZ/2 layers) with the matrix blocks, and the second half with the fractures. In such runs NDIVIZ must therefore be even; ECLIPSE checks that this is the case.

Transmissibility calculations

The matrix-fracture coupling transmissibility terms that exist between each cell of the matrix grid and the corresponding cell in the fracture grid are proportional to the cell bulk volume, being of the form:

$$\mathrm{TR} = \mathrm{CDARCY} \cdot K \cdot V \cdot \sigma \qquad \text{[EQ 16.1]}$$

where by default:

K is taken as the X-direction permeability of the matrix blocks (unless PERMMF is also specified; see below),

V is the matrix cell bulk volume (note that this is not the pore volume, having no porosity factor), and

σ is a factor of dimensionality LENGTH^{-2}, to account for the matrix/fracture interface area per unit volume, that is the size of the blocks in the matrix volume.

The default use of the X-direction permeability can be overridden by using the matrix-fracture permeability specified by the PERMMF keyword. Note that for one-dimensional (Y- or Z-direction) cases in the global grid with X-direction extension in an LGR, the X-direction permeability will be set to zero in all grids and PERMMF *must* be used to enable dual porosity NNCs to be generated. (This is unlikely in real simulations, but many test problems are affected by this.)

Kazemi [Ref. 5] has proposed the following form for σ:

$$\sigma = 4\left(\frac{1}{l_x^2} + \frac{1}{l_y^2} + \frac{1}{l_z^2}\right) \qquad \text{[EQ 16.2]}$$

where l_x, l_y and l_z are typical X, Y and Z dimensions of the blocks of material making up the matrix volume. (l_x, l_y and l_z are thus not related to the simulation grid dimensions). Alternatively, as σ acts as a multiplier on the matrix-fracture coupling, it may simply be treated as a history matching parameter.

σ can be specified as a single value for the whole field or on a cell by cell basis using the keyword SIGMAV. If σ is defined on a cell by cell basis then the values corresponding to the first NDIVIZ/2 layers are used. If σ is printed out (RPTGRID keyword) then the values in the first NDIVIZ/2 layers are copied in to the lower NDIVIZ/2 layers.

The intrinsic permeabilities of cells in the fracture system are equal to the specified values multiplied by the fracture porosities to yield effective fracture permeabilities. If the NODPPM keyword is used then the intrinsic permeability is used rather than the effective permeability.

That is, when NODPPM is **not** specified, then within the fracture cells, the input PERMX, PERMY and PERMZ are modified using $PERMX(fr) = PERMX(fr) \times PORO(fr)$.

ECLIPSE 100

An alternative form of the above transmissibility is available using the LTOSIGMA keyword if the matrix block dimensions are input explicitly using the LX, LY and LZ keywords. In this case the transmissibility is given by:

$$\mathrm{TR} = \mathrm{CDARCY} \cdot V \cdot \left(\frac{f_x \cdot K_x}{l_x^2} + \frac{f_y \cdot K_y}{l_y^2} + \frac{f_z \cdot K_z}{l_z^2}\right) \qquad \text{[EQ 16.3]}$$

where:

$f_x\, f_y\, f_z$ are input in `LTOSIGMA`

$K_x\, K_y\, K_z$ are the matrix permeability values

$l_x\, l_y\, l_z$ are the matrix block dimensions input using `LX`, `LY` and `LZ`.

If the dual porosity but not the Dual Permeability option is selected, the matrix blocks have no mutual transmissibilities. If dual porosity and dual permeability is chosen, the matrix blocks have their normal transmissibilities.

Figure 16.1 A simple, dual porosity, dual permeability system.

Block to block transmissibility calculations

It is now also possible to represent re-imbibition from a fracture cell by the matrix cell below it. In this case the transmissibility between the matrix and fracture cells is a conventional spatial one, since the two cells are not superimposed on one another, and the properties of the two cells contribute to the transmissibility in exactly the same way as for the transmissibility between two neighboring fracture cells, or two neighboring matrix cells. See"Transmissibility calculations" on page 1104 for the details of this calculation. This modelling is most appropriate when the physical matrix block size (l_z as used in the Kazemi formula for the usual matrix-fracture transmissibility multiplier represented by the `SIGMA` and `SIGMAV` keywords) is comparable to the size of the grid cell rather than significantly different from it, so that the environment of a lower matrix physical block also comprises neighboring fracture cells rather than just its own fracture cell.

Figure 16.2 Physical view of the matrix and fracture cells for the block to block connection.

Figure 16.2 shows this new transmissibility on a picture of the two physical cells (i.e. the matrix and fracture cells occupy the same physical location.)This figure appears to show the lower matrix disconnected from the upper fracture by its own surrounding fracture, but this is only for presentation purposes: there will always be some contact between the upper fracture and lower matrix. This is schematically represented in Figure 16.3, where the lower matrix and fracture are shown as single adjacent blocks, but the upper matrix is split in two, one half abutting the lower matrix (a disconnection in dual porosity but a connection in dual permeability) and one half abutting the lower fracture (not represented in the model, but conceptually possible). The upper fracture contacts both lower fracture and lower matrix.

Figure 16.3 Schematic view of the contacts between the upper and lower cells of the two porosity systems.

The extent of the contact between the upper fracture and lower matrix is a variable of the model, and hence a new multiplier is applied to the transmissibility between these cells, to represent this effective contact area between the matrix and the fracture, which will be different from the contact area between the two vertically neighboring fracture cells, or two vertically neighboring matrix cells. This contact area multiplier is specified by the keywords `BTOBALFA` or `BTOBALFV`; the former specifies a single value for the whole grid, while the latter specifies it cell by cell. The presence of these keywords also activates these additional transmissibilities, which will otherwise not be present.

ECLIPSE 100

In ECLIPSE 100, block-to-block connections between LGRs and their parents or adjacent LGRS are only supported for in-place LGRs.

Note Block to block connections cannot be made when a multi porosity model is specified (`NMATRIX` or `TRPLPORO`).

Recovery mechanisms

In a dual porosity system the majority of the oil is contained in the matrix system, but the production of oil to the wells is through the high permeability fracture system. In such a system an injected fluid does not sweep out oil from the matrix block. Production from the matrix blocks can be associated with various physical mechanisms including:

- Oil expansion
- Imbibition
- Gravity imbibition/drainage
- Diffusion
- Viscous displacement.

Oil expansion

As the pressure drops in the fracture system, oil flows from the matrix to equilibrate the matrix pressure with the surrounding fracture pressure. This production mechanism can be thought of as expansion of the oil within the matrix block, either above the bubble point or by solution gas drive below the bubble point.

Imbibition

In a typical water wet system the matrix rock has a positive water-oil capillary pressure. If water is introduced into the fracture, the water flows under capillary forces into the matrix system, displacing oil. Note that if the gravity drainage model (see below) is not active, then the production continues until the oil saturation in the matrix has reached residual oil.

The water imbibition process is modeled in ECLIPSE by specifying different saturation table numbers for the matrix and fracture cells respectively. The matrix cells typically have a water-oil capillary pressure, while the fracture cells usually have zero capillary pressure.

In gas-oil systems the oil is the wetting phase and tends to imbibe into the matrix. In practice this means that if the gravity drainage model is not active then no oil production occurs from a matrix block when the associated fracture block full of gas.

Gravity imbibition/drainage

Fluid exchange between the fracture and matrix due to gravity is modeled in one of two ways. One method is to consider each of the matrix and fracture cells as separately in vertical equilibrium, and then to calculate additional potential due to differences in contact heights between the matrix and fracture. There are two options for this, which are activated by including either of the keywords `GRAVDR` or `GRAVDRM` (described in the next two sections) in the `RUNSPEC` section. This method leaves the number of porosities in the grid as 2.

The other method (described in the section following) uses a modification of the Discrete Matrix model (E100) or multi porosity model (E300), and uses N matrix porosities, where N is user-defined, to create a vertical stack of finely spaced matrix cells which describe the distribution of properties within a single block of matrix material (with height as defined by the DZMTRX keyword). This option is not available with dual permeability runs, which should use a more detailed discretization of the whole grid; the advantage of discretizing only the matrix is that its equations can be eliminated from the solution before the linear solution is performed, so that no additional time is spent in that part of the code.

The DZMTRX keyword is also required for the GRAVDR or GRAVDRM models, and a non-zero value should be given for it in the GRID section in all gravity drainage cases.

Standard Gravity Drainage model (GRAVDR)

The illustration shows a typical block of matrix material containing oil and water. The fractional height of the water table in the fracture is X_W and the fractional height water displacement front in the block of matrix material is X_w.

Figure 16.4 A typical block of matrix material containing oil and water

The pressure difference due to gravity alone is

$$\Delta P = DZ_{mat}(X_W - X_w)(\rho_w - \rho_o)g \qquad \text{[EQ 16.4]}$$

where

ρ_w is the density of water and

ρ_o is the density of oil at reservoir conditions.

DZ_{mat} is the same as l_z in Kazemi's formula.

Integrating ΔP over all blocks of matrix material within a reservoir simulation cell leads to the same formula for ΔP, except that it now represents the gravitational pressure difference between reservoir simulation fracture and matrix cells, and X_W and X_w are the partial volumes of mobile water in the simulation fracture and matrix cells. In ECLIPSE, the effect is modeled by introducing additional pseudo capillary pressures, P_S

$$P_{Scow} = \mathrm{DZ}_{mat} \cdot X_W(\rho_o - \rho_w)g \text{ (fracture cells)}$$

$$P_{Scow} = \mathrm{DZ}_{mat} \cdot X_w(\rho_o - \rho_w)g \text{ (matrix cells)}$$

$$P_{Scog} = \mathrm{DZ}_{mat} \cdot X_G(\rho_o - \rho_g)g \text{ (fracture cells)}$$

$$P_{Scog} = \mathrm{DZ}_{mat} \cdot X_g(\rho_o - \rho_g)g \text{ (matrix cells)}$$

$$P_{Scwg} = \mathrm{DZ}_{mat} \cdot X_G(\rho_w - \rho_g)g \text{ (fracture cells)}$$

$$P_{Scwg} = \mathrm{DZ}_{mat} \cdot X_g(\rho_w - \rho_g)g \text{ (matrix cells)}$$

Thus, the flow of oil and gas from a fracture to a matrix cell in a gas oil system is computed as

$$F_g = \mathrm{TR} \cdot \mathrm{GMOB} \times \left(P_{of} - P_{om} + d_{fm}\rho_g g + P_{cogf} - P_{cogm} + \frac{\mathrm{DZ}_{mat}(X_G - X_g)(\rho_o - \rho_g)g}{2}\right) \quad \text{[EQ 16.5]}$$

$$F_o = \mathrm{TR} \cdot \mathrm{OMOB}\left(P_{of} - P_{om} + d_{fm}\rho_o g - \frac{\mathrm{DZ}_{mat}(X_G - X_g)(\rho_o - \rho_g)g}{2}\right) \quad \text{[EQ 16.6]}$$

where

TR is the transmissibility between the fracture and matrix cells,

GMOB is the gas mobility in the (upstream) fracture cell,

OMOB is the oil mobility in the (upstream) fracture cell,

P_{of} is the oil phase pressure in the fracture cell,

P_{om} is the oil phase pressure in the matrix cell,

d_{fm} is the difference in depth between the fracture and matrix cells (usually zero),

ρ_g is the density of gas at reservoir conditions,

ρ_o is the density of oil at reservoir conditions,

g is the acceleration due to gravity,

P_{cogf} is the capillary pressure of gas in the fracture cell (normally zero)

P_{cogm} is the capillary pressure of gas in the matrix cell.

Note Note that the gravity drainage head $\mathrm{DZ}_{mat}(X_G - X_g)(\rho_o - \rho_g)g$ has been split between the two phases.

In dual porosity / dual permeability models in which the gravity imbibition / drainage mechanism is active, the calculation of the initial reservoir state (see "Initializing the Study" on page 543) occurs independently of the value of DZ_{mat} that is the reservoir is initialized without regard for the gravity imbibition / drainage forces acting between the matrix and fracture cells. When the simulation begins, the gravity imbibition / drainage forces may cause fluid movement between matrix cells and their corresponding fracture cells due to the different fluid levels in

the matrix / fracture cells. If the redistribution of fluids causes a significant transient when the simulation is started, you can overcome this by activating the 11th argument of the OPTIONS keyword. If this switch is activated, the effect is to cause modifications to the phase pressures to make the initial solution a true steady state. These phase pressure modifications are applied for the duration of the run.

ECLIPSE 300

A different value of the σ coupling may be specified for gravity drainage by using the SIGMAGD keyword. If the values entered with SIGMA and SIGMAGD, [denoted σ and σ_{gd} respectively] differ, the program uses an interpolated σ value for each phase which depends on the relative strengths of the capillary pressure and gravity drainage terms:

$$\sigma_p^{\text{eff}} = f_p \cdot \sigma + (1 - f_p) \cdot \sigma_{gd} \qquad \text{[EQ 16.7]}$$

where p =oil, gas, water, and

$$f_p = \frac{|\Delta P_p|}{|\Delta P_p| + |\Delta P_{pgd}|} \qquad \text{[EQ 16.8]}$$

ΔP_{pij} is the usual potential difference term between two cells i and j:

$$\Delta P_{pij} = P_{pi} - P_{pj} + P_{pci} - P_{pcj} - G\bar{\rho}_p \Delta d \qquad \text{[EQ 16.9]}$$

where (P_{pi}, P_{pj}) are the cell pressures, (P_{pci}, P_{pcj}) are the cell capillary pressures, Δd is the height difference between the cells, G is the gravitational constant and $\bar{\rho}_p$ is the saturation weighted average density of phase p.

ΔP_{gd} is the gravity drainage term, which depends on the phase:

$$\Delta P_o = -\frac{1}{2}(\Delta P_{gd}^{og} + \Delta P_{gd}^{ow}) \qquad \text{[EQ 16.10]}$$

$$\Delta P_g = \frac{1}{2}\Delta P_{gd}^{og} \qquad \text{[EQ 16.11]}$$

$$\Delta P_w = \frac{1}{2}\Delta P_{gd}^{ow} \qquad \text{[EQ 16.12]}$$

$$\Delta P_{gd}^{og} = G \times DZ_{\text{mat}}(\bar{\rho}_o - \bar{\rho}_g)(X_{ig} - X_{jg}) \qquad \text{[EQ 16.13]}$$

$$\Delta P_{gd}^{ow} = G \times DZ_{\text{mat}}(\bar{\rho}_w - \bar{\rho}_o)(X_{iw} - X_{jw}) \qquad \text{[EQ 16.14]}$$

where X_g and X_w are the fractions of the cell containing mobile gas and water respectively. These are estimated from the cell saturation using a vertical equilibrium model. If the gravity drainage term is dominant, $f_p \to 0$ and $\sigma_{\text{eff}} \to \sigma_{gd}$.

Alternative Gravity Drainage model (GRAVDRM)

An alternative gravity drainage model is available based on the formulation proposed by Quandalle and Sabathier in [Ref. 8].

The matrix-fracture flow is taken to be the sum of three flows from the center of the matrix to the fracture system: one horizontal, one vertically upwards and one vertically downwards. This formulation allows for different flow paths and hence transmissibilities in the vertical and horizontal directions. In some circumstances, typically in mixed wetability systems, the two transmissibilities give a better match to a finely gridded matrix block model.

The matrix fracture flow is given by:

$$F = F_h + F_{up} + F_{down} \qquad \text{[EQ 16.15]}$$

Where the horizontal flow is:

$$F_h = TR_{sigma} \cdot MOB_h \cdot (P_f + P_{cf} - P_m - P_{cm}) \qquad \text{[EQ 16.16]}$$

TR_{sigma} is the transmissibility based on the SIGMA keyword

MOB_h is the phase (upstream) mobility

P_f , P_m are the oil phase pressures in the fracture and matrix cells

P_{cf} , P_{cm} are the capillary pressures.

The vertical flows are:

$$F_v = TR_{sigmagd} \cdot MOB_v \cdot (P_f + P_{cf} - P_m - P_{cm} + (\rho_m - \rho_f^{star}) \cdot g \cdot (DZ_{\text{mat}})/2) \qquad \text{[EQ 16.17]}$$

$TR_{sigmagd}$ is the transmissibility based on the SIGMAGD keyword

MOB_v is the phase (upstream) mobility based on the potential difference including the gravity term

DZ_{mat} is the matrix block height given in the DZMATRIX keyword

g is the acceleration due to gravity

ρ_m is the phase density in the matrix

ρ_f^{star} is given by $\rho_f^{star} = S_w \cdot \rho_w + S_o \cdot \rho_o + S_g \cdot \rho_g$

One drawback with this formulation is that the final recovery from a block can become a function of the transmissibility in the case when the final recovery is given by $F = 0$ when the vertical flows balance the horizontal $F_h = -F_{up} - F_{down}$. This effect can be avoided (in the case of oil) by invoking an option to prevent the re-infiltration of oil; see the GRAVDRM keyword in the RUNSPEC section.

ECLIPSE 100

Note that the SIGMA and SIGMAGD values can be calculated from the block dimensions by using the LTOSIGMA keyword in conjunction with the LX, LY and LZ keywords.

Vertical Discrete Matrix Gravity Drainage Model

The third alternative gravity drainage model uses the Discrete Matrix model (ECLIPSE 100) or Multi porosity model (ECLIPSE 300), both of which provide a number of matrix porosities connected together in a chain, with the outermost connected to the fracture. If the NMATOPTS keyword is set to the value VERTICAL, then this chain of matrix porosities will be interpreted as a vertical stack of matrix cells connected within a single block of matrix material of height DZMATRIX. This value of the keyword will also connect all submatrix porosities to the fracture. The number of porosities used will still be defined by the NMATRIX keyword.

When these options are activated, the physical picture of the fracture cell and its associated matrix cells is as shown in Figure 16.5. Each matrix cell sees the same fracture cell conditions. To prevent this limitation reducing the accuracy of the solution unacceptably, the properties of the fracture cell are altered for each NNC describing the matrix-fracture links to agree better with the property variations with height which would be experienced if the fracture were similarly discretized. This alteration takes into account the change with height of gravitational potential and any phase discontinuities (which are described in the same way as for the GRAVDR model).

Figure 16.5 The discretized matrix material block within a fracture cell.

Note The connections to the fracture are shown explicitly; the discrete matrix model connections are shown as a single arrow.

ECLIPSE 300

In ECLIPSE 300, the input requirements for multi porosity give greater flexibility in the variation of properties at different heights of the matrix, but it is recommended that all the porosities are given the same property values.

Diffusion

Molecular diffusion of gas and oil between the matrix and fracture may be a significant production mechanism from the matrix. The ECLIPSE Diffusion option is described in "Diffusion" on page 211.

Viscous displacement (ECLIPSE 100)

Viscous displacement of a fluid is simply the movement of that fluid when a pressure differential is applied. In a dual porosity system there is a pressure gradient in the fracture system moving the fluid through the fracture, towards the production wells. In many cases this pressure gradient is small as the fracture system has a very high effective permeability. In these cases it is reasonable to ignore the viscous displacement of fluids from the matrix by the fracture pressure gradient. However, if the fracture system has a more moderate permeability then flow to and from the matrix caused by the fracture pressure gradient may be expected to act as a significant production mechanism.

The viscous displacement option that models this effect is activated using the VISCD keyword in the RUNSPEC section. The representative matrix block sizes are input using the LX, LY and LZ keywords in the GRID section.

Provided that the representative matrix block sizes have been input using the LX, LY and LZ keywords, it is possible to calculate the SIGMA factor from these matrix block dimensions using the LTOSIGMA keyword. Note that if SIGMA is calculated in this way then any input SIGMA data will be ignored.

The ECLIPSE viscous displacement model is based on a technique described by J. R. Gilman and H. Kazemi (see [Ref. 9]).

Figure 16.6 A representative matrix block in a fracture potential gradient

Consider a representative matrix block in a fracture potential gradient. The average fracture potential is given by P_f, with an effective pressure drop across the block of $P_1 - P_2$.

If the matrix pressure is P_{m1}, which is greater than both P_1 and P_2 then clearly the flow will be from matrix to fracture,

$$F = \frac{T}{2} \cdot M_m(P_{m1} - P_1) + \frac{T}{2} \cdot M_m(P_{m1} - P_2) \qquad \text{[EQ 16.18]}$$

where:

T is the matrix-fracture transmissibility

M_m is the matrix mobility.

Hence

$$F = T \cdot M_m(P_{m1} - P_f) \qquad \text{[EQ 16.19]}$$

that is the usual matrix-fracture flow since $P_f = (P_1 + P_2)/2$.

If the matrix pressure is lower than both P_1 and P_2 (e.g. P_{m3}) then the flow is from fracture to matrix, and as above can be expressed as

$$F = T \cdot M_f(P_f - P_{m3}) \qquad \text{[EQ 16.20]}$$

where:

M_f is the fracture mobility.

But in the case where the matrix pressure is P_{m2}, where $P_1 > P_{m2} > P_2$, flow will occur from fracture to matrix at one end and from matrix to fracture at the other. In this case the flow is given by

$$F = \frac{T}{2} \cdot M_f \cdot (P_1 - P_{m2}) - \frac{T}{2} \cdot M_m \cdot (P_{m2} - P_2) \qquad \text{[EQ 16.21]}$$

This can be written as

$$F = \frac{T}{2} \cdot M_f \cdot \left(P_f + \frac{(P_1 - P_2)}{2} - P_{m2}\right) - \frac{T}{2} \cdot M_m \cdot \left(P_{m2} - P_f + \frac{(P_1 - P_2)}{2}\right) \qquad \text{[EQ 16.22]}$$

P1 and P2 are not known directly in the simulator, but an estimate of the fracture potential gradient can be made from the adjacent cell pressures. Now

$$P_1 - P_2 = G \cdot L$$

where:

G is the fracture pressure gradient

L is the representative dimension of the matrix block in the appropriate direction, input using the LX, LY and LZ keywords.

Equation [EQ 16.22] can be written as a standard matrix-fracture flow with modified up-winding and an extra term associated with the viscous displacement

$$F = \frac{T}{2} \cdot M_f \cdot (P_f - P_{m2}) - \frac{T}{2} \cdot M_m \cdot (P_{m2} - P_f) + \frac{T}{4} \cdot G \cdot L \cdot (M_f - M_m) \qquad \text{[EQ 16.23]}$$

The matrix-fracture flows are modified in a similar manner for all three directions.

The fracture potential gradient, G, is estimated from the potential differences to the surrounding cells.

Figure 16.7 Estimating the potential differences to the surrounding cells

$$G = \frac{|P_l - P|}{2D_l} + \frac{|P - P_r|}{2D_r} \qquad \text{[EQ 16.24]}$$

where:

$|P_l - P|$ and $|P - P_r|$ are the absolute values of the potential difference to adjacent cells

D_l and D_r are the distance between the cell centers.

In this version of ECLIPSE the fracture potential gradients are calculated from the potential differences at the previous time step.

Facilities specific to dual porosity runs

Fracture permeabilities

It is necessary to be clear about what the input fracture permeability represents. By default ECLIPSE multiplies the input fracture permeabilities by the fracture porosity to generate an effective permeability. If the input fracture permeabilities themselves represent the effective value, then the NODPPM keyword should be used in the GRID section.

Matrix-fracture relative permeabilities

Usually two sets of saturation functions (relative permeabilities and capillary pressures) are supplied, one for the matrix cells and one for the fracture cells. The SATNUM keyword in the REGIONS section is used to associate the tables with the appropriate grid blocks. As ECLIPSE 'upwinds' the flows, a phase flow from matrix to fracture takes the relative permeability calculated from the phase saturation in the matrix using the matrix table. Flow from the fracture to the matrix uses the fracture saturation and table.

ECLIPSE 100

If a separate relative permeability table is required for the matrix/fracture flow, this can be specified using the KRNUMMF keyword in the REGIONS section. If the Hysteresis option is active, then a corresponding keyword IMBNUMMF can be used to specify the imbibition relative permeability table.

When flow occurs from fracture to matrix it may be more reasonable to regard the maximum relative permeability as the matrix relative permeability at the residual saturation of the displaced phase. This effect can be easily achieved without specifying separate matrix-fracture tables (KRNUMMF), by scaling the fracture relative permeability. This scaling can be activated by using the third data item in the DPKRMOD keyword.

Simplified grid input

The keyword DPGRID may be used to simplify the construction of grids for dual porosity runs. This keyword enables grid data to be entered for the matrix cells only (the first NDIVIZ/2 layers), the missing values for the remaining fracture layers being obtained from the corresponding matrix cell.

Use of FLUXNUM/MULTNUM or OPERNUM

The use of these keywords differs between ECLIPSE 100 and ECLIPSE 300.

ECLIPSE 100

Only the matrix cells need to be input. ECLIPSE 100 will copy these values to the fracture cells automatically. If both matrix and fracture cells are given, a warning message would be output complaining about too much data.

ECLIPSE 300

Both matrix and fracture cells should be provided. FLUXNUM and MULTNUM use with MULTREGT or MULTREGP, will automatically copy all matrix cells to the corresponding fracture cells, so dummy fracture values could be input. For OPERNUM use, and other FLUXNUM and MULTNUM uses, there is no copying of data from the matrix to fracture cells.

Partially fractured reservoirs

In a dual-porosity single-permeability run, the simulation model may be specified to be dual porosity only in part of the reservoir, using the keyword DPNUM. ECLIPSE treats the rest of the reservoir as in a conventional single porosity run. In this case, the single porosity cells are taken to be matrix cells for input purposes, and any data input for fracture cells in the single porosity region is ignored.

Diffusion

The Molecular Diffusion option can be limited to calculate only the matrix/fracture diffusive terms, by using the DIFFDP keyword. Here we are making the assumption that diffusion in the fracture system is negligible compared to the fracture flows. The matrix/fracture diffusivity can be modified using the DIFFMMF keyword in the GRID and SCHEDULE sections.

Modifying the recovery versus time response

ECLIPSE 100

The shape of the recovery versus time response can be modified by changing the matrix relative permeability curves. ECLIPSE provides a simplified method of changing the input relative permeabilities using only a single parameter. Typically this is useful when matching a dual porosity cell to a finely gridded single porosity model. The facility is described in more detail in a subsequent section and is activated with the DPKRMOD keyword.

Integrated capillary pressure option

This option automatically generates a modified capillary pressure function with the aim of improving the prediction of final recovery from a matrix cell. The option is described in detail in a subsequent section and is activated with the INTPC keyword.

Calculating SIGMA from the matrix dimensions

ECLIPSE 100

If the Viscous Displacement option has been used, the representative matrix block sizes will have to be entered using the LX, LY and LZ keywords. It is possible to calculate the SIGMA value from these block dimensions by specifying the LTOSIGMA keyword. Note that if this keyword is used then any input SIGMA values are ignored.

Restrictions on dual porosity runs

The following restrictions apply to dual porosity (DUALPORO) runs (except in single porosity regions specified using DPNUM), but **not** to dual porosity dual permeability (DUALPERM) runs.

- Wells connect only to fracture cells - not to matrix cells.
- Non-neighbor connections (keyword NNC) may not be used with matrix cells. The internal connection of each matrix cell to its appropriate fracture cell is made automatically by ECLIPSE.
- Each active matrix cell must connect with an active fracture cell.

In single porosity regions, there are no active fracture cells. Within these regions, data is only required for the matrix cells; any data for the fracture cells are ignored. Wells should connect to matrix blocks within single porosity regions.

Keyword summary

Table 16.1 shows a summary of dual porosity keywords.

Table 16.1 Dual porosity keywords

Keyword	Status	Brief Description of Data	Availability
BTOBALFA	GRID keyword	Initiates block to block connections and specifies a single contact area multiplier.	E300 only
BTOBALFV	GRID keyword	Initiates block to block connections and specifies contact area multipliers.	E300 only
DUALPORO	Switch in RUNSPEC	Initiates the dual porosity model.	Both
DUALPERM	Switch in RUNSPEC	Initiates the dual permeability model.	Both
GRAVDR	Switch in RUNSPEC	Requests the gravity drainage model.	Both
GRAVDRM	Switch in RUNSPEC	Requests alternative gravity drainage model	Both
NMATRIX	RUNSPEC keyword	Requests the discretized matrix model.	E100 only
SIGMA	SIGMA or SIGMAV required	Matrix-to-fracture coupling factor for the entire grid.	Both
SIGMAV	SIGMA or SIGMAV required	Matrix-to-fracture coupling factor for the current box.	Both
MULTSIG	SIGMA or SIGMAV required	Matrix-to-fracture coupling multiplier for the entire grid.	Both
MULTSIGV	SIGMA or SIGMAV required	Matrix-to-fracture coupling multiplier for the current box.	Both
DZMTRX	DZMTRX or DZMTRXV required if gravity drainage is active	Specifies the typical block height for gravity drainage for the entire grid.	Both
DZMTRXV	DZMTRX or DZMTRXV required if gravity drainage is active	Specifies the typical block height for gravity drainage for the current box.	Both
LX	Optional when the viscous displacement option is active	Specifies the representative X dimension of the matrix block.	E100 only
LY	Optional when the viscous displacement option is active	Specifies the representative Y dimension of the matrix block.	E100 only
LZ	Optional when the viscous displacement option is active	Specifies the representative Z dimension of the matrix block.	E100 only
DIFFDP	Optional if diffusion option is active	Restricts the molecular diffusion calculation to just the matrix-fracture flows.	E100 only
DPGRID	Optional	Enables user to enter grid data for the matrix cells only.	Both

Table 16.1 Dual porosity keywords (Continued)

Keyword	Status	Brief Description of Data	Availability
DPNUM	Optional	Identifies the extent of the regions in which the single porosity model may be applied in a dual porosity run.	E100 only
IMBNUMMF	Optional if the hysteresis option is active	Imbibition table region for matrix-fracture flows.	E100 only
KRNUMMF	Optional	K_r region for matrix-fracture flows.	E100 only
NODPPM	Optional	Specifies that permeabilities in the fracture cells are not to be multiplied by the fracture porosity.	Both
SATNUM	Recommended	To specify different saturation region to the matrix and fracture cells.	Both
SIGMAGD	Optional	Matrix-to-fracture coupling for oil-gas gravity drainage, for the entire grid.	Both
SIGMAGDV	Optional	Matrix-to-fracture coupling for oil-gas gravity drainage, for the current box.	Both
MULSGGD	Optional	Matrix-to-fracture multiplier for oil-gas gravity drainage, for the entire grid.	Both
MULSGGDV	Optional	Matrix-to-fracture multiplier for oil-gas gravity drainage, for the current box.	Both
INTPC	Optional	Invokes the integrated capillary pressure option	Both
DPKRMOD	Optional	Modifies oil K_r curve for matching fine grid single porosity recovery vs. time, and allows scaling of the fracture relative permeability.	E100 only
LTOSIGMA	Optional if the viscous displacement option is active	Requests that the SIGMA value is to be calculated from the LX, LY and LZ data.	E100 only
NMATOPTS	Optional if the discretized matrix option is active	Specifies the geometry type of the matrix and the size of the outer matrix cell.	E100 only
PERMMF	Optional	Permeability for Matrix-Fracture Coupling.	E300 only
MULTMF	Optional	Multiplier for Matrix-Fracture Permeability.	E300 only
DIFFMMF	Optional	Multiplier for Matrix-Fracture Diffusivities.	Both

Transfer functions

Transfer functions are curves representing the oil expelled from a matrix element as a function of time. To simulate a gas/oil transfer function, set up a reservoir grid as shown below. Make the bottom matrix cell inactive using keyword ACTNUM. Fill the fracture cells, F1 and F2, with gas and the matrix cell, M1, with oil and connate water using the appropriate keywords in the SOLUTION section. Include keyword GRAVDR in the RUNSPEC section. Record the oil saturation in cell F2 using BOSAT in the SUMMARY section. Run the simulation for a few years. Display the transfer function using GRAF. Adjust DZMTRX and/or P_{cogm} to match the observed data or match to a finely gridded single porosity model.

Figure 16.8 A simple reservoir grid used to simulate a gas/oil transfer function

Modifying the recovery versus time response (ECLIPSE 100)

The eventual recovery of oil from a matrix block is governed by the final balance between the gravity head and the capillary pressure. The initial rate of drainage is governed by the transmissibility between the matrix and fracture, which in turn is controlled by the shape factor SIGMA. The 'shape' of the recovery curve between the end points can be modified by changing the relative permeability function for the matrix-fracture flow.

This can be done by modifying the relative permeability curves directly. However, when matching a finely gridded single porosity model or experimental results it is often simpler to modify a single parameter to change the 'shape' of the recovery versus time response. The DPKRMOD keyword applies a quadratic modification to the oil relative permeability to achieve this aim.

The modification factor is not a physical quantity and should be regarded only as a tuning parameter to match a dual porosity block with a finely gridded single porosity model.

The scaling for oil-in-water relative permeability is as follows for the case where m_w (the modification factor set in DPKRMOD) is greater than 0.0:

$$K_r = \overline{K_r} + M(K_r(s) - \overline{K_r}) \qquad \text{[EQ 16.25]}$$

where

$\overline{K_r}$ is the relative permeability at $\bar{s} = (\mathrm{SOWCR} + 1.0 - \mathrm{SWCO})/2$

$K_r(s)$ is the input table relative permeability

SOWCR is the critical oil to water saturation

SWCO is the connate water saturation

M is the modification function, which is given by $M = 4.0 m_w K_r(s)^2 - 4.0 m_w K_r(s) + 1$

If m_w is less than 0.0, then the same scaling is applied but with the X and Y axes $(s, K_r(s))$ reversed:

For a given saturation s, a new saturation s' is calculated such that:

$$\bar{s} + M(s' - \bar{s}) = s \qquad \text{[EQ 16.26]}$$

where

$\bar{s}$ is $(\mathrm{SOWCR} + 1.0 - \mathrm{SWCO})/2$

M is the modification function, which is given by $M = 4.0 m_w K_r(s')^2 - 4.0 m_w K_r(s') + 1$

K_r is then taken as $K_r(s')$.

Scaling the fracture relative permeability (ECLIPSE 100)

By default, both the fracture-to-fracture flow and the fracture-to-matrix flow uses the fracture relative permeability table. Typically this will have a maximum relative permeability to water of 1.0, whereas the water relative permeability in the matrix at residual oil saturation is likely to be lower than 1.0. In some cases the matrix relative permeability may provide a better description of the fracture-to-matrix flow.

The fracture scaling option in the `DPKRMOD` keyword modifies the fracture relative permeability for the fracture-to-matrix flow while leaving the fracture-to-fracture relative permeability alone.

For the water phase the fracture relative permeability is scaled as follows:

$$K_{rf}(S_w) = K_{rf}^{\text{tab}}(S_w) \cdot K_{rm\,\max} / K_{rf\,\max} \qquad \text{[EQ 16.27]}$$

where

$K_{rf}^{\text{tab}}(S_w)$ is the fracture relative permeability looked up using the input table

$K_{rm\,\max}$ is the matrix relative permeability at $S_w = 1.0 - S_{owcr}$

$K_{rf\,\max}$ is the maximum fracture relative permeability

S_{owcr} is the critical oil saturation to water.

The other phases are treated in an analogous manner. In the gas case the relative permeability is scaled to the matrix value at $S_g = 1.0 - S_{ogcr}$ and in the oil case at $S_o = 1.0 - \max(S_{crg}, S_{crw})$.

Solution of the linear equations

The linear equations associated with dual porosity/permeability simulations can be written in the form

$$Ax + Uy = R_m$$

$$Lx + By = R_f$$

where

A is the usual banded matrix for the matrix cells

B is the banded matrix for the fracture cells

Rm is the matrix residual

R_f is the fracture residual

L and U are diagonal matrices coupling the fracture and matrix systems

x and y are the solutions (pressure and saturation changes) in the matrix and fracture cells respectively.

In the dual porosity single permeability case, there is no direct communication between neighboring matrix cells and the matrix A is diagonal. This greatly enhances the efficiency of the solution procedure used by ECLIPSE because the equations can now be simplified to the form

$$(B - L \cdot A^{-1} U)y = R_f - L \cdot A^{-1} R_m \qquad \text{[EQ 16.28]}$$

$$x = A^{-1}(R_m - Uy) \qquad \text{[EQ 16.29]}$$

where A^{-1} is the inverse of A.

Because $L \cdot A^{-1} U$ is diagonal it does not alter the banded structure of B. The problem size is effectively halved because the equation for y only involves fracture cells. The equations are solved sequentially, first for the fracture solution, y, and then for the matrix solution, x. In the more expensive dual porosity/dual permeability case above simplification does not apply and the equations are solved simultaneously.

Data for single porosity regions in a dual porosity run is copied internally to the fracture cells, so that the above simplification still applies.

ECLIPSE 100

In dual permeability simulations, the linear equations can be solved more efficiently by pairing together the corresponding matrix and fracture cells. For a 3-phase run, the diagonal entries of the Jacobian matrix are then 6×6 matrices of the form.

$$D = \begin{bmatrix} D_m & U \\ L & D_f \end{bmatrix} \qquad \text{[EQ 16.30]}$$

where D_m and D_f are the diagonal terms of the matrix and fracture Jacobian matrices. To account for the extra dimension, a fourth nesting has been used in the Nested Factorization preconditioner (see "Solution of the Linear Equations" on page 937). Using the notation of that chapter, an approximation, B, is constructed to the full Jacobian matrix

$$A = D + L1 + U1 + L2 + U2 + L3 + U3 \ , \qquad \text{[EQ 16.31]}$$

where the band terms $L1$ and $U1$ etc. represent flows between matrix or fracture cells. The following nested sequence of factorizations is used

$$
\begin{aligned}
B &= (P+L3)P^{-1}(P+U3) \\
P &= (T+L2)T^{-1}(T+U2) \\
T &= (G+L1)G^{-1}(G+U1) \\
G &= (\Gamma+L)\Gamma^{-1}(\Gamma+U)
\end{aligned}
\qquad \text{[EQ 16.32]}
$$

where Γ is a diagonal matrix (whose elements are all 3×3 matrices in the 3-phase case), and G has the same structure as D.

ECLIPSE 100

If the 60th switch of the `OPTIONS` keyword is set greater than zero, the less efficient dual permeability solver for pre-97A versions of ECLIPSE is restored. If you encounter problems with your simulation run you may also wish to use this option.

Calculation of fractional volumes for gravity drainage

The initial fractional volume of water below the water contact, X_{wi}, is

$$X_{wi} = \frac{S_{wi} - S_{wco}}{1 - S_{cohy} - S_{wco}} \qquad \text{[EQ 16.33]}$$

Figure 16.9 Initial fractional volume of water below the water contact

where

S_{wi} is the initial water saturation

S_{wcr} is the critical water saturation

S_{wco} is the connate water saturation

S_{cohy} is the connate hydrocarbon saturation

$= S_{ocow}$ (oil/water systems)

$= S_{ocow} + S_{gco}$ (oil/water/gas systems)

$= S_{gco}$ (gas/water systems)

S_{crhy} is the critical hydrocarbon saturation.

If $S_w > S_{wi}$ then

$X_w = 1$ when

$S_w = S_{w\,\max} = X_{wi}(1 - S_{cohy}) + (1 - X_{wi})(1 - S_{crhy})$

$X_w = (S_w - X_{wi}(S_{crhy} - S_{cohy}) - S_{wco}) / (1 - S_{crhy} - S_{wco})$ when $S_w < S_{w\,\max}$

If $S_w < S_{wi}$ then

$X_w = 0$ when $S_w = S_{w\,\min} = X_{wi}S_{wcr} + (1 - X_{wi})S_{wco}$

$X_w = (S_w - X_{wi}(S_{wcr} - S_{wco}) - S_{wco}) / (1 - S_{cohy} - S_{wcr})$ when $S_w > S_{w\,\min}$

The initial fractional volume of gas above the gas contact, X_{gi}, is

$$X_{gi} = \frac{S_{gi} - S_{gco}}{1 - S_{lco} - S_{gco}} \qquad \text{[EQ 16.34]}$$

Figure 16.10 Initial fractional volume of gas above the gas contact

where

S_{gi} is the initial gas saturation

S_{gco} is the connate gas saturation

S_{gcr} is the critical gas saturation

S_{lco} is the connate liquid saturation

$= S_{ocog}$ (oil/gas systems)

$= S_{ocog} + S_{wco}$ (oil/water/gas systems)

$= S_{wco}$ (gas/water systems)

S_{lcr} is the critical liquid saturation

If $S_g > S_{gi}$ then

$$X_g = 1 \text{ when } S_g = S_{g\,\max} = X_{gi}(1 - S_{lco}) + (1 - X_{gi})(1 - S_{lcr})$$

$$X_g = (S_g - X_{gi}(S_{lcr} - S_{lco}) - S_{gco}) / (1 - S_{lcr} - S_{gco}) \text{ when } S_g < S_{g\,\max}$$

If $S_g < S_{gi}$ then

$$X_g = 0 \text{ when } S_g = S_{g\,\min} = X_{gi}S_{gcr} + (1 - X_{gi})S_{gco}$$

$$X_g = (S_g - X_{gi}(S_{gcr} - S_{gco}) - S_{gco}) / (1 - S_{lco} - S_{gcr}) \text{ when } S_g > S_{g\,\min}$$

Variation of the sigma factor with drainage process

In a typical water-wet system, production of oil from the matrix will be dominated by two different processes in different regions of the reservoir, depending on the displacing fluid. In the water invaded zone the production will be by water imbibition (assisted by gravity drainage), and in the gas invaded zone by gravity drainage (retarded by capillary effects).

A single sigma factor may not be sufficient to model both these cases. Typically, gravity drainage is a slower process than imbibition because the displaced fluid flows mainly in one direction (vertically).

ECLIPSE 100

The `SIGMAGD` keyword is available in 3-phase dual-porosity runs to input a second sigma value for gas-oil recovery processes. The matrix-fracture transmissibility based on `SIGMAGD` will be used to calculate the oil flow when the following conditions are satisfied:

- The gravity drainage model is active,
- Oil flow is from the matrix to the fracture,
- The gravity drainage head associated with gas in the fracture is greater than the head associated with water.

Rock volume and cell activation in dual porosity thermal calculations

Rock Volume

In single porosity calculations, the rock volume is simply the cell volume less the pore volume. In dual porosity calculations, this leads to a double counting of rock volume, as the cell volume is in reality common to all the cells of different porosity sharing the same location. To prevent this, the rock volume, defined as the average cell volume for all porosities less the total pore volume in all cells in that location, is shared between the porosities. Each cell receives the same rock volume as its pore volume. Any remaining rock volume is given to the matrix. If there is not enough rock volume to achieve this, the rock volume is shared in proportion to the cell pore volume.

As a special case, if a cell in one porosity has ACTNUM=2 (see below), meaning that the cell is rock only, then the remaining rock is shared equally among all the porosities for that cell location.

As a special case, if a cell in one porosity has ACTNUM=3 (see below), meaning that the cell is pore-volume only, then the pore volume assumed is the physical volume divided by the number of porosities. If an asymmetric pore volume distribution is required, the net-to-gross value can be adjusted as a weighting factor.

If this option does not achieve the desired rock distribution, the user can override it with the ROCKSPLV keyword.

ACTNUM

The ACTNUM keyword allows the user to deactivate cells. In thermal calculations, the user can also deactivate either the fluids or the rock to give an energy-only or flow-only calculation in that cell.

In dual porosity, total deactivation of the matrix causes deactivation of the fracture, and the other way round. But rock-only or flow-only deactivation only affects the porosity specified. The specification of rock-only or flow-only cells does not affect the distribution of rock among the porosities, as described above; however, the rock or pore volume calculated may be set to zero by the ACTNUM flag.

There is no automatic default for synchronizing ACTNUM rock-only or flow-only deactivations with the rock distribution. It is suggested that the user takes advantage of the possibility to deactivate using porosity (keyword PORO) or pore volume (keyword PORV) and ROCKSPLV to control deactivation through the volume sizes on a per cell, per porosity basis.

If a cell is rock-only (ACTNUM=2) then the rules for apportioning rock volume are modified; see above. If a cell is pore-volume only (ACTNUM=3) then the rules for apportioning pore volume are modified; see above.

Integrated capillary pressure option

When a gravity drainage option is in use, the final recovery from the matrix is determined by the balance between the capillary pressure and gravity forces. The final saturation distribution in the matrix block can be thought of as a transition zone where at each height the capillary pressure is $P_c = \Delta\rho \cdot g \cdot h$.

Within the dual porosity matrix representation, it is desirable that the final recovery should be equal to the final recovery from the continuous matrix medium. This can be achieved by modifying the capillary pressure curves in the matrix to take account of the fact that the rock P_c curves need to be integrated over the matrix block height to calculate the average saturation.

The integrated P_c option is activated by using the `INTPC` keyword in the `PROPS` section. The appropriate modified capillary pressure curves are written to the `PRINT` file for inspection if either the `SWFN` or `SGFN` mnemonics are used in the `RPTPROPS` keyword.

The integration procedure for the gas-oil capillary pressure is as follows:

We start with a piece-wise linear table

$p_c = P_c(S_g)$ and $S_g = P_c^{-1}(p_c)$

where P_c^{-1} is the inverse capillary pressure function.

Now consider a matrix block of height h with a contact $(p_c = 0)$ at the bottom of the block and a capillary pressure P at the top.

Figure 16.11 Integrated capillary pressure

The average gas saturation $\overline{S_g}$ in the matrix block up to the height h equivalent to $p_c = P$ is given by

$$\overline{S_g} = \frac{1}{P} \cdot \int_0^P P_c^{-1}(p)\, dp \qquad \text{[EQ 16.35]}$$

We can now set up a modified capillary pressure function such that the capillary pressure at a saturation $\overline{S_g}$ is equal to the gravity drainage head for a block of height h and saturation $\overline{S_g}$.

The gravity drainage model accounts for the gravity head within a grid block, by assuming vertical equilibrium. (Note that if the gravity drainage option is not used the gravity head within a grid block is ignored.)

For the gravity drainage model selected by GRAVDR, the gravity head P_{gdr} is proportional to the mobile oil saturation, assuming the fracture is filled with gas:

$$P_{gdr} = \frac{S_t - \overline{S_g}}{S_t} \cdot \Delta\rho \cdot g \cdot h \qquad \text{[EQ 16.36]}$$

Where

S_t is $1 - \mathrm{SWCO} - \mathrm{SOGCR}$

SWCO is the connate water saturation

SOGCR is the oil saturation at critical gas

$\Delta\rho$ is the difference between the gas and oil densities

g is the gravity constant

h is the block height.

Now the pressure at the top of the matrix block P is given by

$P = \Delta\rho \cdot g \cdot h$

We can thus construct a new capillary pressure function, such that at a saturation $\overline{S_g}$ the capillary pressure is exactly equal to P_{gdr} and hence the equilibrium saturation will be at (and only at) $S_g = \overline{S_g}$.

The constructed capillary pressure function P_c' should have $P'_c(\overline{S_g}) = P_{gdr}$, and thus

$$P'_c(\overline{S_g}) = \frac{S_t - \overline{S_g}}{S_t} \cdot P \qquad \text{[EQ 16.37]}$$

where P is the true capillary pressure at the top of the block when the equilibrium saturation is $\overline{S_g}$. This can be calculated by looking up the table of $\overline{S_g}$ versus P given by Equation [EQ 16.35], which is stored in ECLIPSE as a table.

For the alternative gravity drainage model selected by GRAVDRM, the gravity head P_{gdr} is proportional to *P/2*, assuming the fracture is filled with gas. This condition requires that oil is not re-infiltrating into the matrix, and this possibility should be excluded using the GRAVDRM keyword.

Notes

1 By construction, for any block height h the ‘correct’ final saturation is achieved by running the simulator to a stable state when:

$P'_c(S)$ = gravity drainage head,

$S = \bar{S}$ for that block height.

2 At other saturations ($S \neq \bar{S}$) a potential difference generates the usual flow to and from the block.

3 During equilibration the modified capillary pressure is accounted for. Note however that the equilibrium saturation is not the same as in the equivalent drained block, as the fluids above the contact are assumed to be only connate water + gas, where a drained block includes the residual oil.

Discretized matrix model (ECLIPSE 100)

Traditional dual porosity models assume that the matrix to fracture flow is in steady state, and thus the matrix cell can be regarded as a single cell. In some cases, however, for example a well test in a dual porosity system, the transient nature of the matrix to fracture flow can be important. To model these systems a discretized matrix model is available, which sub-divides each matrix cell into a series of nested sub-cells, allowing the simulator to predict the transient behavior.

The matrix cells are still only connected to their corresponding fracture cell, but flow to the matrix surface is supplied from a 1D grid system. Each matrix grid cell could be regarded as a "Russian Doll" of nested matrix sub-cells.

Matrix sub-grid

The matrix sub-division is set up automatically by ECLIPSE once you have specified the number of sub-cells to split each matrix cell into, using the `NMATRIX` keyword in the `RUNSPEC` section. The differing properties across the field are specified in the usual way by using the standard `GRID` section keywords including `SIGMAV` to specify the matrix-fracture transmissibility.

The generated grid is logarithmic away from the fracture wall and can represent a linear (1D), radial (2D) or spherical (3D) geometry. You can control the size of the outer matrix sub-cell (as a fraction of the corresponding fracture cell) and the geometry type using the `NMATOPTS` keyword. Once the outer sub-cell size and the number of sub-cells required are known, the growth factor for each individual matrix cell can be calculated. The maximum and minimum growth factors across the field are reported in the `PRINT` file.

Linear geometry

If the geometry is assumed to be 1D then the `SIGMA` factor and hence the matrix-fracture transmissibility are assumed to represent 1D flow.

Figure 16.12 Nested matrix sub-cells in linear geometry

The matrix-fracture transmissibility is assumed to be the transmissibility associated with flow over the distance d, through area A. The sub-cell transmissibilities are then calculated assuming a 1D logarithmic grid over the same matrix volume.

Cylindrical and spherical geometries

In the 2D (Cylindrical) and 3D (Spherical) flow options the geometry is taken to be Cartesian to enable simple mapping from the input matrix-fracture transmissibility.

Figure 16.13 Nested matrix sub-cells in cylindrical and spherical geometry

In the 2D case the diagram represents a square cylinder with the matrix-fracture transmissibility assumed to represent four times the flow over the distance d through area A. In the 3D case the diagram represents a cube with the transmissibility taken as sum of six flows out of the six faces.

In each case the sub-cell transmissibilities are calculated assuming a logarithmic distribution of sub-cell sizes away from the fracture walls.

Initialization, solution method and efficiency

The internal sub-matrix cells cannot be explicitly initialized in the way that the normal matrix cells can, and so the `EQUIL` keyword must be used to perform an initialization.

The matrix sub-grid forms a series of separate 1D flow systems connected through the outer sub-cells only to the corresponding fracture cells. In the solution matrix structure these flows lead to a set of tri-diagonals connected to the fracture system. This structure is used in the pre-conditioning step by eliminating the tri-diagonals before the fracture system is solved, effectively reducing the problem to a set of flow equations for the fracture only. Once the fracture equations have been solved a back substitution gives the matrix sub-grid solution.

Clearly, discretizing the matrix increases the total number of cells in the model. However, the efficiency of the above procedure ensures that increasing the number matrix sub-cells will have a significantly smaller effect on the run time than a similar increase in the number of simulation grid blocks.

Reporting the matrix solution

The solution for separate sub-cells (or rings) of the matrix can be output as arrays over the simulation grid by setting any of the first 5 data items in `RPTSCHED` to 2. If an individual sub-cell solution is required as a function of time, the following summary keywords are available:

`BPR`, `BRS`, `BRV`, `BOSAT` and `BWSAT`.

The matrix sub-cell is specified by appending its ring number to the end of the keyword, for example:

```
BOSAT7
 1 1 2 /
/
```

The ring number starts from 1 for the outer sub-cell and increases towards the center of the matrix cell. Note that ring 1 is equivalent to the keyword without a ring number and represents the matrix sub-cell adjacent to the fracture.

Restrictions

The option cannot at present be used with the following:

- gravity drainage models (GRAVDR, GRAVDRM)
- Local Grid Refinement option
- Dual Permeability model
- Parallel option

and must use the following to initialize the sub-matrix cells

- EQUIL

Single Medium Conductive Fractures (ECLIPSE 300)

An alternative dual porosity formulation in ECLIPSE 300 is the Single Medium Conductive Fractures development. This is based on a generalization of the technique described in van Lingen et al, 2001 [Ref. 100].

The technique allows the incorporation of the effects of conductive fractures into a single medium model (that is not explicitly using a dual system of porosities or permeabilities) by modification of grid properties and the generation of pseudo tables. It is focused on modeling fractures on scales that are too large to be homogenized using the traditional approach.

For further details of the technique, please refer to [Ref. 100]. The sections below assume that the reader has familiarity with the paper, and outline the approach taken in ECLIPSE 300, including extensions and generalizations on the technique described in the paper, and how to use the ECLIPSE 300 development.

The Single Medium Approach

For a given input fracture swarm distribution map, and associated fracture properties, there are three subsequent main areas of implementation:

- Grid property modifications
- Pseudo generation
- Well PI modifications

The implementation of the technique in these areas is described in the following paragraphs.

Grid Property Modifications

Firstly grid block properties are modified to take account of the physical void introduced by the fractures. The porosities are increased by proportional volume averaging, and permeabilities flow-averaged using Darcy's Law, either arithmetically (parallel to fracture) or harmonically (perpendicular to the fracture).

The approach taken in Eclipse 300 differed to that in the paper in

- the full use of all cell dimensions (rather than assuming square cells)
- the calculation of directional permeability enhancements (parallel and perpendicular components)
- allowing for vertical permeability enhancement
- allowing for fracture volumes in porosity modifications.

One further modification is to recognize fractures being defined in a zig-zag rasterized format. When the path of a fracture changes direction by 90 degrees within one cell, the effective length of the fracture is halved. This attempts to model the intention that the fracture does not traverse the cell in question twice, but rather somehow ‘turns a corner’.

Pseudo Generation

The saturation curves of the fractures and affected grid blocks are combined under the assumption that oil is displaced from fractures by either gas or water before any oil is displaced from the matrix volume of a grid block, which requires that any capillary imbibition is dominated by viscous and gravity effects. Under this assumption, the saturation tables are averaged by determining end point saturations from mass conservation, and end point relative permeabilities by Darcy's Law arithmetic averaging of effective phase flows. The original fracture tables are then used in a scaled form during the fracture invasion by gas or water, and scaled matrix tables are used for matrix invasion. Notice that (as in the paper) the fracture tables are assumed straight lines; no attempt is made to combine the full shape of the curve with that of the matrix.

Since the permeabilities and saturation tables used potentially vary on a grid block basis, the above could feasibly result in one set of pseudo tables for each affected cell. To reduce the dimensionality, a grouping technique is used that averages the pseudo end points used over cells with common attributes, these being saturation region, permeability value, and the number of contained fractures. This extended the paper's treatment which assumed that layers represented the saturation regions.

Note Note that capillary curves are not included in this treatment, and are merely extrapolated at constant value from the matrix end-point values.

Well PI

By default, wells completed in grid blocks containing conductive fractures will experience enhanced flows due to the increased effective permeability, due to both the permeability and relative permeability changes. [Ref. 100] argues that this only arises if the fracture actually cuts the well bore in a connection cell. Because of this, you are required to specify whether or not a well connection cell is to be modified by this treatment by indicating whether it is cut by fractures or not. If not, then the well PI is modified so as to reverse the changes in permeability, and the connection cell saturation table reverts to that of the original matrix block. You are also given the chance to enter a PI multiplier to indicate perhaps some degree of connection with conductive faults.

Using the Conductive Fractures development

The conductive fractures model is activated by specification of the `SCFDIMS` keyword in the `RUNSPEC` section, which also sets up dimensioning defaults for reading in the fracture details, and also defaults for the pseudo grouping algorithms used.

The conductive fractures themselves should be specified using the `CONDFRAC` keyword, one for each fracture being modeled. This specifies the path of fracture planar segments through the grid blocks, the effective aperture, fracture permeability, and saturation table number to use. The format is similar to the `FAULTS` keyword, but note that the fractures are assumed to pass through a block, whereas faults run alongside blocks. For this reason, when a 'corner' is detected in the fracture, the contribution from the two intersecting segments is halved, to reflect the fact that the corner is formed notionally at the center of the cell, rather than around the edges.

The well intersection behavior is controlled using the `WELLCF` keyword.

Compatibility

The feature has been made explicitly compatible with the restart mechanism, and will run in parallel mode. It is currently restricted to family (ii) saturation table keywords - `SWFN`, `SGFN` and `SOF2/3`.

The development is known not to be compatible with the following options:

- radial grids
- analytic saturation functions
- geomechanics
- MPFA
- Up-front memory
- LGRs

Environmental Tracers

Chapter 17

Introduction

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL

The Environmental Tracers option is an extension to the ECLIPSE 100 passive Tracer Tracking model; it is not available in ECLIPSE 300. The option enables the modeling of contaminants and other substances as they flow within a host water, oil or gas phase.

The passive tracer tracking models give you the ability to track up to 50 tracer substances within a single model. The Environmental Tracer option extends the modeling to account for adsorption of the tracer on to the bulk rock, for decay of the tracer over time, and for molecular diffusion of the tracer. It is possible to model adsorption, decay and diffusion within a single tracer.

The Environmental Tracer model

As for passive tracers, the flow of an environmental tracer through the porous medium is assumed to have no influence on the flow of the water and hydrocarbon phases as well as the flow of other tracers. Environmental tracers are therefore solved in a similar fashion to passive tracers at the end of a time step after convergence of the oil, water and gas equations.

For an environmental tracer present in a single phase the governing equation is:

$$\frac{d}{dt}\left(\frac{VSC}{B}\right) + \frac{d}{dt}\left(V\rho_r C^a \frac{1-\phi}{\phi}\right) = \sum\left[\frac{Tk_r}{B\mu}(\delta P - \rho g D_z)C + DFD_c S\delta C\right] + QC - V\frac{S}{B}\lambda C \qquad \text{[EQ 17.1]}$$

where

S denotes the host phase saturation

C denotes the flowing tracer concentration

C^a denotes the adsorped tracer concentration

ρ_r denotes the mass density of the rock formation

ϕ denotes the porosity

ρ denotes the host phase density

μ denotes host phase viscosity

D_z denotes the cell center depth.

B denotes the host phase formation volume factor

T denotes the transmissibility

k_r denotes the host phase relative permeability

V denotes the block pore volume

Q denotes the host phase production rate

P denotes the host phase pressure

g denotes the gravity acceleration

D_c denotes the tracer diffusion coefficient

DF denotes the diffusivity

λ denotes the tracer decay constant

Σ denotes the sum over neighboring cells

Treatment of adsorption

Adsorption is treated as an instantaneous effect in the model. The effect of tracer adsorption is to strip tracer from the leading edge of a tracer front. Desorption effects may occur as the tracer concentration decreases.

The adsorption model can handle both stripping and desorption effects. You specify an adsorption isotherm, which tabulates the saturated rock adsorbed concentration versus the local tracer concentration in solution. As an alternative to tabulating, the isotherm coefficients can be supplied for an analytical isotherm which allows for dependencies of adsorption on rock permeability.

There are currently two adsorption models which can be selected. The first model ensures that each grid cell retraces the adsorption isotherm as the tracer concentration rises and falls in the cell. The second model assumes that the adsorbed tracer concentration on the rock may not decrease with time, and hence does not allow for any desorption. More complex models of the desorption process may be implemented if required.

Initial input tracer concentrations can either partition between the fluid and solid or can be honored with additional adsorbed tracer added to be in equilibrium.

Treatment of decay

Decay of the tracer is modeled as a simple half-life process. You specify a half-life for each tracer, and the tracer starts decaying as soon it is injected into the reservoir.

Treatment of diffusion

The diffusive flow of tracer from cell i to a connected cell j is given by [EQ 17.1]:

$$F_t = DF \cdot D_c S(C_{ci} - C_{cj}) \qquad \text{[EQ 17.2]}$$

The diffusivity (DF) is analogous to the transmissibility and has the form:

$$DF = \frac{A \cdot \phi}{d} \qquad \text{[EQ 17.3]}$$

where

A denotes the interface area between the two cells

d denotes the distance between the cell centers

The details of the diffusivity calculation can be found under "Diffusion" on page 211.

In a system where there is significant convection of the tracer, molecular diffusion is likely to be a relatively small effect. Numerical dispersion associated with the discrete approximation of the flow equation is likely to dominate any molecular dispersion. However, the diffusion model is useful when the convection is small. In the limiting case of zero permeability, the only mechanism for flow is by diffusion; this kind of situation may occur when modeling a contaminant initially stored in concrete.

Partitioned tracers

Partitioned tracers allow ECLIPSE 100 to solve a range of problems that could not be tackled otherwise. For example, if marked gas is injected into a reservoir it is possible that the marker may dissolve in the water. The partitioned tracer option can model this behavior. With a single phase tracer in, say, the gas phase, ECLIPSE solves a conservation equation for the total amount of tracer in a grid block taking into account the inflow and outflow of gas, cf. [EQ 17.1]. With a partitioned tracer one or more additional phases can be nominated (in the `TRACER` keyword) into which the tracer may also dissolve.

Note The partitioned tracer functionality can be used independently of adsorption, decay and diffusion. Hence, it is possible to have a standard passive tracer which partitions into multiple phases.

Standard partitioned tracer

A standard partitioned tracer can exist in two phases which are specified in the `TRACER` keyword. For input and output purposes these two phases are regarded as being the ‘free’ and ‘solution’ phases, so that the conventional ECLIPSE 100 keywords for two-phase tracers can be used. The ‘free’ phase can be thought of as the reference phase for the tracer.

If we use subscripts ‘f’ and ‘s’ to denote the free and solution phases, respectively, the governing equation for a standard environmental partitioned tracer is:

$$\frac{d}{dt}\left(V\left[\frac{S_f}{B_f}C_f + \frac{S_s}{B_s}C_s\right]\right) + \frac{d}{dt}\left(V\rho_r C^a(C_s)\frac{1-\phi}{\phi}\right) = \sum[F_f + F_s] + Q_f C_f + Q_s C_s - V\frac{S_s}{B_s}\lambda C_s \qquad \text{[EQ 17.4]}$$

$$F_f = \frac{Tk_{rf}}{B_f\mu_f}(\delta P - \rho_f g D_z)C_f \qquad \text{[EQ 17.5]}$$

$$F_s = \frac{Tk_{rs}}{B_s\mu_s}(\delta P - \rho_s g D_z)C_s + DFD_c S_s \delta C_s$$

The concentration of the tracer in the solution phase is not independent of the concentration in the free phase. The concentrations are related through the partitioning functions:

$$K(P) = \frac{C_s}{C_f} \qquad \text{[EQ 17.6]}$$

where the functions $K(P)$ are entered in tabular form in the TRACERKP keyword and are required for each grid block. The particular table to use is specified by using the TRKPFxxx keyword in the REGIONS section. A table number to use within the wellbore is also required and is specified in item 5 of the TRACER keyword.

As shown in [EQ 17.4] and [EQ 17.5], when a standard partitioned tracer is used in conjunction with adsorption, decay and diffusion, it is assumed that these processes happen only in the solution phase. For example, the adsorption function, $C^a(C_s)$, in [EQ 17.4] is evaluated based on the tracer concentration in the solution phase.

Multi-partitioned tracers

The multi-partitioned tracer option is an extension and generalization of standard partitioned tracers. A multi-partitioned tracer, which is defined by specifying MULT in the fourth item of the TRACER keyword, can partition into any number of phases and have phase specific adsorption, decay and diffusion parameters.

The governing equation for a generalized, multi-partitioned tracer is:

$$\frac{d}{dt}\left(V\sum_j \frac{S_j}{B_j}C_j\right) + \frac{d}{dt}\left(V\rho_r C^a(C)\frac{1-\phi}{\phi}\right) = \sum\sum_j F_j + \sum_j Q_j C_j - V\sum_j \frac{S_j}{B_j}\lambda_j C_j \qquad \text{[EQ 17.7]}$$

$$F_j = \frac{Tk_{rj}}{B_j\mu_j}(\delta P - \rho_j g D_z)C_j + DFD_{cj}S_j\delta C_j \qquad \text{[EQ 17.8]}$$

where the summation over j indicates the phases in which the tracer is present, typically (a subset of) oil, water and gas.

As for standard partitioned tracers, the concentrations of the tracer in different phases are related through pressure dependent partitioning functions. For a multi-partitioned tracer these functions are entered in tabular form in the TRACERKM keyword. The reference phase is taken as the phase specified in the second item of the TRACER keyword. TRACERKM then specifies partitioning functions relating the concentration in each of the solution phases to the concentration in the reference phase. For example, for a tracer partitioning into oil, water and gas with water as the reference phase, TRACERKM holds two columns of data specifying the functions:

$$K_{ow}(P) = \frac{C_o}{C_w} \qquad K_{gw}(P) = \frac{C_g}{C_w} \qquad \text{[EQ 17.9]}$$

Decay and diffusion can be modeled in all phases with phase specific decay and diffusion constants. For adsorption, the concentration to use when evaluating the adsorption function can be specified in the sixth item of the TRACER keyword. The options are to use either one of the concentrations in the phases that the tracer partitions into, or to use the overall, pore volume averaged concentration, that is:

$$C = \sum_j \frac{S_j}{B_j}C_j \qquad \text{[EQ 17.10]}$$

Initialization of partitioned tracers

The initial partitioned tracer concentrations are input in tabular form using either of the keywords TBLK or TVDP. The keyword TBLK specifies the initial concentrations of a tracer in each grid block, whereas the keyword TVDP specifies tables of depth values versus tracer concentration to be used to initialize the tracer concentrations. Only the tracer concentration for the free (reference) phase is specified in the SOLUTION section. The initial free phase tracer concentrations are then updated using the formula

$$C_f = \frac{C_f^{ini}\ FIP_f}{\sum K_{jf}(P) FIP_j} \qquad \text{[EQ 17.11]}$$

where FIP_j is the fluid in place for phase j and $K = 1$ for $j = f$. The concentration of the solution phase(s) is then calculated from the relationship $C_j / C_f = K_{jf}(P)$ which has been input using the TRACERKP or TRACERKM keyword. In this way the concentration specified in the SOLUTION section is distributed between the phases while honoring the partitioning functions.

Injection and production

The tracer concentration in the free and solution phase in the wellbore is calculated depending on whether the well is of production or injection type. If the well is injecting, the concentration of the free phase is taken from the keyword WTRACER and the concentration of the solution phase(s) is then calculated using the relationship $C_j / C_f = K_{jf}(P)$, where P is taken to be the pressure in the wellbore. If the well is producing, the concentration in the solution and free phases are calculated by summing the concentrations in the well connections, where the connection concentration of the free and solution phases are taken to be the grid block values.

Note It is possible to combine the partitioned tracer functionality with the surfactant model, thus allowing surfactant to partition into more than one phase. This is done by specifying a tracer called 'SUR' which ECLIPSE recognizes as being the surfactant. For a surfactant, adsorption is calculated based on the water phase concentration, that is the 'free' phase.

Using the environmental tracer option

A passive tracer is set up in the normal manner using the following ECLIPSE 100 keywords to specify the tracer data:

Table 17.1 Keywords used to specify environmental tracer data

Keyword	Section	Purpose
TRACERS	RUNSPEC	Specify the number of tracers in each phase (NOTRAC, NWTRAC, NGTRAC) and the number of environmental tracers (NETRAC).
TRACER	PROPS	Specify the host phase of each tracer.
TNUM	REGION	Specify the region numbers to be used for initializing the concentration of tracer in each grid block and specify link cells to tables of initial tracer concentration with respect to depth.
TBLK or TVDP	SOLUTION	Specify the initial tracer concentration as either the numerical value or as tables of depth versus tracer concentration.
WTRACER	SCHEDULE	Inject tracer through wells.
RIVTRACE	SCHEDULE	Inject tracer in rivers.

To activate the Environmental Tracer option, the parameter NETRAC in keyword TRACERS in the RUNSPEC section should be set greater than zero. NETRAC should be set equal to (or greater than) the number of tracers within the model that either adsorb, decay or diffuse; it need not be set equal to the total number of tracers in the model, if not all of them adsorb, decay or diffuse.

To activate the partitioned tracer option the keyword PARTTRAC must be entered in RUNSPEC section.

The adsorption, decay and diffusion data is supplied with the keywords TRADS, TRDCY and TRDIF respectively. These keywords require the tracer name to be appended to the keyword to indicate which tracer the data applies to. For adsorption the ADSORP keyword may be used with the tracer name as argument as an alternative to TRADS. The ADSORP keyword supplies coefficients for a generalized Langmuir isotherm with permeability dependence. For standard partitioned tracers the data entered in TRDIF and TRDCY apply to diffusion and decay in the solution phase. For multi-partitioned tracers, data specific to each phase can be entered in TRDIF and TRDCY.

If the adsorption model is required, the rock mass density needs to be specified by using the TRROCK keyword. The grid block masses can be output by using the 'ROCKMASS' mnemonic in the RPTSOL keyword. The TRROCK keyword is also used to specify the adsorption model and the initial distribution of adsorbed tracer. For multi-partitioned tracers the concentration to use in adsorption calculations can be specified in the sixth item of the TRACER keyword.

A passive tracer can be thought of as a dimensionless marker of the host fluid. However, in the environmental tracer case it may be convenient to ascribe a unit to the tracer. A general unit name can be given to a tracer using the third item in the TRACER keyword. Note that the adsorption data (TRADS or ADSORP keyword) should be based on this unit.

The diffusivity values can be output by setting mnemonics 'DIFFX', 'DIFFY' and 'DIFFZ' in the RPTGRID keyword. Note that the diffusivity associated with non-neighbor connections can be found in the non-neighbor connection list generated by RPTGRID mnemonic 'ALLNNC'.

The tables of tracer adsorption functions, tracer decay and tracer rock properties can be output to the print file by setting the mnemonics 'TRADS', 'TRDCY' and 'TRROCK' in the RPTPROPS keyword.

The quantity of the tracer that has decayed or been adsorbed can be output to the print file by setting either mnemonic 'TRADS' or 'TRDCY' in the RPTSCHED keyword. Two families of SUMMARY mnemonics are available to output the adsorption and decay: FTADS, RTADS, BTADS and FTDCY, RTDCY, BTDCY. As with other tracer SUMMARY section mnemonics, the tracer name should be appended on to the keyword.

Tracer in place reports can be output to the print file by specifying the 'FIPTR' mnemonic in RPTSCHED. For a standard partitioned tracer the amount of the tracer in the free and solution phases will be reported. For a multi-partitioned tracer the amount reported in the 'solution' phase is the sum over all solution phases.

Restrictions

At present the adsorption, decay and diffusion options are only available for single phase and partitioned tracers. It is not possible to model adsorption, decay or diffusion for tracers existing in a phase that can dissolve in another phase. For example, it is not possible to model a decaying tracer in the gas phase of a model in which the gas can dissolve in the oil.

Single phase environmental tracers employing adsorption and decay can use either the standard tracer solver method or the "Cascade" or TVD algorithms, which limit numerical diffusion (for the latter two, specify 'DIFF' in item 5 of the RUNSPEC section keyword TRACERS and for the TVD algorithm use the TRACTVD keyword). However, if a tracer uses the diffusion model it cannot use the Cascade algorithm. Hence even if 'DIFF' is set in TRACERS, the default solution method is employed for any tracers with the diffusion option active. TRACTVD can be used with diffusion (and dispersion), but not when LGRs are present.

All partitioned tracers are solved using the standard tracer solver.

If the adsorption option is active, the tracer equations are non-linear. Because of this non-linearity, it is likely that the Cascade method is faster than the default method, providing, of course the tracer does not employ the diffusion option as well.

Environmental tracers cannot be introduced in a restart run.

Changing the pore volume in the EDIT section using PORV is not allowed with the environmental tracer option.

Summary of keywords

RUNSPEC section

The RUNSPEC keywords are:

Table 17.2 RUNSPEC keywords

Keyword	Description
TRACERS	Specify dimensions and options for tracers.
EQLDIMS	Specify dimensions for tracer concentration versus depth tables used for equilibration.
PARTTRAC	Enables and dimensions the partitioned tracer option.

PROPS section

The PROPS keywords are:

Table 17.3 PROPS keywords

Keyword	Description
RPTPROPS	Controls output from the PROPS section: TRACER outputs passive tracer names TRADSxxx, TRROCK and TRDCYxxx output environmental tracer properties.
TRACER	Define the tracers.
TRACERKP	Define the partitioning functions K(P) for partitioned tracers (max two phases).
TRACERKM	Define the partitioning functions K(P) for partitioned tracers (any no. of phases).
TRADS	Specify adsorption data (tabulated isotherm).
ADSORP	Specify adsorption data (coefficients for analytical isotherm).
TRDCY	Specify decay data.
TRDIF	Specify diffusion data.
TRROCK	Specify rock data for adsorption.

REGIONS section

The REGIONS keywords are:

Table 17.4 REGIONS keywords

Keyword	Description
RPTREGS	Controls output from the REGIONS section: TNUM outputs tracer versus depth region numbers.
TNUM	Specify tracer concentration regions.
TRKPFxxx	Specify partitioning function regions for tracer xxx.

SOLUTION section

The SOLUTION keywords are:

Table 17.5 SOLUTION keywords

Keyword	Description
RPTSOL	Controls output from the SOLUTION section: FIPTR outputs tracer fluid in place reports ROCKMASS outputs rock mass TBLK outputs initial grid block tracer concentrations TVDP outputs initial tracer concentration versus depth tables.
TBLK	Specify initial tracer concentrations.
TVDP	Specify initial tracer concentration versus depth.

SUMMARY section

The following keywords control output of data specific to tracers:

Table 17.6 Environmental tracer SUMMARY output controls

Field	Group	Well	Connection	Region	Block	Information
			CTFR			Tracer Flow Rate (+ or -)
FTPR	GTPR	WTPR				Tracer Production Rate
FTPT	GTPT	WTPT	CTPT			Tracer Production Total
FTPC	GTPC	WTPC				Tracer Production Concentration
FTIR	GTIR	WTIR				Tracer Injection Rate
FTIT	GTIT	WTIT	CTIT			Tracer Injection Total
FTIC	GTIC	WTIC				Tracer Injection Concentration
					BTCNF	Tracer Concentration (Free)

Table 17.6 Environmental tracer SUMMARY output controls

Field	Group	Well	Connection	Region	Block	Information
					BTCNS	Tracer concentration (Solution)
					BTCN#	Tracer concentration in phase #
FTIPT				RTIPT	BTIPT	Tracer In Place (Total)
FTIPF				RTIPF	BTIPF	Tracer In Place (Free)
FTIPS				RTIPS	BTIPS	Tracer In Place (Solution)
FTIP#				RTIP#	BTIP#	Tracer In Place in phase # (1,2,3,...)
				RTFTT		Tracer Flow (Total)
				RTFTF		Tracer Flow (Free)
				RTFTS		Tracer Flow (Solution)
				RTFT#		Tracer Flow in phase # (1,2,3,...)
FTADS				RTADS	BTADS	Tracer adsorption
FTDCY				RTDCY	BTDCY	Decayed tracer
FTIRF	GTIRF	WTIRF	CTIRF			Tracer Injection Rate (Free)
FTIRS	GTIRS	WTIRS	CTIRS			Tracer Injection Rate (Solution)
FTPRF	GTPRF	WTPRF	CTPRF			Tracer Production Rate (Free)
FTPRS	GTPRS	WTPRS	CTPRS			Tracer Production Rate (Solution)
FTITF	GTITF	WTITF	CTITF			Tracer Injection Total (Free)
FTITS	GTITS	WTITS	CTITS			Tracer Injection Total (Solution)
FTPTF	GTPTF	WTPTF	CTPTF			Tracer Production Total (Free)
FTPTS	GTPTS	WTPTS	CTPTS			Tracer Production Total (Solution)
FTICF	GTICF	WTICF	CTICF			Tracer Injection Concentration (Free)
FTICS	GTICS	WTICS	CTICS			Tracer Injection Concentration (Solution)
FTPCF	GTPCF	WTPCF	CTPCF			Tracer Production Concentration (Free)
FTPCS	GTPCS	WTPCS	CTPCS			Tracer Production Concentration (Solution)

The above keywords should be compounded with the name of the tracer to which they refer. For example, if the field production rate of a tracer called OL1 is required on the summary files, then the keyword FTPROL1 should be used in the SUMMARY section.

For partitioned tracers, the concentration, flow, and amount in place in a specific phase can be output by using either the F (free) and S (solution) specifiers, or by using the phase number #. For a tracer partitioning into more than two phases, the F and S specifiers are no longer sufficient to uniquely determine the phase of interest. In this case the phase number must be used. For example, BTCN3OL1 refers to the block concentration of tracer OL1 in phase 3 as specified in the list of phases when defining the tracer partitioning functions (see keyword TRACERKM) . If the S (Solution) specifier is still used for a tracer partitioning into 3 or more phases, the concentration output will be the concentration in the last phase as defined in TRACERKM.

Note BTADS reports the amount of adsorped tracer per unit mass of rock in a given block (e.g. 'kg/kg'). The corresponding region and field mnemonics, RTADS and FTADS, report the total amount of adsorped tracer (e.g. 'kg').

SCHEDULE section

The SCHEDULE keywords are

Table 17.7 SCHEDULE keywords

Keyword	Description
RPTSCHED	Controls output from the SCHEDULE section: FIPTR outputs tracer fluid in place reports TBLK outputs initial grid block tracer concentrations TRADS or TRDCY output tracer adsorption and decay.
WTRACER	Set tracer concentration in injection wells.
RIVTRACE	Set tracer concentration injection in rivers.

Example problem

Tracer names

BBB is the water tracer with no decay or adsorption

XXX is the water tracer that adsorbs and decays

YYY is the water tracer that only decays

ZZZ is the water tracer that diffuses

RUNSPEC

RUNSPEC section

```
RUNSPEC     ================================
TITLE
 ENVIRONMENTAL TRACER OPTION TEST
DIMENS
 100    3    1  /
NONNC
OIL
WATER
METRIC
TRACERS
 0    4    0    2 'NODIFF' /
TABDIMS
 2    1   20   20    6   20 /
REGDIMS
 6    1    0    0    0    1 /
WELLDIMS
 2    3    1    2 /
START
 1 'MAY' 1990  /
```

GRID

GRID section

```
GRID    ============================================================
==
EQUALS
 'DX'    10    /
 'DY'    100   /
 'DZ'    10    /
 'PERMX' 1000  /
 'PERMY' 1000  /
 'PORO'  0.25  /
 'TOPS'  2000   1 10  /
 'TOPS'  2005  11 20  /
 'TOPS'  2010  21 30  /
 'TOPS'  2015  31 40  /
 'TOPS'  2020  41 50  /
 'TOPS'  2025  51 60  /
 'TOPS'  2030  61 70  /
 'TOPS'  2035  71 80  /
 'TOPS'  2040  81 90  /
 'TOPS'  2055  91 100 /
--
-- Zero permeability for tracer ZZZ
--
 'PERMX' 0.0  41 50  2 2  1 1 /
 'PERMY' 0.0  41 50  2 2  1 1 /
/
```

PROPS

PROPS section

```
PROPS     ============================================================
==
SWFN
    .145 .0000 .75
    .220 .0001 .05
    .270 .0004 1*
    .320 .009  1*
    .365 .018  1*
    .438 .043  1*
    .510 .082  1*
    .558 .118  1*
    .631 .187  -.05
    .703 .270  -.25
    .752 .310  -.50
    .795 .370  -1.50
/
    0.0    0.0   0.0
    1.0    1.0   0.0
/
```

```
SOF2
    .205 .000
    .250 .00006
    .300 .0009
    .370 .008
    .440 .027
    .490 .052
    .560 .120
    .635 .228
    .683 .334
    .730 .470
    .780 .644
    .855 1.00
/
    0.0    0.0
    1.0    1.0
/
PVTW
  270 1.030 4.6E-5 0.34 0.0  /
PVDO
  200    1.0     0.47
  280    0.999   0.47
  300    0.998   0.47
/
ROCK
 270 .3E-5 /
DENSITY
 850. 1000. 10. /
--
--  Define the tracers
--
TRACER
 'BBB' 'WAT' /
 'XXX' 'WAT' /
 'YYY' 'WAT' /
 'ZZZ' 'WAT' /
/
--
-- Tracer adsorption data for tracer XXX
--
TRADSXXX
  0.0     0.0000
  1.0     0.0005
 30.0     0.0005  /
  0.0     0.0000
  1.0     0.0005
 30.0     0.0005  /
--
-- Tracer decay data for tracer XXX
--
TRDCYXXX
 10.0 /
```

```
--
-- Tracer decay data for tracer YYY
--
TRDCYYYY
 15.0 /
--
-- Tracer diffusion data for tracer ZZZ
--
TRDIFZZZ
 0.3  /
--
-- Rock data required for tracer adsorption
--
TRROCK
 1  2650 /
 2  2650 /
--
-- Output of Enviromental tracer data
---
RPTPROPS
 'TRROCK'  /
```

REGIONS

REGIONS section

```
REGIONS  ============================================================
FIPNUM
 3*1 2 3 4 94*5
 3*1 2 3 4 34*5 10*6 50*5
 3*1 2 3 4 94*5 /
EQUALS
 'SATNUM' 1 /
/
RPTREGS
 /
```

SOLUTION

SOLUTION section

```
SOLUTION  ===========================================================
EQUIL
 2600 270 2025 /
TBLKFBBB
 300*0.0 /
TBLKFXXX
 300*0.0 /
TBLKFYYY
300*0.0 /
```

```
TBLKFZZZ
 100*0.0
  40*0.0  10*1.0  50*0.0
 100*0.0   /
RPTSOL
 'PRES'  'SOIL'  'SWAT'  'RESTART=2'  'FIP=2'
 'OILAPI'  'FIPTR=2'  'TBLK'  'FIPPLY=2'
 'SURFBLK'  'FIPSURF=2'  'TRADS'  /
```

SUMMARY

SUMMARY section

```
SUMMARY  ============================================================
WBHP
/
FWIR
FOPR
FOPT
FTPRBBB
FTPTBBB
FTIRBBB
FTITBBB
FTICBBB
BTCNFBBB
 1 1 1 /
/
FTPRXXX
FTPTXXX
FTIRXXX
FTITXXX
FTIPTXXX
BTCNFXXX
 1 1 1 /
/
FTADSXXX
RTADSXXX
 2 /
BTADSXXX
 2 1 1 /
/
FTPRYYY
FTPTYYY
FTIRYYY
FTITYYY
--
-- Plot log(FTIPTYYY) :vs: time to see the pure decay.
--
FTIPTYYY
BTCNFYYY
 1 1 1 /
/
```

```
FTADSYYY
FTDCYYYY
RTADSYYY
 2 /
BTADSYYY
 2 1 1 /
/
BTDCYYYY
 2 1 1 /
/
BOSAT
 1 1 1 /
/
WTPRBBB
'OP' /
WTPRXXX
'OP' /
WTPRYYY
'OP' /
WTPRZZZ
 'OP' /
RUNSUM
```

SCHEDULE

SCHEDULE section

```
SCHEDULE  ============================================================
RPTSCHED
 'PRES'  'SOIL'  'SWAT'  'RESTART=2'  'FIP=2'
 'WELLS=2'  'SUMMARY=2'  'CPU=2'  'NEWTON=2'
 'OILAPI'  'FIPTR=2'  'TBLK'  'FIPSALT=2'
 'TUNING'  'SURFBLK'  'SURFADS'  'FIPSURF=2'
 'TRADS'  /
WELSPECS
 'OP'  'G'   100 2   2600  'OIL'  /
 'INJ' 'G'   1   2   2600  'WAT'  /
/
COMPDAT
 'OP      '   2*   1   1 'OPEN'   0  .0   157E-3 /
 'INJ     '   2*   1   1 'OPEN'   0  .0   157E-3 /
/
WCONPROD
'OP' 'OPEN' 'RESV' 4* 250 0.0 4* /
/
WCONINJ
'INJ' 'WAT' 'OPEN' 'RESV' 1* 250 0.0 'NONE' 4* /
/
```

```
--
-- Inject tracer for 40 days.
--
WTRACER
 'INJ' 'BBB'  30.0 /
 /
WTRACER
 'INJ' 'XXX'  30.0 /
 /
WTRACER
 'INJ' 'YYY'  30.0 /
 /
TSTEP
 1 9 30
--
-- End tracer injection
--
WTRACER
 'INJ' 'BBB'  0.0 /
 /
WTRACER
 'INJ' 'XXX'  0.0 /
 /
WTRACER
 'INJ' 'YYY'  0.0 /
 /
TSTEP
 1 9 90 9*100   /
END
```

Equations of State

Chapter 18

Two-parameter equations of state

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

ECLIPSE 300 incorporates four equations of state and two additional variations to the Peng-Robinson equation.

When an equation of state is selected, this is used to obtain Z-factors and phase fugacities, to be used to define inter-phase equilibrium and fluid densities.

The equations of state are:

- PR: Peng-Robinson
- RK: Redlich-Kwong
- SRK: Soave-Redlich-Kwong
- ZJ: Zudkevitch-Joffe-Redlich-Kwong.

These equations of state are implemented in generalized form using Martin's equation, [Ref. 10] and [Ref. 11].

The generalized form of such an equation of state is:

$$Z^3 + E_2 Z^2 + E_1 Z + E_0 = 0 \qquad \text{[EQ 18.1]}$$

with:

$$E_2 = (m_1 + m_2 - 1)B - 1 \qquad \text{[EQ 18.2]}$$

$$E_1 = A - (m_1 + m_2 - m_1 m_2)B^2 - (m_1 + m_2)B$$

$$E_0 = -[AB + m_1 m_2 B^2 (B + 1)] \qquad \text{[EQ 18.3]}$$

The coefficients m_1 and m_2 depend upon the equation used. Refer to Table 18.1 for details.

Table 18.1 The coefficients, m1 and m2, variation with the equation of state

Equation of State	m1	m2
Redlich-Kwong	0	1
Soave-Redlich-Kwong	0	1
Zudkevitch-Joffe	0	1
Peng-Robinson	$1+\sqrt{2}$	$1-\sqrt{2}$

The cubic equation for the Z-factor may be solved to obtain Z-factors for the liquid and vapor phases. Generally three solutions are obtained.

The distinction between the liquid and the vapor phase is then made by choosing the smallest root as Z-factor for the liquid phase, and the largest root as Z-factor for the vapor phase.

Fugacity coefficients are calculated using:

$$\ln(f_i/(px_i)) = -\ln(Z-B) + \frac{A}{(m_1-m_2)B}\left(\frac{2\Sigma_i}{A}-\frac{B_i}{B}\right)\ln\left(\frac{Z+m_2B}{Z+m_1B}\right)+\frac{B_i}{B}(Z-1) \qquad \text{[EQ 18.4]}$$

where:

$$\Sigma_i = \sum_j A_{ij}x_j \qquad \text{[EQ 18.5]}$$

$$A = \sum_{j=1}^{n}\sum_{k=1}^{n} x_jx_kA_{jk} \qquad \text{[EQ 18.6]}$$

$$B = \sum_{j=1}^{n} x_jB_j \qquad \text{[EQ 18.7]}$$

$$A_{jk} = (1-\delta_{jk})(A_jA_k)^{1/2} \qquad \text{[EQ 18.8]}$$

and δ_{jk} are binary interaction coefficients, normally between hydrocarbons and non-hydrocarbons.

These four equations express the mixing laws used in all the equations of state.

The variables are defined by the following equations:

$$A_j = \Omega_a(T,j)\frac{P_{rj}}{T_{rj}^2} \qquad \text{[EQ 18.9]}$$

$$B_j = \Omega_b(T,j)\frac{P_{rj}}{T_{rj}} \qquad \text{[EQ 18.10]}$$

$\Omega_a(T,j)$ and $\Omega_b(T,j)$ in [EQ 18.9] and [EQ 18.10] respectively are functions of the acentric factor w_j and the reduced temperature T_{rj}.

- For Redlich-Kwong:

  $$\Omega_a(T,j) = \Omega_{a_o}T_{rj}^{-1/2}$$

$$\Omega_b(T,j) = \Omega_{b_o}$$

- For Soave-Redlich-Kwong:

$$\Omega_a(T,j) = \Omega_{a_o}[1 + (0.48 + 1.574w_j - 0.176w_j^2)(1 - T_{rj}^{1/2})]^2$$

$$\Omega_b(T,j) = \Omega_{b_o}$$

- For Zudkevitch-Joffe:

$$\Omega_a(T,j) = \Omega_{a_o} F_{aj}(T) T_{rj}^{-1/2}$$

$$\Omega_b(T,j) = \Omega_{b_o} F_{bj}(T)$$

- For Peng-Robinson:

$$\Omega_a(T,j) = \Omega_{a_o}[1 + (0.37464 + 1.54226w_j - 0.26992w_j^2)(1 - T_{rj}^{1/2})]^2$$

$$\Omega_b(T,j) = \Omega_{b_o}$$

As $T_r = T/T_c$, the SRK and PR forms for Ω_a and Ω_b may be expanded as a polynomial in $1/T^{1/2}$, for example,

$$\Omega_a(T,j) = A + B \cdot T^{1/2} + CT \qquad \text{[EQ 18.11]}$$

This expansion is used if TEMPVD is specified, so that each cell and well has its own temperature.

The normal PR form can be optionally modified for a large acentric factor, using the factor $(0.379642 + 1.48503w_j - 0.164423w_j^2 + 0.016666w_j^3)$ rather than $(0.37464 + 1.53226w_j - 0.2699w_j^2)$ for $w_j > 0.49$. This correction is invoked by use of the PRCORR keyword.

Ω_{a_0} and Ω_{b_0} are constants depending upon the equation of state, as shown in Table 18.2.

Table 18.2 The dependence of Ω_{a_0} and Ω_{b_0} on the equation of state

Equation of state	Ω_{a_0}	Ω_{b_0}
RK, SRK, ZJ	0.4274802	0.08664035
PR	0.457235529	0.07796074

In the program, these default values may be over-written if required by the use of the OMEGAA and OMEGAB keywords.

The Zudkevitch-Joffe equation contains additional temperature factors, denoted $F_{aj}(T)$ and $F_{bj}(T)$ multiplying the usual RK temperature dependence. These are adjusted to match the pure-component fugacity values along the vapor pressure line, and to observe the correct component liquid density.

In the program the required variation of component saturation pressure and liquid density with temperature are obtained using correlations of Reidel, and Gunn and Yamada. These correlations require the input of the normal boiling point and the reference liquid density at a specified reference temperature.

Flash calculations

For a thermodynamic system to be in equilibrium, it is further required that the fugacities in the liquid and vapor phases must be equal for each component:

$$f_{iL} = f_{iV} \quad \text{[EQ 18.12]}$$

As described above, the fugacities are functions of temperature, pressure and composition:

$$f_i = f_i(T, p, x_i) \quad \text{[EQ 18.13]}$$

The fugacities can be calculated directly from an equation of state.

Equilibrium constants (otherwise known as K-values) for each component can be defined as:

$$K_i = \frac{y_i}{x_i} \quad \text{[EQ 18.14]}$$

The mole fractions of each component in the liquid and vapor phases are given as:

$$x_i = \frac{z_i}{1 + (K_i - 1)V} \quad \text{[EQ 18.15]}$$

$$y_i = \frac{K_i z_i}{1 + (K_i - 1)V} \quad \text{[EQ 18.16]}$$

If K-values are used to specify inter-phase equilibria, these two equations are used to specify the liquid and vapor compositions directly, with K-values obtained by table look-up as functions of pressure.

Viscosity evaluation

In compositional mode the phase viscosity values are obtained using either the Lorentz-Bray-Clark method or the Pedersen et al. method.

The Lorentz-Bray-Clark method

By default the Lorentz-Bray-Clark method is used [Ref. 12]. The viscosity μ for each phase is given by

$$[(\mu - \mu^{\circ}) \cdot \xi + 0.0001]^{1/4} = \sum_{i=1}^{5} a_i \cdot b_r^{\,i-1} \qquad \text{[EQ 18.17]}$$

μ°, and ξ are functions of the composition x_i, the component molecular weights, critical pressures and critical temperatures.

The coefficients a_i are set equal to the values given in. [Ref. 12]. These values can be modified by using the `LBCCOEF` keyword.

The reduced molar densities b_r are given by

$$b_r = \frac{b_p}{b_c} \qquad \text{[EQ 18.18]}$$

where the phase volumes b_p are calculated from the pressure and compressibility (and volume shift if used), and the critical molar density b_c for each phase is given by

$$\frac{1}{b_c} = V_c = \sum x_i \cdot V_{c,i} \qquad \text{[EQ 18.19]}$$

The critical molar volumes for each component $V_{c,i}$ can be specified directly by either the `VCRIT` or `VCRITVIS` keywords, or can be calculated from critical z-factors entered by the `ZCRIT` or `ZCRITVIS` keywords.

The modified Lorentz-Bray-Clark method

A modified form of the Lorentz-Bray-Clark method can also be used:

$$[(\mu - \mu^{\circ}) \cdot \xi + 0.0001] = \left(\sum_{i=1}^{5} a_i \cdot b_r^{\,i-1} \right)^{4} \cdot \exp\left(\frac{b_r}{b_{max,r} - b_r} \right) \qquad \text{[EQ 18.20]}$$

This modified equation is intended to improve the modeling of the behavior of heavy oils. It was requested by a client and has not been published.

In this equation the viscosity increases exponentially as the reduced molar density b_r approaches a maximum $b_{max,r}$. For small values of b_r, $b_r \ll b_{max,r}$ the modified Lorentz-Bray-Clark has the same form as the original [EQ 18.17]. The maximum reduced molar density can be expressed in terms of a maximum molar density b_{max}

$$b_{max,r} = \frac{b_{max}}{b_c}$$

which in turn can be expressed in terms of minimum component volumes $V_{min,i}$

$$\frac{1}{b_{max}} = V_{min} = \sum x_i \cdot V_{min,i} \qquad \text{[EQ 18.21]}$$

The minimum component volumes $V_{min,i}$ are entered using keyword VCOMPACT. The modified Lorentz-Bray-Clark method will be used instead of the Lorentz-Bray-Clark method if this keyword is found.

When entering $V_{min,i}$ data it is important to ensure that the actual molar density never exceeds the maximum, that is

$$b_p < b_{max} \qquad \text{[EQ 18.22]}$$

If the molar densities do exceed this value then a large constant value is used for the viscosity.

A simplified version of the modified Lorentz-Bray-Clark method can be used if the minimum component volumes $V_{min,i}$ are set proportional to the critical component volumes

$$V_{min,i} = \frac{1}{b_0} \cdot V_{c,i} \qquad \text{[EQ 18.23]}$$

In this case the maximum reduced molar density in [EQ 18.20] will be a constant:

$$b_{max,r} = b_0 \qquad \text{[EQ 18.24]}$$

Example

In this example of the simplified version of the modified Lorentz-Bray-Clark method, we wish to set the reduced molar density $b_{max,r}$ to a constant value of 4.776. Given values of VCRIT for each of the three components, the three VCOMPACT values are obtained by dividing each VCRIT value by 4.776.

```
VCRIT
 99.20  93.90 148.30 /

VCOMPACT
 20.77  19.66  31.05 /
```

The Pedersen method

If the Pedersen at al. viscosity calculation is required instead of the default method, then the PEDERSEN keyword should be added to the PROPS section. Refer to [Ref. 81] for more information.

Three parameter equations of state

The traditional weakness of the two-parameter equations of state (EoS), such as the Peng-Robinson, Redlich-Kwong, etc. is their poor prediction of liquid properties, especially liquid densities and saturations.

The critical compressibilities predicted by the Van der Waals, Redlich-Kwong and Peng-Robinson equations of state are $Z_c^{VdW} = 0.375$, $Z_c^{RK} = 0.333$ and $Z_c^{pr} = 0.307$. But hydrocarbons are known to have $Z_c < 0.29$.

Peneloux *et al.* proposed a molar volume correction, referred to as volume shift, which adds a third parameter to the EoS, and, in turn, greatly improves liquid property estimation.

For a mixture of N components, the phase molar volume $V_{\mathrm{mol},p}$ is given by:

$$V_{\mathrm{mol},p} = V_{\mathrm{mol},p}^{EoS} - \sum_{i=1}^{N} z_i c_i \qquad \text{[EQ 18.25]}$$

where

$p = (\text{liquid, vapor})$ represents the phase of the system

$V_{\mathrm{mol},p}^{EoS}$ is the molar volume of the phase predicted by the traditional 2-parameter EoS

$z_i = (x_i, y_i)$ are the liquid and vapor mole compositions

c_i constitute a set of volume corrections.

The component corrections are usually related to set of dimensionless shift parameters s_i, by:

$$s_i = \frac{c_i}{b_i} \qquad \text{[EQ 18.26]}$$

where:

$$b_i = \frac{\Omega_{b,i} R T_{ci}}{p_{ci}} \qquad \text{[EQ 18.27]}$$

The shift parameters are entered in the dimensionless form using the keyword `SSHIFT`.

Use of the equation of state for hydrocarbon mixtures

A typical oil is composed of many millions of components. The equation of state has one value of A and one value of B to model the phase behavior of this mixture.

For example the PR EoS is as follows:

$$P = \frac{RT}{V-B} - \frac{A}{V(V+B)+B(V-B)} \qquad \text{[EQ 18.28]}$$

In a typical reservoir simulation model, 4-8 components and pseudo-components are used to define the mixture. Each component has a value of T_c, P_c, V_c or Z_c, ω, Ω_a, Ω_b and binary interaction coefficients, δ_{ij} (that increase or decrease the forces between component pairs), that must be specified in the fluid characterization process.

These parameters for pure components, such as Methane or Nitrogen, and usually considered fixed. Thus, you must alter the remaining component parameters so that the resulting calculated phase behavior matches the experimental data. This phase behavior matching process is carried out with the PVT*i* program.

Calculation process

This determines the amount of liquid and vapor present in a grid block and the phase compositions at equilibrium

The ECLIPSE 300 iteration process to converge a solution consists of linear, non-linear (Newton) and flash iterations. The final solution at the end of a time step for each grid block consists of a pressure; water, oil and gas saturations; liquid and vapor molar fractions; and mole fractions of each component in the liquid and vapor phase.

This section describes the iteration process to flash the mixture in each grid block to determine the molar volumes and composition at equilibrium.

After the linear problem converges, we have a new pressure, water and molar densities for each grid block.

A stability test tells us if there are one or two hydrocarbon phases. If there are two hydrocarbon phases, then K-values, K_i, are assigned to each component, i, from the previous iteration, or, if none exist, from Wilson's formula, stated below:

$$K_i = \frac{\exp\left[5.37(1+\omega_i)\left(1-\frac{1}{T_{ri}}\right)\right]}{P_{ri}} \qquad \text{[EQ 18.29]}$$

Then, given z_i and K_i, ECLIPSE 300 solves the Flash Equation for the molar fraction of vapor, V. Details of this Flash Equation are shown overleaf.

$$g(V) = \sum_{i=1}^{n_c} \frac{z_i(K_i-1)}{1+V(K_i-1)} = 0 \qquad \text{[EQ 18.30]}$$

ECLIPSE 300 then solves this equation for the mole fractions of each component in the liquid and vapor phases,

$$x_i = z_i/[1+V(K_i-1)]\,,\ i = 1, 2, \ldots\ldots n_c \qquad \text{[EQ 18.31]}$$

$$y_i = (K_i z_i)/[1 + V(K_i - 1)]\,,\ i = 1, 2, \ldots\ldots n_c \qquad \text{[EQ 18.32]}$$

The compressibility factor is defined as $Z = \frac{PV}{RT}$. As a result, the Peng-Robinson equation of state can be expressed in terms of Z:

$$Z^3 - Z^2(1-b) + Z(a - 3b^2 - 2b) - (ab - b^2 - b^3) = 0 \qquad \text{[EQ 18.33]}$$

where

$$a = \frac{AP}{R^2 T^2} \qquad \text{[EQ 18.34]}$$

$$b = \frac{BP}{RT} \qquad \text{[EQ 18.35]}$$

The solution of the cubic equation gives, in a two phase region, three real roots. The largest root is the compressibility factor of the vapor phase, and the smallest positive root is the compressibility factor of the liquid phase.

With the current information, ECLIPSE 300 calculates the fugacity of each component in the liquid and vapor phases.

When a grid block is in equilibrium at a given pressure and temperature, the fugacities of all components must be the same in the liquid and the vapor. In other words,

$f_{Li} = f_{Vi}$.

A convergence criteria is then used to determine if a grid block is in equilibrium (if ε is very small),

$$\sum_{i=1}^{n_c} \left(\frac{f_{iL}}{f_{iV}} - 1 \right)^2 < \varepsilon\,. \qquad \text{[EQ 18.36]}$$

If equilibrium is not achieved, then a new estimate for the K-values are made and the iteration process continues with a new solution of the Flash Equation, etc. Using successive substitution, the new estimate of the K-value is:

$$K_i^{NEW} = K_i^{OLD} \cdot \left[\frac{f_{iL}}{f_{iV}} \right]. \qquad \text{[EQ 18.37]}$$

After equilibrium is achieved, ECLIPSE 300 begins the next non-linear iteration.

Calculation of phase states

A cell in a compositional run may be water-filled, or in a one or two phase hydrocarbon state. The one phase states may possess a saturation pressure, usually lower than the current pressure, at which a second hydrocarbon phase appears. For temperatures above the cricondotherm however, such a pressure does not exist, and the cell composition leads to a single hydrocarbon phase at all pressures.

The appearance of a two phase state is detected by application of the Michelsen stability test [Ref. 13]. This is carried out anew at each time step, and does not involve assumptions concerning the state of the cell carried over from previous steps.

By default ECLIPSE 300 uses the stability test for all cells, avoiding saturation pressure tracking altogether.

The Michelsen stability test involves the minimization of a function G* as a function of a trial phase composition Y, starting from both liquid and vapor-like initial estimates. If a non-trivial minimum can be found with $G^* < 0$ the hydrocarbon mixture is unstable, and a flash calculation may be performed to establish the phase split.

It is possible to cast the Michelsen stability test in a residual form and use a Newtonian method, but this is not entirely suitable, as:

- There are generally two minima to G*, and the Newton method can converge towards a local maximum as easily as a minimum.
- Newton's method involves order N^3 operations per iteration, due to the matrix operations involved. As a number of iterations are required from each initial Y, this becomes expensive.

A better method, which also yields quadratic convergence near the solution, is the BFGS iteration [Ref. 14]. This is a minimization method, using an approximate Hessian constrained to be symmetric and positive definite. In the early stages of the iteration an accelerated successive substitution method is effective, using a two-point GDEM technique [Ref. 15].

Fast flash option

In some cases a rigorous stability test may not be necessary. In the Fast Flash option, the simulator checks for single phase cells in the middle of single phase regions. The stability calculation is not performed for these cells. Instead, the state is assumed to remain the same, as long as the state of the neighboring cells is unchanged. This can lead to a considerable improvement in simulator speed - depending on the proportion of single phase cells in the simulation.

The fast flash option can be invoked using the `SKIPSTAB` keyword. It can be switched off and on using item 60 of the `OPTIONS3` keyword. Item 109 of the `DEBUG3` keyword can be used to gauge the usefulness of the method for a particular data set, and its accuracy.

Equations of State Regions

Chapter 19

Introduction

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

It is possible to set up more than one equation of state description, and associate these with sets of cells using the EOSNUM keyword in the REGIONS section. The most common application of this is a study containing two or more disconnected reservoirs, coupled through the production system.

Define NR as the number of reservoir equations of state, and NS as the number of surface equations of state. These are specified in TABDIMS, and both default to 1. If a value is entered for NR, this becomes the default for NS. There is a limit of 10 on both NR and NS.

Reservoir equation of state

For each reservoir equation of state (EoS), the program expects information corresponding to the following keywords:

Table 19.1 Reservoir EOS keywords

Keyword	Alternative	Notes
RTEMP	or TEMPVD	
EOS		
TCRIT		
PCRIT		
VCRIT	or ZCRIT	
VCRITVIS	or ZCRITVIS	(optional - default VCRIT or ZCRIT)
MW		
ACF		
OMEGAA		
OMEGAB		
SSHIFT		(optional - default 0)
TBOIL		(ZJ equation only)
TREF		(ZJ equation or Thermal)
DREF	or GREF	(ZJ equation or Thermal
CREF		(Thermal only)
PREF		(Thermal only)
THERMEX1		(Thermal only)
PARACHOR		(optional - miscible only)
BIC		
OILVISCC		(Thermal only)
OILVISCF		(Thermal only)
GASVISCF		(Thermal only)
HEATVAP		(Thermal only)
HEATVAPE		(Thermal only)
SPECHA		(Thermal only)
SPECHG		(Thermal only)
KVCR		(Thermal only)
ZFACTOR		(Thermal only)

This is provided by entering the required data NR times. In the case of NR=2, for example:

```
TCRIT
548.4 227.16 343.0 549.7 665.64 806.54 838.1 1058.0 1291.8 /
548.4 227.16 343.0 549.7 665.64 808.3  860.6 1107.5 1402.0 /
```

The same number of components must exist in all equation of state regions, and they have the same names. It would be most common to alter the properties of the heavy components, as for example, when performing regressions to a number of reservoir fluids.

Having defined the EoS properties to be associated with the EoS regions, these are associated with cells using EOSNUM, which acts like SATNUM. For example, in the case of a $35 \times 14 \times 12$ study, with the top 4 layers using EoS1 and the bottom 8 using EoS2:

```
EOSNUM
490*1 490*1 490*1 490*1
490*2 490*2 490*2 490*2 490*2 490*2 490*2 490*2 /
```

Alternatively, the EQUALS keyword may be used in the usual manner:

```
EQUALS
EOSNUM 1 4* 1  4 /
EOSNUM 2 4* 5 12 /
/
```

If cells with different EoS region numbers are in the same equilibration region, the initial state is not generally in hydrostatic equilibrium, as the EoS determines the phase densities. The code issues a warning if this occurs.

Surface equation of state

Turning to the surface facilities, the default value for NS is NR, that is, the same number of surface equations of state as reservoir equations of state.

The properties of each surface EoS default to those of the corresponding reservoir equation of state, but any or all of these may be over-written using the following keywords:

Table 19.2 Surface EOS keywords

Keyword	Alternative	Notes
EOSS		
TCRITS		
PCRITS		
VCRITS	or ZCRITS	
EOSS		
MWS		
ACFS		
OMEGAAS		
OMEGABS		
SSHIFTS		(optional - default 0)
TBOILS		(ZJ equation only)
TREFS		(ZJ equation or Thermal)
DREFS	or GREFS	(ZJ equation or Thermal)
PREFS		(Thermal only)
BICS		
RTEMP		parachors and viscosity values are not employed in surface calculations.

Items in these keywords default to the reservoir equation of state values. For example, to modify just the last two TCRIT values, with NS=2, it is possible to enter:

```
TCRITS
7* 1078.0 1345.0 /
7* 1067.0 1356.0 /
```

If NR is set to a value less than NS, then the last (NS-NR) surface EoS values are taken from the last (NR^{th}) EoS.

When performing separator calculations, a choice of surface equation of state must be made. By default, the flash calculations for a well use the EoS number corresponding to the cells in which that well is completed. For example, a well completed in cells in reservoir EoS region 2, uses surface EoS2. If a well is completed in cells in more than one reservoir EoS region, a warning is issued, and the first value encountered used.

It is possible to over-ride this choice of surface EoS using the keywords FIELDSEP and SEPCOND. The eighth argument of FIELDSEP and the tenth argument of SEPCOND may be used to specify a non-default EoS number to be used in separator fluid-in-place and surface rate calculations, respectively.

By default, FIELDSEP obtains separator volumes in place for the whole field and for fluid-in-place regions using the equation of state specified for the cell, as specified with EOSNUM.

File Handling in ECLIPSE

Chapter 20

Introduction

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

A number of files may be produced by ECLIPSE for each simulation run. These are:

Table 20.1 Files produced by ECLIPSE for a simulation run

Simulator	File	Purpose	Contents
	PRINT file	Main printer output	(RPT... keywords)
	DEBUG file	Debug output	(DEBUG keyword)
	MSG file	Message output (XML format)	(MSGFILE keyword)
	GRID file	Grid geometry file	(NOGGF and GRIDFILE keyword)
	SAVE file	Non-recurrent data for fast restarts	(SAVE keyword)
	INITIAL file	Initialization file for GRAF/Pseudo	(PSEUDOS keyword and INIT keyword)
	INSPEC file	Initial index file	(INSPEC keyword in RUNSPEC section)
	RESTART file	Restart runs / grid graphics file	(RPTSCHED keyword)
	SMSPEC file	Specifications for summary file	(SUMMARY section)
	SUMMARY file	Line graphics file	(SUMMARY section)
	RSM file	Run summary file	(SEPARATE/LOTUS keywords)
	RFT file	RFT and PLT graphics file	(WRFTPLT keyword)

Table 20.1 Files produced by ECLIPSE for a simulation run

Simulator	File	Purpose	Contents
	RSSPEC file	Restart index file	(RSSPEC keyword in RUNSPEC section)
	EDESC file	Report step communication file for VISAGE external geomechanics workflow.	
	VDESC file	Stress step communication file for VISAGE external geomechanics workflow.	
ECLIPSE 100 Gradient Option	HMD file	Gradients file for SimOpt	(HMDIMS in RUNSPEC section)
ECLIPSE 100 Gradient Option	GRFT file	RFT gradients for SimOpt	(GRADRFT in SCHEDULE section)

The production of all of these files, except for the main printer output and VDESC files, may be controlled by the user with the keywords indicated.

The EDESC and VDESC files are produced by runs which use the VISAGE external geomechanics workflow. This workflow is activated by the VISAGE keyword in RUNSPEC section. These small formatted files contain information on either all report dates (in the case of EDESC), or just the report dates at which stress step output has been generated (in the case of VDESC). It is intended to be processed by the VISAGE Coupler application which controls the ECLIPSE-VISAGE interaction.

The contents, and hence the size, of the printer output file are controlled using the RPTGRID, RPTPROPS, RPTREGS, RPTSCHED, RPTSMRY, RPTSOL, OUTSOL and RPTRST keywords.

In addition, you can choose whether input and output files are to be formatted or unformatted, (see the RUNSPEC section keywords FMTIN and FMTOUT), and whether RESTART and SUMMARY files are to be unified (see the RUNSPEC section keywords UNIFIN and UNIFOUT). You should be aware of which types of file it is appropriate to use on your system. Some guidelines are given here.

Formatted files are standard 80 character card image files, which may be straightforwardly transferred between different computers. All the files generated by the ECLIPSE programs can be converted from formatted to unformatted files or vice-versa using the utility program Convert.

In general, unformatted non-unified files should be used, giving efficient use of disk space and allowing you to examine output as the simulation progresses. However, it may be necessary to use formatted files if the simulation is performed on one type of computer but restarts, graphics or pseudos are to be performed on another type of computer. Unified files should be used if the number of files being produced is a problem. Unified files also reduce the size of your catalog considerably (but not the amount of disk space used).

When reading an input file into ECLIPSE following any of the keywords INCLUDE, IMPORT, RESTART or GDFILE, it is possible to set up aliases for pathname roots in order to facilitate transfer between different working directories and to avoid unnecessarily long pathnames.

ECLIPSE 100

A set of aliases for pathnames, each beginning with the character $, can be defined with the RUNSPEC section keyword PATHS.

File internal format

All ECLIPSE files consist of a series of data blocks each headed by a descriptor record. The block descriptor record comprises

- An 8-character keyword which identifies the data in the block.
- An integer (4 byte) containing the number of elements NEL in the block.
- A 4-character field specifying the type of the data in the block.

  Permitted values are

  ‘`INTE`’ for standard (4 byte) integers

  ‘`REAL`’ for single precision (4 byte) floating point reals

  ‘`LOGI`’ for standard (4 byte) logicals

  ‘`DOUB`’ for double precision (8 byte) floating point reals

  ‘`CHAR`’ for characters

Note Note that if the data type is ‘`CHAR`’, then NEL is the total number of CHARACTER*8 elements.

The contents of the block follows the descriptor (on a new record). This may be formatted or unformatted. If formatted, the form of output used may be read back by FORTRAN list input. If unformatted, each data block must be read back in the same block structure as it was written out.

The header/record structure enables files to be converted from formatted to unformatted form, or scanned for a specific item. For example, a `RESTART` file contains a record header with an identifier of ‘`PRESSURE`‘, type ‘`REAL`’ and with the number of elements set to the number of active cells in the study.

In the formatted case the actual formats used for the data blocks are

Integers 6(1X,I11)

Reals 4(1X,E16.8)

Double Precisions 3(1X,D22.14)

Logicals 25(1X,L2)

Character strings 7(1X,A1,A8,A1)

:

The header line is written using a format (1X, A1, A8, A1, 1X, I11, 1X, A1, A4, A1). The A1 fields are used to output quotes so that the data may be read using list input.

```
'LOGIHEAD'             20 'LOGI'
 F  F  F  T  F  T  F  F  T  F  F  F  F  F  F  F  F  F  F  F
'ZGRP    '           1 'CHAR'
'G       '
'IWEL    '          72 'INTE'
        1          1          3           3           1           1
        3          2          2           0           1           0
        1          2          0           2           0        -100
        0          0          0           0           1           1
        0          0          0           0           0           0
        0          7          0           0           0           0
        5          1          1           1           1           1
        1          4          4           0           1           0
        1          2          0           1           0        -100
        0          0          0           0           1           1
       -1          0          0           0           0           0
        0          7          0           0           0           0
'PRESSURE'         15 'REAL'
   .39684106+004    .39684106+004    .39684106+004    .39684106+004
   .39684106+004    .39872349+004    .39872349+004    .39872349+004
   .39872349+004    .39872349+004    .40060627+004    .40060627+004
   .40060627+004    .40060627+004    .40060627+004
```

For further information on formats of specific files, please contact our Support office.

GRAF files

Files written by ECLIPSE to be accessed by the GRAF program comprise:

- The SUMMARY and SPEC files

  These are used to produce line graphs. Data is written to these files at every simulator time step. Items to be written in this way are selected by keywords in the SUMMARY section of the input dataset.

- The GRID file

  The GRID file, containing the locations of cell corner points. This file is always produced by ECLIPSE, unless a NOGGF keyword is placed in the GRID section to suppress it. The grid file contains information for plotting geometry displays of cell positions; it is also required for plotting solution data.

- The solution or RESTART file

  This file contains solution data (pressures, saturations etc.) for all the cells at each report step.

ECLIPSE 100 The file content and report frequency may be controlled with keyword RPTRST.

ECLIPSE 300 The solutions to be written must be chosen using the RPTRST or OUTSOL keyword in the SOLUTION or SCHEDULE sections; otherwise the file is empty.

Flux Boundary Conditions

Chapter 21

Introduction

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL

Flux boundary conditions enable runs to be performed on a small section of a field using boundary conditions established from a full field run. Flows across the boundary of the reduced field are written to a `FLUX` file at each mini time-step of the full field run.

The `FLUX` file is read during the reduced field run to generate the appropriate boundary conditions. Time steps of the reduced field run may be quite different from those of the full field run.

A small amount of editing is required to convert a full field run that generates a `FLUX` file into a run, which runs one or two flux regions with the appropriate boundary flows being read from the `FLUX` file.

The following sections describe the use of the Flux option in some simple cases.

Using flux boundary conditions

Single flux region

Producing the FLUX file

To produce the FLUX file:

ECLIPSE 100

1 Set NTFREG, the number of flux regions, to 1 in item 4 of keyword REGDIMS in the RUNSPEC section.

```
REGDIMS
-- NTFIP NMFIPR NRFREG NTFREG MHISTM NMHSTR
    1*     1*     1*     1      1*     1*
/
```

2 Enter the keyword DUMPFLUX in the GRID section, to indicate that this is the full field run and a flux file should be written.

3 Use the FLUXNUM keyword in the GRID section to establish the cells of the reduced run. For example, to establish flux boundary conditions for a reduced 2*2*2 run from a 4*4*3 full field run use:

```
DUMPFLUX
FLUXNUM
 1 1 0 0
 1 1 0 0
 0 0 0 0
 0 0 0 0
 1 1 0 0
 1 1 0 0
 0 0 0 0
 0 0 0 0
 0 0 0 0
 0 0 0 0
 0 0 0 0
 0 0 0 0
/
```

4 The flux file is called ROOT.FLUX (unformatted) or ROOT.FFLUX (formatted), where 'ROOT' is the root name of the run. The file will be formatted if keyword FMTOUT is specified in the RUNSPEC section.

Using the FLUX file in reduced runs

The reduced run is based on the original dataset used for the full field (DUMPFLUX) run.

The reduced run should use the same grid dimensions as the full field run and employ the same flux region definition.

You nominate which flux regions are active in the reduced run. ECLIPSE then only simulates the reduced run within this active flux region. The support to this reduced flux region from the full field model is provided by a set of flows between flux regions which are supplied from the FLUX file dumped by the full field run.

To use the FLUX file in a reduced run:

ECLIPSE 100

1 Set NTFREG to 1 in item 4 of keyword REGDIMS in the RUNSPEC section.

2 Use the USEFLUX and FLUXNUM keywords in the GRID section, for example:

```
USEFLUX
 'ROOT' /
FLUXNUM
 1 1 0 0
 1 1 0 0
 0 0 0 0
 0 0 0 0
 1 1 0 0
 1 1 0 0
 0 0 0 0
 0 0 0 0
 0 0 0 0
 0 0 0 0
 0 0 0 0
 0 0 0 0
/
```

3 If the FLUX file is formatted specify keyword FMTIN in the RUNSPEC section

ECLIPSE 100

4 If user-specified non-neighbor connections are specified in the GRID section, the NNC keyword must be placed after the USEFLUX and FLUXNUM keywords.

5 The FLUXNUM array can be output to the PRINT file using the FLUXNUM mnemonic in the RPTGRID keyword.

In both the full field and reduced runs, the FLUXNUM keyword should be entered after the DUMPFLUX or USEFLUX keyword.

Multiple flux regions

Producing the FLUX file

Several flux regions may be defined and all relevant flows stored on the FLUX file. For example, 9 separate regions may be defined by setting NTFREG = 9 in the RUNSPEC section keyword REGDIMS and using keywords DUMPFLUX and FLUXNUM in the GRID section.

```
DUMPFLUX
FLUXNUM
 1 1 2 2
 1 1 2 2
 3 3 3 3
 4 4 4 4
 5 5 5 5
 6 7 8 8
 6 7 8 8
 6 7 8 8
 9 9 9 9
 9 9 9 9
 9 9 9 9
 9 9 9 9
/
```

Using the FLUX file in reduced runs

To perform a reduced run on a sector of the field defined by the 3 regions 2, 3 and 5:

ECLIPSE 100

1. Set NTFREG = 3 in item 4 of the RUNSPEC section keyword REGDIMS.
2. Use the USEFLUX, FLUXNUM and FLUXREG keywords in the GRID section. The FLUXREG keyword tells ECLIPSE which flux regions are active in the reduced run. The FLUXNUM keyword must be exactly the same as in the full field run. The FLUXREG and FLUXNUM keywords should follow the USEFLUX keyword.

```
USEFLUX
 'ROOT' /
FLUXREG
 2 3 5 /
FLUXNUM
 1 1 2 2
 1 1 2 2
 3 3 3 3
 4 4 4 4
 5 5 5 5
 6 7 8 8
 6 7 8 8
 6 7 8 8
 9 9 9 9
 9 9 9 9
 9 9 9 9
 9 9 9 9  /
```

Treatment of wells

In the full field run, the oil, water, gas and reservoir fluid volume rates of each well are written to the FLUX file at each time step in ECLIPSE 100. The oil, water, gas, voidage, wet gas and component well rates are written to the flux file at each time step in ECLIPSE 300.

In the reduced run, the wells are sorted into two categories: those completed inside the flux boundary and those completed outside it. An error is flagged if a well has completions on both sides of the flux boundary. The wells outside the boundary are essentially ignored in the simulation, except that their flow rates are read from the `FLUX` file and added to the group and field totals. These wells are omitted from the well reports.

Because the wells outside the flux boundary do not exist as part of the simulation model, the ways in which they can influence the behavior of the wells inside the boundary are very limited. Essentially, if a group contains wells on both sides of the boundary (the FIELD will probably do so), then the only group control or constraint that can be applied to it is control of the group flow rate. The wells outside the boundary are treated as unavailable for group control, and their flows are read from the flux file and summed. The wells inside the boundary then make up the rest of the group's target rate. Other group constraints (for example economic limits, rate limits resulting in workovers, workover rig availability, artificial lift capacity, limits on the number of open wells) should not be applied to groups containing wells on both sides of the boundary, as events happening outside the flux boundary are not taken into account. If the drilling queue contains wells both inside and outside the flux boundary, drilling events outside the boundary are only visible by the additional contribution of the new wells in the flux file at the end of the corresponding time step of the full-field run. If the group falls short of its production target before the full contribution of the newly-drilled wells outside the boundary is seen, wells inside the flux boundary may be drilled prematurely in the `USEFLUX` run. It is best therefore to avoid using a drilling queue in these circumstances.

Drainage regions

ECLIPSE 100

Drainage regions are areas of the field that are drained separately from each other. These are written to the debug file if option 25 of the `OPTIONS` keyword is set > 0. Since flows are minimal between different drainage regions, this provides a convenient method of splitting the field into separate flux regions. If option 25 is set > 1, then location of the main production well within each drainage region is identified by a negative region number. `FLUXNUM` treats negative numbers as if they were positive.

Aquifers

Any aquifer data in the reduced run is processed in the usual way. Numerical aquifers declared outside the `USEFLUX` region are ignored in the reduced run (the aquifer support is accounted for by the flows across the flux boundary generated by the `DUMPFLUX` run). When determining aquifer connections in the reduced run, no extra connections are generated across the flux boundary when the flux boundary is enclosed within the box defined in `AQUCON`, `AQUFET` or `AQUANCON`. Hence no new aquifer connections are made in the reduced run that were not in the base run, and support from aquifer connections outside the reduced flux region is accounted for by fluxes in the reduced run.

Restart runs

The reduced run can use flux information from a full field run that consists of a sequence of restarts. The USEFLUX keyword in the reduced run should specify the root name of the last ECLIPSE run in the full field restart sequence. The FLUX files from all the restarts must exist and either be all formatted or all unformatted. Keyword FMTIN should be specified in the RUNSPEC section if the files are formatted.

Figure 21.1 A sequence of restarts that together make a full field run

The reduced run SMALL.DATA includes the USEFLUX keyword as follows:

```
USEFLUX
 'RUN3'
/
```

This run uses files RUN1.FLUX, RUN2.FLUX and RUN3.FLUX to read flux information for the time sequence represented by *'s. The number of restarts in the sequence is limited to 20.

It is possible to restart a reduced run from the corresponding full field run by specifying the full field root name in the RESTART keyword in the usual manner.

Note The periods covered by the restarts in the sequence should not overlap.

ECLIPSE 300

Note When doing a reduced run from a restart, the GRID file from the restart run must also be present in order for the simulation to initialize correctly.

Pressure boundary conditions (ECLIPSE 100)

Instead of using the flows of each phase from the full field run as boundary conditions on the reduced run, an alternative treatment using pressures and saturations is available (see the FLUXTYPE keyword). The pressures, saturations, R_s and R_v values in the 'halo' cells surrounding each flux region are stored in the FLUX file from the full field run. In the reduced run, ECLIPSE 100 calculates the flows across the flux boundary using the pressures and saturations in the halo cells surrounding the reduced flux region.

When the total flux across the boundary of the reduced region does not alter appreciably, the default flow boundary condition is adequate. This is the case, for example, when the flux option is used to examine the effect of changing the well data in the reduced region when the flux boundary is wholly an inflow boundary or wholly an outflow boundary and the distribution of fluids on the flux boundary does not alter appreciably in the reduced run. When more severe

changes are made to the flow regime and fluid distribution, the pressure boundary condition should be used to allow the phases to re-distribute adjacent to the boundary and allow the switch from inflow/outflow in the full field run to outflow/inflow in the reduced run. In this case, ECLIPSE 100 works out whether the flow is into or out of the reduced region by upwinding, which, in general, improves stability.

No-flow boundary conditions (ECLIPSE 100)

As an alternative to defined-flux or defined-pressure boundary conditions, a USEFLUX run may be performed with no-flow boundary conditions. The USENOFLO keyword should be entered in the GRID section. In this case it is not necessary to provide a FLUX file. This option provides a simple way of simulating just a section of the reservoir in isolation (provided that no-flow boundary conditions are a reasonable approximation) without having to perform a DUMPFLUX run.

Local grid refinement

ECLIPSE 100 Within ECLIPSE 100, refined grids must not be placed at flux boundaries in a DUMPFLUX run.

ECLIPSE 300 There is no restriction on the position of LGRs relative to flux boundaries in a DUMPFLUX run in ECLIPSE 300.

In a reduced flux USEFLUX run, local grids cannot straddle or be placed at boundaries of the reduced flux region (defined by FLUXREG).

However, they can be placed within flux regions provided that there is at least one cell between the refined cells and the flux boundary. Refined grids outside the reduced field region are ignored in a reduced field run. Thus, as with wells, it is unnecessary to edit out local grid refinements in the inactive region of a reduced field run. It is not possible for individual LGRs within a single LGR amalgamation to belong to different flux regions.

Reading ECLIPSE 100 Flux Files into ECLIPSE 300

Flux runs are usually run by ECLIPSE 300 and use a FLUX file generated by ECLIPSE 300. It is possible to read an ECLIPSE 100 flux file. An ECLIPSE 100 FLUX file contains flows in terms of stock tank oil and gas volumes, and you must then also enter XSTC and YSTC, the stock tank oil and gas compositions, in order that the program can generate component flows.

It is not possible to use an ECLIPSE 100 flux file to run a thermal reduced data.

Thermal Option

ECLIPSE 300 The flux boundary option can be used for 2 different scenarios:

1. Full field model is a Thermal model, and the reduced model is also a Thermal model
2. Full field model is either an ECLIPSE 300 Blackoil model or a Compositional model, and the reduced run is a Thermal model.

For option 1, a thermal reduced run can be carried out from a thermal base run. In this case, the energy flows across the boundary are written out to the FLUX file in the full field run.

For option 2, a thermal reduced run can be carried out from a compositional or an ECLIPSE 300 blackoil base case. In this case the energy flows across the boundary are calculated in the reduced run.

The conductive term is defined as $C_{ei} = -\sum_n \Psi_{ni} \cdot \Lambda \cdot (T_i - T_n)$,

where

Ψ_{ni} is the heat transmissibility between cells n and i. This term is calculated as follows:

$\Psi_{ni} = \alpha \times \Upsilon_{ni}$ where Υ_{ni} is the transmissibility between cells n and i taken from the base run, and α is the heat transmissibility factor, defined using the `THFLXTF` keyword.

The Λ term is the saturation dependent transmissibility multiplier of the cell inside the active flux region.

The convective term is defined as $F_{ei} = \sum_n \Upsilon_{ni} \sum_p M_p^e \cdot dP_{pni}$,

where

M_p^e is the energy generalized mobility, and is calculated as follows:

$$M_p^e = H_p \cdot k_{rp}(S_p) \cdot \frac{b_p}{\mu_p}.$$

Referring to the above form, the $k_{rp}(S_p) \cdot \frac{b_p}{\mu_p}$ part is taken from the base run, and the phase enthalpy H_p is calculated in the cell inside the active flux region during the reduced run.

To enable a thermal reduced run from a non-thermal base run, the `FLUXOPTS` keyword should be used in the base run and set to YES. This will allow ECLIPSE 300 to provide all the essential information to perform the reduced thermal simulation.

Current limitations

1. `DUMPFLUX` and `USEFLUX` cannot be used simultaneously in the same run.

ECLIPSE 100

2. When using the flow boundary conditions, flux files contain only oil, water, gas and component flows (or pressures, saturations, R_s and R_v values if pressure boundary conditions are selected in ECLIPSE 100). Thus, when using the flow boundary conditions, the Flux Boundary option cannot be used with polymers, tracers, API and brine tracking, the Solvent model and other special features involving additional conservation equations.

ECLIPSE 100

3. The pressure boundary conditions account for solvent, API and brine tracking.

Gas Field Operations Model

4. The Flux Boundary option should not be used in conjunction with automatic DCQ reduction in the Gas Field Operations model.

Parallel Option

5. Using `DUMPFLUX`, the WARP linear solver cannot be used, instead the pre-2002A linear solver is activated.
6. The nine-point schemes (`NINEPOIN` keyword) cannot be used when the flux boundary option is used.

Summary of keywords

RUNSPEC section

The RUNSPEC keywords are;

Table 21.1 RUNSPEC keywords

Keyword	Description
REGDIMS	Item 4 specifies the maximum number of flux regions in the run.

GRID section

The GRID keywords are:

Table 21.2 GRID keywords

Simulator	Keyword	Description
	ADDREG	Adds a constant to the specified array in a flux region.
	COPY	Copies data from one array to another.
	DUMPFLUX	Makes a full field run write a FLUX file.
	EQUALREG	Sets an array to a constant in a flux region.
	FLUXNUM	Identifies extent of each flux region.
	FLUXREG	Identifies which flux regions are active.
ECLIPSE 100 only	FLUXTYPE	Specifies the type of flux boundary condition.
	MULTIREG	Multiplies an array by a constant in a given flux region.
	RPTGRID	Controls on output from the GRID section: FLUXNUM outputs flux region numbers.
	USEFLUX	Specifies the FLUX file used for a reduced run.
ECLIPSE 100 only	USENOFLO	Allows a USEFLUX data set to be run without a FLUX file.
ECLIPSE 300 only	XSTC	Specifies oil composition at standard conditions.
ECLIPSE 300 only	YSTC	Specifies gas composition at standard conditions.
ECLIPSE 300 only	THFLXTF	Specifies the heat transmissibility/ flow transmissibility ratio

Options keywords

The Options keywords are:

Table 21.3 Options keywords

Simulator	Keyword	Description
ECLIPSE 100 only	OPTIONS	Activates special program options 25: Drainage regions are written to the DEBUG file. 65: Remove certain error messages about wells connecting in multiple flux regions, or for local grids adjacent to flux boundaries in a DUMPFLUX run.
ECLIPSE 300 only	OPTIONS3	Activates special program options 69: Remove certain error messages about wells connecting in multiple flux regions in a DUMPFLUX run.
ECLIPSE 300 only	FLUXOPTS	YES: Makes a full-field run output necessary information for a thermal reduced run.

The Foam Model

Chapter 22

Introduction

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL

Foam can be used in a number of ways to increase the production from an oil reservoir. The foam acts to decrease the mobility of gas; this effect can be used to slow the break-through of injected gas or to reduce the production of gas cap.

A foam is generated by adding a surfactant to an aqueous phase, and passing a gas through the surfactant to generate a stable dispersion of gas bubbles in the liquid. The foam can be transported with the gas flow into the reservoir. The major beneficial effect of the foam is to reduce the mobility of the gas. Note that in foam flooding the reduction of the interfacial tension (between oil and water) is not a significant effect. The reduction of gas mobility typically depends on a range of factors including pressure and shear rate.

The foam stability has a major effect on the usefulness of foam injection. Typically the foam suffers from adsorption on to the rock matrix, decay over time, and enhanced decay in the presence of water.

The physics of the foam flooding process is in general very complex. For example, when foam bubbles form in a porous medium the bubble size typically fills the pore size of the rock matrix. These bubbles tend not to move until they are compressed (hence reducing their size) by applying a higher pressure. Then in turn more bubbles are generated at the new higher pressure, but with the original bubble size.

The ECLIPSE Foam model does not attempt to model the details of foam generation and collapse. Foam is modeled as tracer which may be transported with either the gas or the water phase with account taken of adsorption on to the rock surface and decay over time.

The simulation model

Foam conservation equation

Although foam is essentially a mixture of gas, water and surfactant, ECLIPSE models it as an effective concentration of surfactant transported in either the gas or the water phase. Hence the foam concentration can be thought of as the surfactant concentration existing in foam form. The distribution of the injected foam is therefore solved by a conservation equation:

$$\frac{d}{dt}\left(\frac{VS_gC_f}{B_rB_g}\right)+\frac{d}{dt}\left(V\rho_rC_f^a\frac{1-\phi}{\phi}\right) = \sum\left[\frac{Tk_{rg}}{B_g\,\mu_g}M_{rf}(\delta P_g-\rho_g gD_z)\right]C_f+Q_gC_f-\lambda(S_w,S_o)VC_f \qquad \text{[EQ 22.1]}$$

as a tracer in the gas phase with decay, or as a tracer in the water phase with decay:

$$\frac{d}{dt}\left(\frac{VS_wC_f}{B_rB_w}\right)+\frac{d}{dt}\left(V\rho_rC_f^a\frac{1-\phi}{\phi}\right) = \sum\left[\frac{Tk_{rw}}{B_w\,\mu_w}(\delta P_w-\rho_w gD_z)\right]C_f+Q_wC_f-\lambda(S_w,S_o)VC_f \qquad \text{[EQ 22.2]}$$

where

C_f denotes the foam concentration

ρ_w, ρ_g denotes the water and gas density respectively

Σ denotes the sum over neighboring cells

C_f^a denotes the foam concentration

μ_w, μ_g denotes the water and gas viscosity respectively

D_z is the cell center depth.

B_r, B_w, B_g is the rock, water and gas formation volume respectively

T is the transmissibility

k_{rw}, k_{rg} is the water and gas relative permeability respectively

S_w, S_g is the water and gas saturation respectively

V is the block pore volume

Q_w, Q_g is the water and gas production rate respectively

P_w, P_g is the water and gas pressure respectively

λ is the rate decay parameter function of oil and water saturation

M_{rf} is the gas mobility reduction factor described below

g is the gravity acceleration

By default, the foam concentrations are updated fully implicitly at the end of a time step after the inter-block phase flows have been determined. If the foam effects on the flow properties are very strong, the foam concentrations can be solved in each Newton iteration so that the tracer concentrations are updated simultaneously with the phase equations by setting the 126th item of the `OPTIONS` keyword to 1. The new foam concentrations are then used to update the flow properties for the subsequent iterations.

The foam is assumed to exist only in either the gas or the water phase and either [EQ 22.1] or [EQ 22.2] only is solved. This is specified using the FOAMOPTS keyword, which is also used to specify which form of gas mobility reduction model is to be used.

Adsorption

The adsorption of foam is assumed to be instantaneous. The isotherm adsorption can be specified as:

- either a look-up table of adsorbed foam as a function of the flowing foam concentration using the FOAMADS keyword
- or by a generic analytical adsorption model using the ADSORP keyword with FOAM as an argument. This model enables the adsorption to depend on rock permeability as well.

The mass adsorbed foam on to the rock is given by:

$$V \cdot \left(\frac{1-\phi}{\phi}\right) \cdot \rho_r \cdot C_f^a \qquad \text{[EQ 22.3]}$$

and the mass density of the rock ρ_r should be supplied using the second argument of FOAMROCK keyword. If desorption is prevented using the first argument of the FOAMROCK keyword, then the adsorbed foam concentration may not decrease with time. If desorption is allowed then each grid block retraces the adsorption isotherm as the foam concentration rises and falls in the cell.

Foam decay

Foam effectiveness will typically reduce over time, even in conditions very favorable to foam stability. This reduction in effectiveness may be accelerated in the presence of water or oil. The reduction in foam effectiveness over time is modeled by foam decay $\lambda(S_w, S_o)$. The half-life of the decay can be a function of both oil and water saturation. If the decay half-life is a function of both oil and water saturation, the foam is assumed to decay with the minimum half-life. The corresponding data is entered using FOAMDCYW and FOAMDCYO.

Gas mobility reduction

The foam modifies the gas mobility by way of a simple multiplier supplied as a function of foam concentration (that is the effective surfactant concentration). The mobility modification is applied explicitly; that is, the modification associated with the conditions at the end of each iteration or time step is applied at the subsequent iteration or time step.

Gas mobility reduction models

Two methods of modeling the gas mobility reduction M_{rf} due to foam are available:

1 Tabular model:

For this model the gas mobility reduction factor is modeled using a set of tables which specify the mobility reduction as tabulated functions of foam concentration (FOAMMOB keyword), pressure (FOAMMOBP keyword) and shear (FOAMMOBS keyword) from which the net mobility reduction factor is calculated.

2 Functional form:

For this model, the gas mobility reduction factor is modeled in terms of a set of functions which represent the individual reduction factors due to surfactant concentration (FOAMFSC keyword), oil saturation (FOAMFSO keyword), water saturation (FOAMFSW keyword) and capillary number (FOAMFCN keyword). These are combined multiplicatively with a reference mobility reduction factor (FOAMFRM keyword) to determine the net mobility reduction factor.

These two methods are selected using the FOAMOPTS keyword. The functional gas mobility reduction model is only available if the transport phase is defined to be water. The formulation of these two models is described below.

Gas mobility reduction tabular model

For the tabular model, the gas mobility reduction factor M_{rf} is expressed in the form:

$$M_{rf} = (1 - M^{cp}{}_{rf})M_v(V_g) + M^{cp}{}_{rf} \qquad \text{[EQ 22.4]}$$

and

$$M^{cp}{}_{rf} = (1 - M_c(C_f))M_p(P) + M_c(C_f) \qquad \text{[EQ 22.5]}$$

where

$M_v(V_g)$ is the mobility reduction modifier due to gas velocity (FOAMMOBS keyword).

$M_c(C_f)$ is the gas mobility reduction factor due to foam concentration (FOAMMOB keyword).

$M_p(P)$ is the mobility reduction modifier due to pressure (FOAMMOBP keyword).

P is the oil phase pressure.

V_g is the gas velocity (see below)

The gas velocity is calculated according to the expression:

$$V_g = B_g\left(\frac{F_g}{\phi \cdot A}\right) \qquad \text{[EQ 22.6]}$$

where

F_g is the gas flow rate in surface units.

B_g is the gas formation volume factor.

ϕ is the average porosity of the two cells.

A is the flow area between the two cells.

The gas mobility reduction modifiers due to pressure and gas velocity are optional and will tend to increase the gas mobility as compared to the mobility reduction due to foam concentration alone.

Gas mobility reduction functional model

For the functional model, the mobility reduction factor M_{rf} is expressed in the form [Ref. 125] and [Ref. 126]:

$$M_{rf} = \frac{1}{1 + (M_r \cdot F_s \cdot F_w \cdot F_o \cdot F_c)} \qquad \text{[EQ 22.7]}$$

where

M_r is the reference mobility reduction factor

F_s is the mobility reduction factor component due to surfactant concentration.

F_w is the mobility reduction factor component due to water saturation

F_o is the mobility reduction factor component due to oil saturation.

F_c is the mobility reduction factor component due to gas velocity (capillary number).

The reference mobility reduction factor M_r is typically in the range of 5 to 100 and corresponds to the normalized resistance to flow for a minimum bubble size in the absence of factors that increase bubble size.

Each mobility reduction factor may optionally be specified or omitted according to whether or not the associated keyword is present. However, at least one of these components must be specified for the mobility reduction factor to be operational.

The mobility reduction factor F_s represents the dependence upon foam (effective surfactant) concentration and is expressed as:

$$F_s = \left(\frac{C_s}{C_s^r}\right)^{e_s} \qquad \text{[EQ 22.8]}$$

where

C_s is the effective surfactant concentration

C_s^r is the reference surfactant concentration.

e_s is an exponent which controls the steepness of the transition about the point $C_s = C_s^r$

The significance of F_s is determined by the reference surfactant concentration above which the presence of surfactant becomes significant in the creation of foam. For low surfactant concentrations $C_s < C_s^r$ (weak foam), the value of F_s will be less than 1 and will tend to 0 as the surfactant concentration decreases to 0. Conversely, for high surfactant concentrations $C_s > C_s^r$ (strong foam), the value of F_s will be greater than 1 and will increase with increasing surfactant concentration. The steepness and curvature of F_s versus C_s is determined by the value of e_s. The rate of change about the point where $C_s = C_s^r$ will increase with increasing e_s. The reference surfactant concentration and the exponent term are specified using the `FOAMFSC` keyword.

The mobility reduction factor F_w represents the dependence upon water saturation and is expressed as:

$$F_w = 0.5 + \frac{\text{atan}[f_w \cdot (S_w - S_w^l)]}{\pi} \qquad \text{[EQ 22.9]}$$

where

S_w is the water saturation.

S_w^l is the limiting water saturation below which the foam ceases to be effective.

f_w is a weighting factor which controls the sharpness in the change in mobility.

The gas mobility increases sharply as water saturation decreases towards this limiting value. The limiting water saturation and the weighting factor are specified using the FOAMFSW keyword.

The mobility reduction factor F_o represents the dependence upon oil saturation and is expressed as:

$$F_o = \left(\frac{S_o^m - S_o}{S_o^m}\right)^{e_o} \qquad \{S_o \le S_o^m\} \qquad \text{[EQ 22.10]}$$

$$F_o = 0.0 \qquad \{S_o > S_o^m\} \qquad \text{[EQ 22.11]}$$

where

S_o is the oil saturation.

S_o^m is the maximum oil saturation above which the foam ceases to be effective.

e_o is an exponent which controls the steepness of the transition about the point $S_o = S_o^m$.

The maximum value of F_o is 1 and occurs when $S_o = 0.0$. For values of $S_o < S_o^m$, the value of F_o will decrease with increasing S_o. The rate of decrease is controlled by the value of the exponent e_o. For values of $S_o \ge S_o^m$, the value of F_o will be 0 and will completely eliminate any gas mobility reduction irrespective of the values of the other multiplicative factors. The maximum oil saturation and the exponent term are specified using the FOAMFSO keyword.

The mobility reduction factor F_c represents the dependence upon capillary number and is expressed as:

$$F_c = \left(\frac{N_c^r}{N_c}\right)^{e_c} \qquad \text{[EQ 22.12]}$$

where

N_c is the capillary number.

N_c^r is the reference capillary number.

e_c is an exponent which controls the steepness of the transition about the point $N_c = N_c^r$.

This component controls the gas mobility for shear-thinning in the low foam quality region. The reference capillary number and the exponent term are specified using the FOAMFCN keyword. The capillary number is a dimensionless parameter which provides a measure of the ratio of viscous to capillary forces and is calculated according to the expression:

$$N_c = \frac{|K \cdot \mathrm{grad}P|}{\sigma_{wg}} C_{\mathrm{unit}} \qquad \text{[EQ 22.13]}$$

where

K is the rock permeability.

P is the potential

σ_{wg} is the gas-water interfacial tension defined by the FOAMFST keyword.

C_{unit} is the conversion factor which depends upon the units used.

The term $|K \cdot \mathrm{grad}P|$ is calculated as:

$$|K \cdot \mathrm{grad}P| = \sqrt{(K_x \cdot \mathrm{grad}P_x)^2 + (K_y \cdot \mathrm{grad}P_y)^2 + (K_z \cdot \mathrm{grad}P_z)^2} \quad \text{[EQ 22.14]}$$

which for a given cell i with neighbor j is calculated in the x direction according to:

$$K_x \cdot \mathrm{grad}P_x = 0.5\left[\left(\frac{K_x}{D_x}\right)_{i-1,i} \cdot (P_i - P_{i-1}) + \left(\frac{K_x}{D_x}\right)_{i,i+1} \cdot (P_{i+1} - P_i)\right] \quad \text{[EQ 22.15]}$$

and similarly for the y and z directions. The K/D value is calculated in an analogous manner to the transmissibility and depends upon how the geometry has been specified.

The gas-water interfacial tension σ_{wg} is tabulated as function of the foam concentration using the `FOAMFST` keyword. The unit conversion C_{unit} constant is defined as shown below:

- `METRIC`

  Pressure bars

  Length m

  Permeability mD

  Surface Tension N/m

  C_{unit} = 9.869234E-11

- `FIELD`

  Pressure psi

  Length ft.

  Permeability mD

  Surface Tension lbs/in

  C_{unit} = 1.274784E-13

- `LAB`

  Pressure atm

  Length cm

  Permeability mD

  Surface Tension dyne/cm

  C_{unit} = 1.0E-5

The mobility reduction factor is only calculated for grid blocks that contain foam with a concentration greater than or equal to the minimum concentration specified by item 3 of the `FOAMFSC` keyword.

Using the foam model

The Foam model is activated by specifying the FOAM keyword in the RUNSPEC section.

It is possible to output the geometric data used to compute the shear thinning effect by using the FOAM mnemonic in the RPTGRID keyword. This outputs a map of the 1/(poro*area) values used to calculate the gas velocity.

The optional FOAMOPTS keyword may be used to select the transport phase for the foam and the method of modeling gas mobility reduction due to foam. If this keyword is not specified, the transport phase will default to gas and the tabular model for gas mobility reduction will be used. If the functional model of gas mobility reduction is specified, the transport phase is constrained to be water.

Gas mobility reduction tabular model

If the tabular method of modeling gas mobility reduction as a function of either or both pressure and shear is selected, there is one obligatory keyword in the PROPS section to describe the reduction in gas mobility as a function of foam concentration. In addition there are five optional keywords which, if present, invoke the adsorption, decay and additional mobility control features of the model.

Table 22.1 Gas mobility keywords

Keyword	Description
FOAMMOB	Tables of the gas mobility reduction factor as a function of foam concentration (Obligatory).
FOAMADS	Foam adsorption data (Optional).
FOAMDCYW	Foam decay data as a function of water saturation (Optional).
FOAMDCYO	Foam decay data as a function of oil saturation (Optional).
FOAMMOBP	Gas mobility reduction as a function of pressure (Optional).
FOAMMOBS	Gas mobility reduction as a function of shear (Optional).

Gas mobility reduction functional model

If the functional method of modeling gas mobility reduction is selected, the following keywords are available in the PROPS section to describe the reduction in gas mobility in the presence of foam.l.

Table 22.2 Gas mobility reduction keywords

Keyword	Description
FOAMDCYO	Foam decay data as a function of oil saturation (Optional).
FOAMFCN	Foam mobility reduction dependence on capillary number (Optional).
FOAMFRM	Foam mobility reference reduction factor (Optional).
FOAMFSC	Foam mobility reduction dependence on foam surfactant concentration (Obligatory).

Table 22.2 Gas mobility reduction keywords

Keyword	Description
FOAMFSO	Foam mobility reduction dependence on oil saturation (Optional).
FOAMFST	Gas-water interfacial tension as a function of foam surfactant concentration (Obligatory if FOAMFCN keyword specified).
FOAMFSW	Foam mobility reduction dependence on water saturation (Optional)

With the exception of the FOAMFSC keyword, which must be specified if the functional model of gas mobility reduction is selected, and the FOAMFST keyword, which must be specified if the FOAMFCN keyword is specified, all of these keywords are optional.

The Foam Model's PROPS section data can be written to the PRINT file by using the FOAM mnemonic in the RPTPROPS keyword.

The injected foam concentration of a water or gas injector is specified using the WFOAM keyword in the SCHEDULE section.

Output controls

Output to the PRINT file can be controlled by using the following mnemonics in the RPTSCHED and RPTSOL keywords:.l.

Table 22.3 Output controls

Mnemonic	Description
FOAM	Foam concentration for each grid block.
FIPFOAM	A foam balance sheet for the field and each region.
FOAMADS	Foam currently adsorbed onto the rock.
FOAMDCY	The decay half life and the total decayed foam.
FOAMMOB	The gas mobility reduction factor. Note that, or the tabular model of gas mobility reduction, the gas mobility reduction includes the pressure dependent effect but **not** the shear effect, because the shear effect is calculated from the local velocities.
WELLS=2	Generates a field/group/well/connections summary of the foam injection and production.

The list of SUMMARY keywords has been extended to output foam-specific data. These keywords follow the passive tracer format with a keyword concatenated with the tracer name. Here foam is identified with the tracer name FOA. For example, the keyword for the field total foam production would be FTPRFOA.

Restrictions

Changing the pore volume in the EDIT section using PORV is not allowed with the foam option.

Summary of keywords

RUNSPEC section

The RUNSPEC keywords are:

Table 22.4 RUNSPEC keywords

Keyword	Description
FOAM	Activates the Foam model

GRID section

The GRID keywords are:

Table 22.5 GRID keywords

Keyword	Description
RPTGRID	Controls output from the GRID section.The FOAM mnemonic outputs a map of the 1/(poro*area) values used to calculate the gas velocity for the tabular model of mobility reduction.

PROPS section

The PROPS keywords are:

Table 22.6 PROPS keywords

Keyword	Description
ADSORP	Analytical adsorption isotherm (alternative to FOAMADS).
FOAMADS	Foam adsorption data.
FOAMDCYO	Foam decay data (as a function of oil saturation).
FOAMDCYW	Foam decay data (as a function of water saturation).
FOAMFCN	Foam mobility reduction functional dependence upon capillary number.
FOAMFRM	Foam mobility reference reduction factor.
FOAMFSC	Foam mobility reduction functional dependence upon foam surfactant concentration.
FOAMFSO	Foam mobility reduction functional dependence upon oil saturation.
FOAMFST	Gas-water interfacial tension as a function of foam surfactant concentration.
FOAMFSW	Foam mobility reduction functional dependence upon water saturation.

Table 22.6 PROPS keywords

Keyword	Description
FOAMMOB	Tables of the gas mobility reduction factor as a function of foam concentration.
FOAMMOBP	Gas mobility reduction as a function of pressure.
FOAMMOBS	Gas mobility reduction as a function of shear.
FOAMOPTS	Foam transport phase and mobility reduction method options.
FOAMROCK	Foam-rock properties.
RPTPROPS	Controls output from the PROPS section (FOAM mnemonic for foam properties).

SUMMARY section

The keywords shown in Table 22.7 control output of data specific to the Foam model.

Table 22.7 Foam SUMMARY output keywords

Field	Group	Well	Connection	Region	Block	Description
			CTFRFOA			Flow Rate (+ or -)
FTPRFOA	GTPRFOA	WTPRFOA				Production Rate
FTPTFOA	GTPTFOA	WTPTFOA	CTPTFOA			Production Total
FTIRFOA	GTIRFOA	WTIRFOA				Injection Rate
FTITFOA	GTITFOA	WTITFOA	CTITFOA			Injection Total
					BTCNFFOA	Concentration
					BTCNMFOA	Capillary number
FTIPTFOA				RTIPTFOA	BTIPTFOA	In Solution
				RTFTTFOA		Flow Total
FTADSFOA				RTADSFOA	BTADSFOA	Adsorption total
FTDCYFOA				RTDCYFOA	BTDCYFOA	Decayed tracer
FTMOBFOA				RTMOBFOA	BTMOBFOA	Gas mobility factor (excluding shear)
					BTHLFFOA	Decay Half life

SOLUTION section

The mnemonics shown in Table 22.8 control output of data specific to the Foam model using the RPTSOL keyword.

Table 22.8 RPTSOL output controls

Mnemonic	Description
FIPFOAM	Output of foam fluid in place reports: 1: gives a report for the whole field. 2: in addition, gives a report for each fluid-in-place region.
FOAM	Output of grid block foam concentration.
FOAMADS	Output of grid block foam adsorption.
FOAMCNM	Output of logarithm base ten of grid block capillary number if the functional model of mobility reduction has been selected and the FOAMFCN keyword specified
FOAMDCY	Output of grid block foam decay.
FOAMMOB	Output of gas mobility factor due to foam.

Note Note that the output of the gas mobility reduction factor for the tabular model will include the effects of pressure dependence (FOAMMOBP) but not the shear effect (FOAMMOBS), as the shear effect is calculated for each flow separately.

SCHEDULE section

The SCHEDULE section foam keywords are given below:

Table 22.9 SCHEDULE section

Keyword	Description
RPTSCHED	Controls output from the SCHEDULE section. Mnemonic FOAM outputs the foam concentrations and the foam adsorption when active.
WFOAM	Foam concentration for injection wells (can be used as a User Defined Argument (UDA)).

The mnemonics shown in Table 22.10 control output of data specific to the Foam model using the RPTSCHED keyword:

Table 22.10 RPTSCHED output controls

Mnemonic/Keyword	Description
FIPFOAM	Output of foam fluid in place reports: 1: gives a report for the whole field. 2: in addition, gives a report for each fluid-in-place region.
FOAM	Output of grid block foam concentration.
FOAMADS	Output of grid block foam adsorption.
FOAMCNM	Output of logarithm base ten of grid block capillary number if the functional model of mobility reduction has been selected and the FOAMFCN keyword specified
FOAMDCY	Output of grid block foam decay.
FOAMMOB	Output of gas mobility factor due to foam.

Note Note that the output of the gas mobility reduction factor for the tabular model will include the effects of pressure dependence (FOAMMOBP) but not the shear effect (FOAMMOBS), as the shear effect is calculated for each flow separately.

Example data set

RUNSPEC

RUNSPEC section

```
RUNSPEC
TITLE
 Simple FOAM model example
DIMENS
 10   10    3  /
OIL
WATER
GAS
DISGAS
FOAM
FIELD
EQLDIMS
1  100   10    1    1 /
TABDIMS
2    1   16   12    2   12    1    1 /
REGDIMS
 2    1    0    0    0    1 /
WELLDIMS
 2    1    1    2 /
NUPCOL
 4 /
START
 19 'APR' 1994  /
NSTACK
 24 /
FMTOUT
```

GRID

In this section, the geometry of the simulation grid and the rock permeabilities and porosities are defined.

The x and y direction cell sizes (DX, DY) and the porosities are constant throughout the grid. These are set in the first 3 lines after the equals keyword. The cell thicknesses (DZ) and permeabilities are then set for each layer. The cell top depths (TOPS) are needed only in the top layer (though they could be set throughout the grid). The specified MULTZ values act as multipliers on the transmissibilities between the current layer and the layer below.

```
GRID
--     ARRAY  VALUE     ------- BOX ------
EQUALS
      'DX'    1000      /
      'DY'    1000      /
      'PORO'  0.3       /
      'DZ'    20        1  10  1  10  1  1  /
      'PERMX' 500       /
      'MULTZ' 0.64      /
      'TOPS'  8325      /
      'DZ'    30        1  10  1  10  2  2  /
      'PERMX' 50        /
      'MULTZ' 0.265625  /
      'DZ'    50        1  10  1  10  3  3  /
      'PERMX' 200       /
/       EQUALS IS TERMINATED BY A NULL RECORD

-- THE Y AND Z DIRECTION PERMEABILITIES ARE COPIED FROM PERMX
--    SOURCE  DESTINATION  ------- BOX ------
COPY
      'PERMX'    'PERMY'   1  10  1  10  1  3  /
      'PERMX'    'PERMZ'   /
/
--
 OUTPUT OF DX, DY, DZ, PERMX, PERMY, PERMZ, MULTZ, PORO AND TOPS DAT
A
-- IS REQUESTED, AND OF THE CALCULATED PORE VOLUMES AND X, Y AND Z
-- TRANSMISSIBILITIES
RPTGRID
 'DX'  'DY'  'DZ'  'PERMX'  'PERMY'  'PERMZ'  'MULTZ'
 'PORO'  'TOPS'  'PORV'  'TRANX'  'TRANY'  'TRANZ'  /
```

PROPS

PROPS section:

```
PROPS ============================================================
--
--  SWAT   KRW   PCOW
--
SWFN
 0.12  0       0
 1.0   0.1     0  /
/
```

```
--
--  SGAS   KRG   PCOG
--
SGFN
    0     0       0
    0.02  0       0
    0.05  0.005   0
    0.12  0.025   0
    0.2   0.075   0
    0.25  0.125   0
    0.3   0.19    0
    0.4   0.41    0
    0.45  0.6     0
    0.5   0.72    0
    0.6   0.87    0
    0.7   0.94    0
    0.85  0.98    0
    1.0   1.0     0
/
/
-- OIL RELATIVE PERMEABILITY IS TABULATED AGAINST OIL SATURATION
-- FOR OIL-WATER AND OIL-GAS-CONNATE WATER CASES
--
--  SOIL     KROW     KROG
SOF3
    0        0        0
    0.18     0        0
    0.28     0.0001   0.0001
    0.38     0.001    0.001
    0.43     0.01     0.01
    0.48     0.021    0.021
    0.58     0.09     0.09
    0.63     0.2      0.2
    0.68     0.35     0.35
    0.76     0.7      0.7
    0.83     0.98     0.98
    0.86     0.997    0.997
    0.879    1        1
    0.88     1        1    /
/
--
--
    REF. PRES. REF. FVF  COMPRESSIBILITY  REF VISCOSITY  VISCOSIBILI
TY
--
PVTW
     4014.7     1.029        3.13D-6           0.31            0 /
--
--    REF. PRES   COMPRESSIBILITY
--
ROCK
        14.7          3.0D-6          /
```

```
--
-- SURFACE DENSITIES OF RESERVOIR FLUIDS
--
--        OIL   WATER   GAS
DENSITY
         49.1   64.79  0.06054  /
--
--   PGAS   BGAS   VISGAS
PVDG
     14.7 166.666   0.008
    264.7  12.093   0.0096
    514.7   6.274   0.0112
   1014.7   3.197   0.014
   2014.7   1.614   0.0189
   2514.7   1.294   0.0208
   3014.7   1.080   0.0228
   4014.7   0.811   0.0268
   5014.7   0.649   0.0309
   9014.7   0.386   0.047   /
--
--   RS      POIL  FVFO  VISO
PVTO
    0.001    14.7 1.062  1.04    /
    0.0905  264.7 1.15   0.975   /
    0.18    514.7 1.207  0.91    /
    0.371  1014.7 1.295  0.83    /
    0.636  2014.7 1.435  0.695   /
    0.775  2514.7 1.5    0.641   /
    0.93   3014.7 1.565  0.594   /
    1.270  4014.7 1.695  0.51
           5014.7 1.671  0.549
           9014.7 1.579  0.74    /
    1.618  5014.7 1.827  0.449
           9014.7 1.726  0.605   /
/
--
-- FOAM ADSOPTION ISOTHERMS
--
FOAMADS
 0.0     0.00000
 1.0     0.00005
30.0     0.00005  /
 0.0     0.00000
 1.0     0.00002
30.0     0.00002  /
--
-- ROCK DENSITY AND ADSORPTION MODEL
--
FOAMROCK
 1  2650 /
 2  2650 /
```

```
--
--  FOAM DECAY DATAAS A FUNCTION OF WATER SATURATION
--
FOAMDCYW
 0.0 3000
 1.0 2000 /
 0.0 3000
 1.0 2000 /
--
--  FOAM DECAY DATA AS A FUNCTION OF OIL SATURATION
--
FOAMDCYO
 0.0 3000.0
 1.0 2500.0  /
 0.0 3000.0
 1.0 2500.0  /
--
--  FOAM MOBILITY REDUCTION
--
FOAMMOB
  0      1
  0.001  0.4
  0.1    0.1
  1.2    0.05   /
--
--  PRESSURE EFFECT ON THE MOBILITY REDUCTION
--  IN THIS CASE HIGHER PRESSURE REDUCES THE FOAM EFFECTIVENESS
--
FOAMMOBP
 3000 0
 6000 0.2  /
--
--  SHEAR RATE EFFECT ON THE MOBILITY
--  IN THIS CASE HIGH SHEAR REDUCES THE FOAM EFFECTIVENESS
--
FOAMMOBS
 0.0 0
 4.0 0.1   /
--
-- OUTPUT CONTROLS FOR PROPS DATA
--
RPTPROPS
 'FOAM'  /
```

REGIONS

REGIONS section

```
REGIONS
EQUALS
 'SATNUM' 1 /
 'FIPNUM' 1 /
 'SATNUM' 2   1 3  1 3  1 3 /
 'FIPNUM' 2   1 3  1 3  1 3 /
/
```

SOLUTION

The SOLUTION section defines the initial state of the solution variables (phase pressures, saturations and gas-oil ratios)

```
SOLUTION
-- DATA FOR INITIALISING FLUIDS TO POTENTIAL EQUILIBRIUM
--
--    DATUM  DATUM   OWC    OWC    GOC    GOC    RSVD   RVVD   SOLN
--    DEPTH  PRESS  DEPTH   PCOW  DEPTH   PCOG  TABLE  TABLE   METH
EQUIL
       8400   4800   8500    0     8200    0      1      0       0
/
-- VARIATION OF INITIAL RS WITH DEPTH
--
--    DEPTH    RS
RSVD
       8200  1.270
       8500  1.270  /
--
-- OUTPUT CONTROLS
--
RPTSOL
 'PRESSURE' 'SWAT' 'SGAS' 'FOAM' /
```

SUMMARY

This section specifies data to be written to the SUMMARY files and which may later be used with the ECLIPSE graphics package.

```
SUMMARY
--REQUEST PRINTED OUTPUT OF SUMMARY FILE DATA
RUNSUM
SEPARATE
-- FIELD OIL PRODUCTION
FOPR
```

```
-- WELL GAS-OIL RATIO FOR PRODUCER
WGOR
'PRODUCER'
/
-- WELL BOTTOM-HOLE PRESSURE
WBHP
'PRODUCER'
/
-- SATURATIONS IN INJECTION AND PRODUCTION CELL
BGSAT
10 10 3
1  1  1
/
BOSAT
10 10 3
1  1  1
/
BWSAT
10 10 3
1  1  1
/
-- PRESSURE IN INJECTION AND PRODUCTION CELL
BPR
10 10 3
1  1  1
/
FTPRFOA
FTPTFOA
FTIRFOA
FTITFOA
FTIPTFOA
FTADSFOA
FTDCYFOA
BTCNFFOA
 1 1 1 /
 2 2 1 /
/
WTPRFOA
 'OP' /
RTIPTFOA
 1 /
BTADSFOA
 1 1 1 /
/
BTDCYFOA
 1 1 1 /
/
BTHLFFOA
 1 1 1 /
/
RTADSFOA
 1 /
```

```
RTDCYFOA
 1 /
```

SCHEDULE

The SCHEDULE section defines the operations to be simulated.

```
SCHEDULE
-- CONTROLS ON OUTPUT AT EACH REPORT TIME
RPTSCHED
 'PRESSURE' 'SWAT' 'SGAS' 'SOIL' 'FOAM' 'WELLS=2' 'NEWTON=2' 'FIPFOA
M=2'
 'TRADS' 'FOAMADS' 'FOAMDCY' 'FOAMMOB' /
RPTRST
 'BASIC=3' 'FIP' 'FREQ=2' /
-- SET 'NO RESOLUTION' OPTION
DRSDT
 0  /
-- WELL SPECIFICATION DATA
--
--     WELL   GROUP LOCATION  BHP   PI
--     NAME   NAME    I  J   DEPTH DEFN
WELSPECS
    'PRODUCER' 'G'   10 10    8400 'OIL'  /
    'INJECTOR' 'G'    1  1    8335 'GAS'  /
/
-- COMPLETION SPECIFICATION DATA
--
--     WELL     -LOCATION- OPEN/ SAT CONN  WELL
--     NAME     I  J K1 K2 SHUT  TAB FACT  DIAM
COMPDAT
    'PRODUCER'  10 10 3  3 'OPEN' 0   -1   0.5  /
    'INJECTOR'   1  1 1  1 'OPEN' 1   -1   0.5  /
/
--
-- PRODUCTION WELL CONTROLS
--
--      WELL     OPEN/  CNTL   OIL  WATER   GAS  LIQU   RES   BHP
--      NAME     SHUT   MODE  RATE   RATE  RATE  RATE  RATE
WCONPROD
     'PRODUCER' 'OPEN' 'ORAT' 20000  4*                     1000 /
/
--
-- INJECTION WELL CONTROLS
--
--      WELL     INJ   OPEN/  CNTL    FLOW
--      NAME    TYPE   SHUT   MODE    RATE
WCONINJ
     'INJECTOR' 'GAS' 'OPEN' 'RATE'  100000  /
/
```

```
TSTEP
 1 9 /
--
-- SPECIFY THE FOAM INJECTOR
--
WFOAM
 'INJECTOR'  1.1 /
/
TSTEP
 1 9 90 20*100  5*1000 /
END
```

Formulation of the Equations

Chapter 23

Introduction

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

ECLIPSE 100 and ECLIPSE 300 contain options for both the fully implicit, IMPES (IMplicit Pressure Explicit Saturation) and AIM (Adaptive Implicit Method) solution procedures. The fully implicit method is totally stable and may be used to solve 'difficult' problems such as coning studies. Although IMPES is potentially unstable, it is less dispersive and sometimes faster than the fully implicit method, and may be used on 'easy' problems such as history matching applications where the time steps are usually small. The AIM method for compositional studies is a compromise between fully implicit and IMPES methods, allowing cells in 'difficult' regions to remain fully implicit while employing the advantage of an IMPES description in 'easy' regions.

ECLIPSE 100

The default solution procedure is fully implicit for all black oil runs. This is robust and efficient for black oil problems, generally allowing for large time steps in the simulation. Both fully implicit and IMPES may be used in the same run by using keywords `DIMPES` or `DIMPLICT` at appropriate places in the `SCHEDULE` section

ECLIPSE 300

The default solution procedure is `AIM` for all compositional runs, `FULLIMP` for all black oil runs, and `JALS` for all thermal runs. The default solution procedure may be changed by using one of the keywords `AIM`, `FULLIMP` or `IMPES` in the `RUNSPEC` section.

Description of equations

The non-linear residual, R_{fl}, for each fluid component in each grid block at each time step is:

$$R_{fl} = \frac{dM}{dt} + F + Q \qquad \text{[EQ 23.1]}$$

where

dM is the mass, per unit surface density, accumulated during the current time step, dt

F is the net flow rate into neighboring grid blocks

Q is the net flow rate into wells during the time step

R_{fl} is defined for each cell and each fluid in the study.

In the black oil case the fluids are oil, water and gas; in the compositional case they are the hydrocarbon components and water.

ECLIPSE 300

In the compositional case, for two hydrocarbon phase cells, there is in addition the requirement that the Gibbs energy of the system is at a minimum. The first order conditions for this yield a component equal fugacity condition:

$$R_{fu} = f_l - f_v = 0 \qquad \text{[EQ 23.2]}$$

To solve the entire system, we require that $R_{fl} \rightarrow 0$ and $R_{fu} \rightarrow 0$ to within a required tolerance.

Variable sets

In order to solve the residual equations, we require a set of solution variables. The number of independent variables must be equal to the number of residual conditions; the residual equations may then be solved using Newton's method. The actual variable set chosen differs between ECLIPSE 100 and ECLIPSE 300, so each is described separately.

ECLIPSE 100 black oil

The primary solution variables X are pressure P and two saturations for a three phase black oil study. The water saturation S_w and either S_g, R_s or R_v are chosen to complete the set. For a 3 component black oil system (oil, water, gas), the residual R and the solution, X, are 3 component vectors in each grid block. By default, the solution procedure is fully implicit;

$$R = \begin{bmatrix} R_o \\ R_w \\ R_g \end{bmatrix} \qquad X = \begin{bmatrix} P_o \\ S_w \\ S_g \text{ or } R_s \text{ or } R_v \end{bmatrix} \qquad \text{[EQ 23.3]}$$

and the Jacobian, $J = \frac{dR}{dX}$, takes the form

$$\frac{dR_i}{dX_j} = \begin{bmatrix} \frac{dR_o}{dP_o} & \frac{dR_o}{dS_w} & \frac{dR_o}{dS_g} \\ \frac{dR_w}{dP_o} & \frac{dR_w}{dS_w} & \frac{dR_w}{dS_g} \\ \frac{dR_g}{dP_o} & \frac{dR_g}{dS_w} & \frac{dR_g}{dS_g} \end{bmatrix}_{ij} \qquad \text{[EQ 23.4]}$$

The mass change during the time step, dt, is then proportional to

$$dM = M_{t+dt} - M_t \qquad \text{[EQ 23.5]}$$

with

$$M = \text{PV} \begin{bmatrix} \frac{S_o}{B_o} + \frac{R_v S_g}{B_g} \\ \frac{S_w}{B_w} \\ \frac{S_g}{B_g} + \frac{R_s S_o}{B_o} \end{bmatrix} \qquad \text{[EQ 23.6]}$$

where

PV is the pore volume

B_o is the oil formation volume factor

B_w is the water formation volume factor

B_g is the gas formation volume factor

R_s is the solution gas/oil ratio

R_v is the vapor oil/gas ratio.

When S_g is zero the solution variable becomes R_s (undersaturated oil) and when S_o is zero the solution variable becomes R_v (undersaturated gas). Terms in the Jacobian are adjusted in accordance with the change of variable. No approximations are made in evaluating the Jacobian in ECLIPSE 100. Great care is taken to compute all derivatives to ensure quadratic convergence of Newton's method.

Hydrocarbon states are:

State 1 Gas only $R_s = 0$, $S_g = 1 - S_w$,

variables are P_o, S_w, R_v.

State 2 Gas & oil, $R_v = R_{v\,\mathrm{sat}}$, $R_s = R_{s\,\mathrm{sat}}$,

variables are P_o, S_w, S_g

State 3 Oil only, $R_v = 0$, $S_g = 0$,

variables are P_o, S_w, R_s.

The number of cells in each state, together with the number of cells changing state during a Newton iteration (`NTRAN`), are printed in the summary of each Newton iteration.

Note Cells almost entirely full of water are in state 2 by default. Therefore the solution variable R_s is set to $R_{s\,\mathrm{sat}}$. A transition from state 2 to state 3 may occur in such blocks, for example due to the influx of under-saturated oil. The cell's R_s value is then limited to not exceed the average GOR of inflows from neighboring cells as this will probably be a better estimate then $R_{s\,\mathrm{sat}}$.

ECLIPSE 300 compositional

The primary solution variables, used for cells in all phase states, are $X = (P, m_1, \ldots, m_N, m_w)$

where

$m_1, \ldots, m_N$ are molar densities of each component, and

m_w is the molar density of water.

In `IMPSAT` mode we add three extra primary solution variables: X_o, X_g, X_w, which represent the phase saturations.

To normalize the residual equations, an additional residual condition is needed, which is usually referred to as the volume balance constraint. This condition requires that the saturations should all add up to unity and is written as

$$R_m = PV - FV = 0 \qquad \text{[EQ 23.7]}$$

where PV is the cell pore volume, and FV is the fluid volume, given by

$$FV = PV\left(\frac{m_t L}{b_o^m} + \frac{m_t V}{b_g^m} + \frac{m_w}{b_w^m}\right) \qquad \text{[EQ 23.8]}$$

with the total hydrocarbon molar density given by

$$m_t = \sum_c m_c \qquad \text{[EQ 23.9]}$$

where

L and V are the liquid and vapor mole fractions, and

b_o^m , b_g^m and b_w^m are the molar densities of oil, gas and water.

The form of the Jacobian matrix condition in the compositional case is then

$$J = \frac{dR_{fl}}{dX}$$

which can be depicted in terms of the primary solution variables (P, S_w, m_c for each hydrocarbon component c) and residual equations (R_w, R_c and R_m) for the fully implicit method as

$$J = \begin{array}{l} \\ \text{Water} \\ \text{Comp1} \\ \dots \\ \text{Comp}N_c \\ \text{Volumebal} \end{array} \begin{array}{c} \begin{array}{ccccc} P & S_w & m_1 & \dots & m_N \end{array} \\ \begin{bmatrix} x & x & x & \dots & x \\ x & x & x & \dots & x \\ \dots & \dots & \dots & \dots & \dots \\ x & x & x & \dots & x \\ x & x & x & \dots & x \end{bmatrix} \end{array} \qquad \text{[EQ 23.10]}$$

The residual equation $R(X) = 0$ is solved for X using Newton's method.

In two phase blocks further secondary solution variables are required. These are the $J_i = \log(K_i)$ values for each component, and the vapor mole fraction V. In terms of these variables the equal fugacity residual condition may be expressed as:

$$R_{fu} = J - H_t + H_v \qquad \text{[EQ 23.11]}$$

where

$$H_l = \ln\left(\frac{f_l}{Px}\right) \qquad \text{[EQ 23.12]}$$

$$H_v = \ln\left(\frac{f_v}{Py}\right) \qquad \text{[EQ 23.13]}$$

An additional normalization residual (the Rachford-Rice equation) exists in the compositional case,

$$R_n = \sum_i \left(z_i \frac{(K_i - 1)}{1 + V(K_i - 1)}\right) = 0 \qquad \text{[EQ 23.14]}$$

where z_i and K_i are component mole fractions and K-values respectively. The solution variables for the flash calculation are $X_f = (J_1, \dots, J_N, V)$.

The flash solution X_f is found at each iteration by solving the equilibrium condition that

$$R_{fu}(X_f) = 0 \,. \qquad \text{[EQ 23.15]}$$

In IMPSAT mode we have three additional residuals to define the saturation variables X_o, X_g, X_w :

$$R_p = PV(X - S_p(P, m)) = 0 \qquad \text{[EQ 23.16]}$$

where $S_p(P, m)$ is the fractional volume occupied by phase p .

The fully implicit method

The fully implicit method is the default for black oil runs in both ECLIPSE 100 and ECLIPSE 300. It is not recommended for compositional runs where there are often too many components to be able to use the fully implicit method efficiently.

Newton's method is used to solve the non-linear residual equations R(X) = 0. Several iterations may be required to reduce the residuals to a 'sufficiently small' value. In ECLIPSE 100, two measures are used to define the meaning of 'sufficiently small'. These are the material balance error and the maximum saturation normalized residual.

The IMPES and AIM methods

IMPES

The IMPES (IMplicit Pressures Explicit Saturations) residual is similar to the fully implicit residual except that all flow and well terms are computed using saturations (or R_S, R_V) in a black oil run; or molar densities in a compositional run at the beginning of each time step.

$$R = \frac{M_{t+dt} - M_t}{dt} + F(P_{t+dt}, S_t) + Q(P_{t+dt}, S_t) \qquad \text{[EQ 23.17]}$$

The mass terms M_{t+dt} are evaluated using both pressures and saturations at the end of the time step. This makes the non-linear residual equation, R = 0 much easier to solve because there are now no non-linearities arising from relative permeabilities that remain fixed throughout the time step. However, to solve the IMPES equations correctly it is still necessary to iterate until all residuals have been reduced to a sufficiently small value.

The linear equations arising from Newton's method are also much easier to solve in the IMPES case because derivatives of flows with respect to saturations are zero. The linear equations are solved sequentially, first for pressure and subsequently for saturation changes. This contrasts with the fully implicit method where the linear equations must be solved simultaneously.

ECLIPSE 100

Time steps are chosen automatically to limit saturation changes to 5%. However a converged time step is accepted if the largest saturation change is less than 10% and if the largest pressure change is less than 200psi. These values may be altered using keyword `DIMPES`.

ECLIPSE 300

Time steps are chosen automatically to limit saturation changes to 5%. The default time step selection may be modified using the `TSCRIT` keyword.

ECLIPSE 300

In ECLIPSE 300, the IMPES formulation is strictly an IMPEM (IMplicit Pressure Explicit Mobility) method. The mass terms M_{t+dt} are evaluated using both pressures and molar densities at the end of the time step. The flow terms between cells are evaluated assuming the saturations, generalized mobilities and reservoir density terms are all fixed at the previous time step.

AIM

ECLIPSE 300

The Adaptive IMplicit method (`AIM`) is a compromise between the fully implicit and IMPES procedures. Cells with a high throughput ratio are chosen to be implicit for stability and to obtain large time steps, while the majority of cells can still be treated as IMPES where the solution may be changing little. All completions are treated implicitly. The target fraction of implicit cells in a compositional run is 1%, but this can be altered by the `AIMFRAC` keyword.

The CFLLIMIT option for AIM and IMPES runs

ECLIPSE 300

This option is based on the paper "IMPES Stability: The Stable Step" by K.H. Coats [Ref. 88]. The reader is referred there for all technical points. By default, AIM and IMPES runs choose the implicit cells/time step by looking at throughput and/or solution change. Whilst this usually works it is not a rigorous procedure, and in some situations, such as when there are large capillary forces, it can make the wrong choice.

The CFLLIMIT option places the time-step selection and implicit cell choice on a much more rigorous footing based on the maximum **stable** time-step allowed in the **explicit** cells. This time step can be calculated from conditions given in the above paper. Due to the robustness of this approach when it is used there is no limit placed on the maximum rate of change of capillary pressure. See keyword DPCDT and compare with the usual default for AIM or IMPES runs.

For those runs where this option does make a difference: aside from differences caused by strictly honoring the rate of change of capillary pressure, the user may notice the following:

IMPES runs

The run takes longer due to smaller, in some cases substantially smaller, time steps being taken. The advantage is that oscillations in the results virtually disappear.

AIM runs

Times can actually improve as now the 'right' cells have been chosen to be implicit, which can lead to fewer convergence problems. In some cases, because the correct cells are being treated implicitly and the time step has been chosen to be stable for the remaining explicit cells, a significant improvement in performance and results are experienced.

There are also cases where performance is severely degraded, although results are not. This occurs when the case can run, with no obvious ill-effects, with time steps very much larger than those required by stability considerations. There is usually some benefit to be seen in a close examination of the results, but if these benefits do not justify a large increase in CPU time, or indeed if the increase makes running the problem uneconomic, then there are two courses of action.

1. items 2 to 4 of the CFLLIMIT keyword can be altered to allow time steps that are theoretically unstable
2. not to use this keyword, but to use the default AIM settings, and perhaps use TUNING to prevent the default AIM settings from selecting some of the larger time steps.

Instability in IMPES

The most common instability using IMPES arises when a phase crosses the critical saturation and becomes mobile. In Figure 23.1 water is flowing from cell A to cell B. Water levels at time t are shown by the dark shading. Because the saturation in cell B is below S_{wcr} at time t, water cannot flow out of cell B. During the time step, dt, the water saturation in cell B can rise significantly above S_{wcr} to a value greater than that in cell A. At the next time step, water is allowed to flow out of cell B at a high rate especially if k_{rw} increases sharply above S_{wcr}, and the saturation may fall below S_{wcr} making water in cell B immobile again at the following time step. Saturations may therefore oscillate about their critical values using IMPES. These difficulties do not arise with the fully-implicit method because as soon as S_w is greater than S_{wcr} it is permitted to flow out of cell B.

Figure 23.1 The most common instability using IMPES

This type of instability may limit the efficiency of the IMPES method. The problem is reduced in some simulators (but not in ECLIPSE) by automatically rescaling the input relative permeability curves to equally spaced saturation intervals. This is not done in ECLIPSE because:

- you may wish the data to be honored precisely
- sharp relative permeability contrasts are handled efficiently by the fully implicit method
- table look-up is equally efficient either way (contrary to popular belief)
- the user has the option of inputting ‘smooth’ k_r curves.

Convergence criteria for residual equations

The Newton solver aims to reduce the residuals $R(X)$ to zero. In ECLIPSE 100 the convergence criteria are primarily based on material balance and maximum residual checks on R. In ECLIPSE 300, the convergence criteria are primarily based on the solution change Δx being 'sufficiently small' in the Newton iteration.

ECLIPSE 100

Material balance

If the residuals are summed over all cells in the reservoir, the flow terms F cancel, because the flow out of one cell is always equal and opposite in sign to the corresponding flow into its neighboring cell. Thus the sum of the residuals for each phase or component corresponds to the net mass accumulation within the reservoir less the net influx through wells. This is the material balance error. For a three-component system we have

$$\sum_i (R_o)_i = \sum_i \left(\frac{dM_o}{dt}\right)_i + \sum_i (Q_o)_i$$

$$\sum_i (R_w)_i = \sum_i \left(\frac{dM_w}{dt}\right)_i + \sum_i (Q_w)_i \qquad \text{[EQ 23.18]}$$

$$\sum_i (R_g)_i = \sum_i \left(\frac{dM_g}{dt}\right)_i + \sum_i (Q_g)_i$$

where

$\sum_i$ refers to the sum over all reservoir cells and

$(R_o)_i$ is the oil residual in cell i etc.

In ECLIPSE 100 the material balance errors are converted to meaningful, problem independent, numbers by scaling to equivalent field saturation values:

$$\mathrm{MB}_o = \overline{B_o} dt \left\{ \left(\sum_i (R_o)_i \right) / \left(\sum_i (\mathrm{PV})_i \right) \right\}$$

$$\mathrm{MB}_w = \overline{B_w} dt \left\{ \left(\sum_i (R_w)_i \right) / \left(\sum_i (\mathrm{PV})_i \right) \right\} \qquad \text{[EQ 23.19]}$$

$$\mathrm{MB}_g = \overline{B_g} dt \left\{ \left(\sum_i (R_g)_i \right) / \left(\sum_i (\mathrm{PV})_i \right) \right\}$$

where $\overline{B_o}$ is the average oil formation factor etc.

The numerical values of $\mathrm{MB_o}$, $\mathrm{MB_w}$ and $\mathrm{MB_g}$ are computed after each Newton iteration and the material balance errors are considered to be sufficiently small if they are all less than 1.0E-7. MB values are printed out in the summary of each Newton iteration. Conventional material balance accounts in conventional units can also be printed at each report time.

Normalized residuals

Although great care is taken to ensure that material balance errors are unusually small in ECLIPSE 100, this is not considered to be a sufficiently rigorous test of convergence. The second test is obtained by computing the maximum saturation normalized residuals

$$\mathrm{CNV}_o = \overline{B_o} dt \cdot \mathrm{MAX}_i \left| \frac{(R_o)_i}{(\mathrm{PV})_i} \right|$$

$$\mathrm{CNV}_w = \overline{B_w} dt \cdot \mathrm{MAX}_i \left| \frac{(R_w)_i}{(\mathrm{PV})_i} \right|$$

$$\mathrm{CNV}_g = \overline{B_g} dt \cdot \mathrm{MAX}_i \left| \frac{(R_g)_i}{(\mathrm{PV})_i} \right| \qquad \text{[EQ 23.20]}$$

where MAX_i is the maximum value over all cells in the reservoir.

By converting each convergence error to an equivalent saturation value we can attach sensible limits to the CNV numbers, which are considered to have converged if they are all less than 0.001.

The convergence criteria can be altered by the `TUNING` keyword. However, the default values are most often sufficient and it is **not** recommended that they should be changed. Loosening the convergence tolerances in a difficult run often results in increased run times.

ECLIPSE 300

The convergence criteria in ECLIPSE 300 are based on solution change targets. Since the primary solution variables x are molar densities, it is natural to use these to control the convergence. However, large saturation changes may occur for very small changes in composition, so the use of an effective saturation is preferred. Consider the expression:

$$\Delta S' = \frac{du^T}{dm_c} \cdot \Delta m_c \qquad \text{[EQ 23.21]}$$

where

$$u^T = \frac{m_t L}{b_o^m} + \frac{m_t V}{b_g^m} + \frac{m_w}{b_w^m} \qquad \text{[EQ 23.22]}$$

u^T is the fluid volume per unit pore volume

$\Delta S'$ is thus the change in fluid volume per unit pore volume for a given molar density change, that is the effective saturation change for the molar density change.

In all cases the maximum solution change over a time step is monitored and compared with the specified target.

Convergence is based on the pressure and saturation changes Δp and $\Delta S'$ being 'sufficiently small'. These targets `SCONVP` and `SCONVS` in the `CVCRIT` keyword can be used to tighten down the convergence.

ECLIPSE 100 Black Oil

The flow rate into cell i from a neighboring cell n, F_{ni}, is

$$F_{ni} = T_{ni}\begin{bmatrix} \frac{k_{ro}}{B_o\mu_o} & 0 & \frac{R_v k_{rg}}{B_g\mu_g} \\ 0 & \frac{k_{rw}}{B_w\mu_w} & 0 \\ \frac{R_s k_{ro}}{B_o\mu_o} & 0 & \frac{k_{rg}}{B_g\mu_g} \end{bmatrix}_u \times \begin{bmatrix} dP_{oni} \\ dP_{wni} \\ dP_{gni} \end{bmatrix} \qquad \text{[EQ 23.23]}$$

where

$$dP_{oni} = P_{on} - P_{oi} - \rho_{oni}G(D_n - D_i)$$

$$dP_{wni} = P_{wn} - P_{wi} - \rho_{wni}G(D_n - D_i)$$

$$= P_{on} - P_{oi} - \rho_{wni}G(D_n - D_i) - P_{cown} + P_{cowi}$$

$$dP_{gni} = P_{gn} - P_{gi} - \rho_{gni}G(D_n - D_i)$$

$$= P_{on} - P_{oi} - \rho_{gni}G(D_n - D_i) + P_{cogn} - P_{cogi}$$

T_{ni} is the transmissibility between cells n and i,

k_r is the relative permeability

(k_{ro} is the relative permeability of oil etc.),

μ is the viscosity (μ_w is the viscosity of water etc.),

dP is the potential difference

(dP_{gni} is the gas potential difference between cells n and i),

ρ is the fluid density

(ρ_{oni} is the density of oil at the interface between cells n and i),

G is the acceleration due to gravity

(0.0000981 in metric units, 0.00694 in field units and 0.000968 in lab units),

D is the cell center depth.

The subscript u indicates that the fluid mobilities are to be evaluated in the upstream cell (cell n if dP_{ni} is positive, cell i if dP_{ni} is negative). The upstream calculation applies separately for each equation (oil, water, gas) so that, for example, oil may flow from cell i to cell n while water flows from cell n to cell i.

The net flow rate from cell i into neighboring cells is obtained by summing over the neighboring cells, $F_i = \sum_n F_{ni}$.

The rate of flow into a production well from cell i is

$$Q_i = -T_{wi}(P_{oi} - H_{iw} - P_{bh}) \begin{bmatrix} \frac{k_{ro}}{B_o \mu_o} + \frac{R_v k_{rg}}{B_g \mu_g} \\ \frac{k_{rw}}{B_w \mu_w} \\ \frac{k_{rg}}{B_g \mu_g} + \frac{R_s k_{ro}}{B_o \mu_o} \end{bmatrix}_i \quad \text{[EQ 23.24]}$$

where

T_{wi} is the well connection transmissibility factor

H is the hydrostatic head correction

P_{bh} is the bottom hole pressure

Well terms are discussed in detail in a separate chapter: see "Well Inflow Performance" on page 1187.

ECLIPSE 300 Compositional

The flow rate of component c embedded in a phase p (p=o,w,g) into cell i from a neighboring cell n, F_{pni}^c , is

$$F_{pni}^c = T_{ni} M_p^c \, dP_{pni} \quad \text{[EQ 23.25]}$$

where

M_p^c is the generalized mobility of component c in phase p and is given by

$$M_p^c = x_p^c \, k_{rp}(S_p) \frac{b_p^m}{\mu_p} \quad \text{[EQ 23.26]}$$

where

x_p^c is the mole fraction of component c in phase p

k_{rp} is the relative permeability of phase p

S_p is the saturation of phase p

b_p^m is the molar density of phase p

μ_p is the viscosity of phase p

dP_{pni} is the potential difference of phase p between cells n and i, given by

$$dP_{pni} = P_{pn} - P_{pi} - \rho_{pni} G(D_n - D_i) \quad \text{[EQ 23.27]}$$

or

$$dP_{pni} = P_n - P_i + P_{cpn} - P_{cpi} - \rho_{pni} G(D_n - D_i) \quad \text{[EQ 23.28]}$$

where

P_{cp} is the capillary pressure for phase p

ρ_p is the mass density of phase p

G is the acceleration due to gravity

(0.0000981 in metric units, 0.00694 in field units and 0.000968 in lab units)

D is the cell center depth

and

T_{ni} is the transmissibility between cells n and i.

The fluid mobilities M_p^c are evaluated in the upstream cell for each phase p separately (cell n if dP_{pni} is positive, cell i if dP_{pni} is negative). The upstream calculation applies separately for each phase (oil, water, gas) so that, for example, oil may flow from cell i to cell n while water flows from cell n to cell i.

The net flow rate of component c from cell i into neighboring cells is obtained by summing over all phases over the neighboring cells, $F_i^c = \sum_n F_{pni}^c$.

For a component c existing in both oil and gas phases with liquid and vapor mole fractions x_c and y_c, the component mobilities in oil and gas can then be written as

$$M_o^c = x_c k_{ro}(S_w, S_g)\frac{b_o^m}{\mu_o} \qquad \text{[EQ 23.29]}$$

and

$$M_g^c = y_c k_{rg}(S_g)\frac{b_g^m}{\mu_g} \qquad \text{[EQ 23.30]}$$

The rate of flow of component c embedded in phase p into a production well from cell i is

$$Q_{pi}^c = -T_{wi}(P_{pi} - H_{iw} - P_{bh})M_{pi}^c \qquad \text{[EQ 23.31]}$$

where

T_{wi} is the well connection transmissibility factor

H is the hydrostatic head correction

P_{bh} is the bottom hole pressure

M_{pi}^c is the generalized mobility of component c in phase p

See "Well Inflow Performance" on page 1187 for further information.

Transport coefficients

ECLIPSE 300 only

The component mobilities can be modified in ECLIPSE 300 by including transport coefficients, [Ref. 102]. The transport coefficients α_p^c act as multipliers to the mobility. Thus [EQ 23.26] becomes:

$$M_p^c = \alpha_p^c \, x_p^c \, k_{rp}\frac{b_p^m}{\mu_p} \qquad \text{[EQ 23.32]}$$

where

$\alpha_p^c = \alpha_p^c(z^t)$ is the transport coefficient given by the ALPHA keyword

z^t is the mole fraction of component t defined by the TRCOEF keyword.

Transport coefficients for drainage and imbibition

ECLIPSE 300 only

When hysteresis is present it is possible in ECLIPSE 300 to modify the component mobilities depending on whether drainage (decreasing wetting phase saturation) or imbibition (increasing wetting phase saturation) is occurring. The component mobilities are modified by introducing transport coefficients which act as multipliers to the mobility. The transport coefficients for the drainage and imbibition process are specified separately and denoted by $\alpha_p^{c,d}$ and $\alpha_p^{c,i}$, respectively.

Thus [EQ 23.26] becomes:

$$M_p^c = \begin{cases} \alpha_p^{c,d} x^c{}_p k_{rp}^d \dfrac{b^m{}_p}{\mu_p} & \text{if drainage process} \\ \alpha_p^{c,i} x^c{}_p k_{rp}^i \dfrac{b^m{}_p}{\mu_p} & \text{if imbibition process} \end{cases}$$

where

$\alpha_p^{c,d} = \alpha_p^{c,d}(z^t)$ is the transport coefficient given by the ALPHAD keyword

$\alpha_p^{c,i} = \alpha_p^{c,i}(z^t)$ is the transport coefficient given by the ALPHAI keyword

z is the mole fraction of component t defined by the TRCOEF keyword.

Reservoir densities

ECLIPSE 100

In the black oil case, reservoir densities (ρ) of oil and gas are computed from surface densities (ρ_s) using

$$\rho_o = (\rho_{so} + CR_s\rho_{sg})/B_o$$
$$\rho_g = (C\rho_{sg} + R_v\rho_{so})/B_g \qquad \text{[EQ 23.33]}$$

with C = 1 for metric and lab units and C = 178.1076 in field units.

The reservoir density at the interface between cells n and i is computed as the average of the reservoir densities in cells n and i

$$\rho_{oni} = (\rho_{on} + \rho_{oi})/2$$
$$\rho_{gni} = (\rho_{gn} + \rho_{gi})/2 \qquad \text{[EQ 23.34]}$$

ECLIPSE 300

In the compositional case, the reservoir densities are obtained from

$$\rho_p = MW_p b_p^m \qquad \text{[EQ 23.35]}$$

where

MW_p is the average molecular weight of the phase p .

The reservoir density at the interface between cells n and i is computed as the saturation weighted average of the reservoir densities in cells n and i

$$\rho_{pni} = \frac{(\rho_{pn}S_{pn} + \rho_{pi}S_{pi})}{(S_{pn} + S_{pi})} \qquad \text{[EQ 23.36]}$$

Newton iteration of the non-linear residual

Given the non-linear residual $R = R_{fl}$ and the solution variables X, we wish to solve the non-linear equation $R(X) = 0$. In each iteration of the non-linear equations the sequence of operations is:

1 Given current solution X, obtain $R(X)$, then find a correction Δx such that $R(X+\Delta x) \sim R(X) + J \cdot \Delta x = 0$, where J is the Jacobian $\frac{d}{dX}R(X)$.

2 To do this, solve the linear equations

$$J \cdot \Delta x = R(X) \qquad \text{[EQ 23.37]}$$

or

$$\Delta x = J^{-1} R(X) \qquad \text{[EQ 23.38]}$$

3 Update the solution vector, $X \rightarrow X + \Delta x$. If some measure of convergence has been achieved, exit; if not, repeat.

In the fully implicit case, these equations may be solved for X, using the linear solver to give

$$\Delta x = J^{-1} \cdot R'_{fl} \qquad \text{[EQ 23.39]}$$

In the `IMPES` case, the matrix J^{-1} has a special structure, in that band terms only appear for pressure derivatives.

ECLIPSE 300

For the compositional case in ECLIPSE 300, with the solution variables described above, the forms of the diagonal and band terms of J^{-1} are

$$J'_D = \begin{array}{l} \\ \text{Water} \\ \text{Comp1} \\ \dots \\ \text{Comp}N_c \\ \text{Volumebal} \end{array} \begin{array}{c} \begin{array}{ccccc} P & S_w & m_1 & \dots & m_N \end{array} \\ \begin{bmatrix} x & x & 0 & \dots & 0 \\ x & 0 & x & \dots & 0 \\ \dots & \dots & \dots & \dots & \dots \\ x & 0 & 0 & \dots & x \\ x & x & x & \dots & x \end{bmatrix} \end{array} \qquad \text{[EQ 23.40]}$$

$$J'_B = \begin{array}{l} \\ \text{Water} \\ \text{Comp1} \\ \dots \\ \text{Comp}N_c \\ \text{Volumebal} \end{array} \begin{array}{c} \begin{array}{ccccc} P & S_w & m_1 & \dots & m_N \end{array} \\ \begin{bmatrix} x & 0 & 0 & \dots & 0 \\ x & 0 & 0 & \dots & 0 \\ \dots & \dots & \dots & \dots & \dots \\ x & 0 & 0 & \dots & 0 \\ 0 & 0 & 0 & \dots & 0 \end{bmatrix} \end{array} \qquad \text{[EQ 23.41]}$$

Thus there is one column, the diagonal, and the bottom row in the diagonal case, and one column only in the band case.

By subtracting a multiple of each line from the last, all terms in the molar balance equations $R_c = 0$ may be zeroed except the first, $\frac{dR'_m}{dP}$. This is a banded pressure equation, which may be solved using the nested factorization linear solver (see "Solution of the Linear Equations" on page 937).

The remaining elements of X may then be found by back-substitution.

ECLIPSE 300

In the Adaptive IMplicit (AIM) method, the molar densities can be eliminated from the explicit cells as in the IMPES case. However there are also equations coupling pressure and molar density changes for all implicit cells. The resulting Jacobian can still be solved by the nested factorization linear solver. Remaining elements of X are obtained by back substitution only in explicit cells.

ECLIPSE 100

Similarly in the black oil case in ECLIPSE 100, the diagonal and band terms of J^{-1} for a three phase oil-water-gas system are

$$J'_D = \begin{array}{c} \\ \text{Water} \\ \text{Oil} \\ \text{Gas} \end{array} \begin{array}{c} \begin{array}{ccc} P & S_w & X_3 \end{array} \\ \begin{bmatrix} x & x & 0 \\ x & 0 & x \\ x & 0 & x \end{bmatrix} \end{array} \qquad \text{[EQ 23.42]}$$

$$J'_B = \begin{array}{c} \\ \text{Water} \\ \text{Oil} \\ \text{Gas} \end{array} \begin{array}{c} \begin{array}{ccc} P & S_w & X_3 \end{array} \\ \begin{bmatrix} x & 0 & 0 \\ x & 0 & 0 \\ x & 0 & 0 \end{bmatrix} \end{array} \qquad \text{[EQ 23.43]}$$

Gas Field Operations Model

Chapter 24

Introduction

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL

The Gas Field Operations model is a Special Extension to both ECLIPSE 100 and ECLIPSE 300. It contains a set of facilities designed to model gas field production constraints and operations. The principal features in this set are:

Seasonality profile

The required rate of gas production is not constant throughout the year, but varies from month to month to reflect the annual pattern of demand. To generate this varying production rate, you enter an annual profile made up of monthly multipliers to the mean rate or DCQ (Daily Contracted Quantity). The field's gas production rate target for each month will be set equal to the DCQ multiplied by the month's profile factor.

Adjustment of DCQ to allow for swing

A typical gas sales contract requires that the field should be able to produce at a rate, which is a given multiple of the DCQ. The given multiple is the Swing Factor, which for example may have a value of 1.67. A suitable value of the DCQ must be calculated that is valid over the period of each contract, depending on the deliverability of the field over the period. The DCQ may be reviewed on an annual basis (contract years) or more frequently (contract quarters, for example). ECLIPSE can determine a suitable value of the DCQ for each contract period by solving the period in two passes. In the first pass, ECLIPSE tests whether the field can produce at the rate of DCQ × Swing Factor, with the DCQ set at the current value. If the field cannot produce at this rate, ECLIPSE works out how much the DCQ should be reduced to make this target sustainable. If requested, ECLIPSE may perform a number of iterations of the first pass,

with different DCQ values, to determine the highest DCQ that obeys the swing requirement over the period, to a specified accuracy. In the second pass, the field is produced at the rate of DCQ (reduced if necessary) × profile factor, to model the actual production profile over the contract period. Note that, because each contract period must be simulated at least twice, use of the DCQ adjustment option requires a corresponding increase in CPU time.

Control and output of sales gas rate.

Sales gas is what is left over after taking account of gas consumption at each group (known as gas fuel usage in ECLIPSE 300). For ECLIPSE 100, the sales gas rate can also include gas imported at a specified rate from outside the field. ECLIPSE normally applies the group or field gas rate targets to the **gross** production rate (that is the gas extracted from the formation). But when the Gas Field Operations model is in use, the targets are applied to the sales gas rate (that is the gas sent to shore). The sales gas rates and totals are included in the production reports, and can be output to the `SUMMARY` file.

Estimation of delivery capacity.

As well as modeling the actual performance of the field at the demanded rate, it is useful to have an idea of its delivery capacity. We define this to be the sales gas production rate that would ensue immediately after the removal of all group and field gas rate targets and limits, at the current grid block conditions. You can specify whether any group limits for other phases, and any rate limits on the wells themselves, should remain in operation. The delivery capacity is calculated at the start of each time step, and can be output to the `SUMMARY` file.

Automatic compressor operation.

ECLIPSE 100

When the Gas Field Operations model is used in conjunction with the Network option, gas compressors in the pipelines can be switched on and off automatically as required. The compressors may consume gas at a specified rate for power, which will be added to the gas consumption rate whenever the compressors are operating. The simplest option has two states: either “compression on” or “compression off”. A more versatile option switches on a series of compressors in a pre-defined sequence, and allows each compressor to have up to five levels of compression.

Seasonality profile generation

The engineer enters 12 profile factors, one for each month starting with January, using the keyword SWINGFAC. The simulation is then advanced one contract period at a time, using the keyword GASYEAR or GASPERIO instead of TIME, TSTEP or DATES. The contract period is assumed to start on the current date at which GASYEAR or GASPERIO is read. ECLIPSE flags an error if this is not the first day of a month. During the contract period, the monthly sales gas production rate target for the field is set equal to the DCQ multiplied by the month's profile factor. At the end of the contract period, ECLIPSE writes an output report and dump a RESTART file if required.

When the profile factor, and hence the production rate, changes from one month to the next, ECLIPSE automatically limits the time step to the end of the month. Thus ECLIPSE models the different flow rate periods individually, rather than taking longer time steps with an average rate. There is also an option to limit the time step to the end of every month irrespective of whether the rate changes or not. This ensures that there is at least one entry in the SUMMARY file for each month. In ECLIPSE 100, time steps limited to the end of the month can be identified by the mnemonic 'EMTH' in the PRINT file.

The field's production rate is constrained to meet the month's target by one of the two standard methods for group control - guide rate control or prioritization. By default, ECLIPSE automatically places the FIELD under guide rate control on reading the GASYEAR or GASPERIO keyword, and set its target gas rate each month from the DCQ and the swing or profile factors. You need to enter GCONPROD only if the field has limits on other phases, or if any groups have rate limits or guide rates. If on the other hand you wish the field's target rate to be met by prioritization instead of guide rate control, you must indicate this by entering the keyword GCONPRI. Place the FIELD under prioritization and give it a gas rate target. Any valid rate will do if you are going to use GASYEAR or GASPERIO, as the target is re-set at each month in the Contract Period to the DCQ multiplied by the month's swing or profile factor. The default priority coefficients must also be set with keyword PRIORITY. If at a later date you wish to revert to guide rate control, place the FIELD under 'GRAT' control in keyword GCONPROD, giving it a nominal gas rate target.

The profile factors can be changed, if required, at the start of any Contract Period by entering the keyword SWINGFAC again.

Normalizing the profile factors

The Annual Contracted Quantity (ACQ) is related to the Daily Contracted Quantity by:

$$ACQ = 365.25 \times DCQ \qquad \text{[EQ 24.1]}$$

The number 365.25 is the average number of days in a year, with a leap year every fourth year.

In order to produce the required ACQ for a given DCQ, the profile factors should be normalized such that the sum over all months of {profile factor x number of days in month} = 365.25, with February allocated 28.25 days. If this "normalization requirement" is obeyed, then when leap years are taken into account every fourth year, the average annual production at a constant DCQ is DCQ x 365.25, which corresponds with the definition of ACQ given above.

When the SWINGFAC keyword is entered, ECLIPSE checks that the profile factors obey this normalization requirement, and issue a warning if they do not.

Operations and reports during the contract period

The keyword GASYEAR or GASPERIO advances the simulation through one or more complete contract periods. Well operations can be performed in the usual manner at the end of a contract period, and a printed report is generated at the end of each contract period depending on the RPTSCHED or RPTPRINT switches. But in order to perform well operations or request printed reports during a contract period, an "Annual Scheduling File" must be set up.

To set up an Annual Scheduling File, enter the keywords defining the required operations before the GASYEAR or GASPERIO keyword on the input data file, bracketed with the keywords GASBEGIN and GASEND. When ECLIPSE reads the keyword GASBEGIN during the simulation, it copies the subsequent keywords up to GASEND to a separate file that it sets up automatically. The times within each year at which the operations should be performed are set with the keyword GASMONTH.

During each subsequent contract period, ECLIPSE reads the keywords from this file and performs the required operations at the appropriate times. The same operations will be performed each year, until you set up a new Annual Scheduling File. See the keyword GASBEGIN for further details of this facility.

Swing factor and DCQ adjustment

A flag in the keyword GASYEAR or GASPERIO can be set to allow the DCQ to be adjusted automatically at the start of each Contract Period to obey a Swing Factor Requirement, that is, that the field should be able to produce at a rate equal to the DCQ multiplied by the swing factor. If the field cannot produce at this rate, the DCQ is reduced to a value that enables the requirement to be obeyed.

If automatic DCQ adjustment is requested, each contract period is solved in two passes.

- The first pass is a test calculation, which has the purpose of deciding how much the DCQ needs to be reduced in order to obey the swing requirement for the current contract period. If requested, ECLIPSE may perform a number of iterations of the first pass, with different DCQ values, to determine the highest DCQ that obeys the swing requirement for the period, to a specified accuracy.
- The second pass simulates the actual performance of the field, with the sales gas rate target set each month to the new value of the DCQ multiplied by the monthly Profile Factor. The RESTART and SUMMARY files are written as required during this pass.

The swing factor is specified separately for each month for flexibility. The monthly swing factors are entered along with the profile factors using the keyword SWINGFAC. For example, the monthly swing factors could be set equal to the profile factors, in which case the Swing Factor Requirement causes the DCQ to be adjusted each period so that the seasonality profile fits inside the delivery capacity decline curve. Alternatively the swing factors could all be set to the same value each month. A check is made to ensure the profile factor does not exceed the swing factor during any month, otherwise obeying the Swing Factor Requirement may not guarantee the field could produce its target rate profile.

The mnemonic FGDCQ (which stands for Field Gas DCQ) outputs the current value of the DCQ to the SUMMARY file.

ECLIPSE 100

By default, ECLIPSE 100 only adjusts the DCQ downwards. If the field has more capacity than is required to meet the swing requirement with the current value of the DCQ, ECLIPSE 100 does not automatically increase the DCQ. If you wish to increase the DCQ during a simulation, this may be done manually by resetting the initial DCQ in item 2 of the GASYEAR keyword (or item 3 of GASPERIO) at the appropriate time. Alternatively, the accelerated iteration scheme may be used by setting item 2 in the GASFIELD keyword. This permits the DCQ to increase again if appropriate, up as far as its initial value.

ECLIPSE 300

ECLIPSE 300 has the accelerated scheme enabled by default, and so after any initial reduction, the DCQ may increase up as far as its initial value should the gas delivery capacity of the field permit this. This scheme can not be switched off in ECLIPSE 300.

Interpretations of the swing requirement

To provide flexibility to handle different forms of contract, there is a choice of interpretations of the swing requirement. The chosen interpretation is selected by entering its associated mnemonic as Item 3 in keyword GASYEAR or Item 4 in keyword GASPERIO. If the contract period is less than a year, keyword GASPERIO must be used.

The choices within GASYEAR for contract years are:

- YEAR or YES

  "The field must be able to produce at the DCQ multiplied by the monthly swing factor, for the whole contract year."

  During the first pass, the field is produced with a target rate of DCQ × SWING for the whole contract year. If this target cannot be met, ECLIPSE determines how much the DCQ should be reduced in order to meet the target.

  This is a suitable interpretation for cases where the monthly swing factors are the same as the profile factors, as it ensures the production profile fits underneath the delivery capacity decline curve. But for cases where the swing factors exceed the profile factors, this interpretation may be too stringent because it would require the field to be able to produce more than the Annual Contracted Quantity.

- PRO

  "The field must be able to raise its production rate from DCQ × profile to DCQ × swing for an instant at any time in the contract year."

  During the first pass, the field is produced with a target rate of DCQ × PROFILE for the whole contract year. In addition, the delivery capacity is calculated and compared against a target of DCQ × SWING. If either of these targets are not met, ECLIPSE determines how much the DCQ should be reduced in order to meet them both. (The delivery capacity is calculated using the grid block conditions at the beginning of the time step, that is, at the end of the previous time step. Therefore it is compared against the DCQ × swing factor for the previous time step.)

  This interpretation ensures the field is able to produce its ACQ (that is, DCQ × profile for each month), and has the potential to increase its production rate to DCQ × swing for a short period. But it does not test whether the increased production rate can be sustained for any length of time.

- ACQ

  "The field must be able to produce at the DCQ multiplied by the monthly swing factor continuously until it has produced its Annual Contracted Quantity."

During the first pass, the field is produced with a target rate of DCQ × SWING, but only until it has produced its ACQ (= 365.25 × DCQ). ECLIPSE simulates through the contract year until the cumulative sum of {monthly swing factor × number of days in the month} equals 365.25. When accumulating this sum, February is allocated 28.25 days. If the target of DCQ × SWING cannot be met, ECLIPSE determines how much the DCQ should be reduced in order to meet the target.

This interpretation ensures the field is able to produce its ACQ, and would be capable of operating at the increased rate of DCQ × swing factor continuously until its ACQ had been produced.

- `JAN`, `FEB`, `MAR`, `APR`, `MAY`, `JUN`, `JUL`, `AUG`, `SEP`, `OCT`, `NOV`, `DEC`

  The field must be able to produce at the DCQ multiplied by the monthly swing factor continuously until it has produced a specified portion of its Annual Contracted Quantity".

  This is similar to the `ACQ` option above, but the first pass is only calculated until a specified portion of the ACQ has been produced. The portion is defined as

  ACQ × sum over months from start of contract year to end of specified month of {profile factor × number of days in month}

  ÷ sum over all months in the contract year of {profile factor × number of days in month}.

  When accumulating these sums, February is allocated 28.25 days.

- `NO`

  No automatic adjustment of DCQ.

  The DCQ is held constant at its current value.

  The first pass of the contract year is omitted, and ECLIPSE goes immediately into the second pass.

In all options, the second pass of the contract year is calculated with the field producing at the (perhaps reduced) DCQ × PROFILE.

The choices within `GASPERIO` for general contract periods are:

- `PER` or `YES`

  The field must be able to produce at the DCQ multiplied by the monthly swing factor, for the whole contract period.

- `PRO`

  "The field must be able to raise its production rate from DCQ × profile to DCQ × swing for an instant at any time in the contract period."

- `PCQ`

  "The field must be able to produce at the DCQ multiplied by the monthly swing factor continuously until it has produced its contracted quantity for the period (= no. of days in the period × DCQ)."

  In this option February has either 28 or 29 days depending on whether or not it is a leap year.

- `NO`

  No automatic adjustment of DCQ.

Calculating the DCQ value

The DCQ value is calculated in the first pass of each contract period. As the field depletes, the DCQ must be reduced each period in order to obey the swing requirement. Several iterations of the first pass of the contract period may be needed to provide an accurate value for the largest DCQ able to meet the swing requirement, if the DCQ reduces substantially from the previous period. The reason is as follows. If the initial estimate of the DCQ is too high, too much gas is extracted during the first pass, making the delivery capacity decline faster than it should. The calculated value of the DCQ that can fit under this steeper decline curve is therefore an under-estimate. Further iterations of the first pass are needed to obtain a more accurate answer.

An "accelerated iteration scheme" is available for converging the DCQ to a desired accuracy. An initial guess is obtained, based on the previous period's DCQ reduction. The first pass of the contract period is simulated and a new DCQ is calculated. Successive iterations of the first pass are performed until the change in DCQ over an iteration lies within a user defined tolerance. The convergence is accelerated with a secant method. A maximum number of iterations can be imposed (item 7 in keyword GASYEAR or item 8 in GASPERIO), and a warning is issued if the DCQ has not converged by then. The accelerated iteration scheme is the only scheme available in ECLIPSE 300. For ECLIPSE 100 however, it must be enabled by setting the 2nd item of the RUNSPEC section keyword GASFIELD to 'YES'. This is the recommended method, although to keep back-compatibility with pre-96A ECLIPSE 100 simulation runs by default the accelerated scheme is not enabled.

ECLIPSE 100

If the accelerated iteration scheme is **not** selected, the pre-96A scheme is employed. In this scheme, the DCQ can only decrease from one iteration to the next, and under-estimates cannot be corrected. Thus, to obtain an accurate answer, the amount by which the DCQ can decrease from one iteration to the next should be constrained. You can supply a limiting reduction factor (item 5 in keyword GASYEAR, item 6 in GASPERIO). If at the end of the first pass the DCQ has to be reduced to a fraction that is smaller than the limiting reduction factor, it is reduced only by the limiting reduction factor and the first pass is repeated again. The process continues until either the DCQ reduction lies within the limiting factor, or the maximum number of iterations has been reached (GASYEAR item 7, GASPERIO item 8) at which the DCQ reduction is no longer restricted. The nearer the reduction factor is to 1.0 the more accurate is the result, but at the expense of requiring more iterations of the first pass to converge it. Supplying an "anticipated reduction factor" (GASYEAR item 6, GASPERIO item 7) could help reduce the number of iterations required to converge the DCQ, by bringing the starting value nearer to the final value. Before the decline from plateau has begun, this factor is ignored. But once the DCQ has been reduced automatically below its initial value, the anticipated reduction factor is applied to the DCQ at the start of each contract period. Of course, you ought to have a reasonable idea of the rate of DCQ reduction before using this option. An anticipated reduction that is too fast results in unduly pessimistic DCQs, since ECLIPSE does not increase the DCQ automatically. A warning message is printed if the DCQ has not been reduced by more than the anticipated rate at the end of a contract period, as this would imply that the anticipated rate is too fast.

Control and output of sales gas rate

ECLIPSE 100

In ECLIPSE 100 the sales gas rate is defined as the **gross** gas production from the formation, **minus** the gas consumption, **plus** any gas imports. Gas can be consumed or imported at a specified rate at any group, as requested using the keyword GCONSUMP. Gas can also be consumed by an automatic compressor when it is operating (see "Automatic compressors" on page 367).

ECLIPSE 300

In ECLIPSE 300 the sales gas rate is defined as the **gross** gas production from the formation, **minus** the gas consumption. Gas can be consumed as fuel using the keyword GRUPFUEL. Gas may only be imported via a gas producing satellite group (see keyword GSATPROD). Note that advanced and make-up gas (see keywords GADVANCE and GINJGAS) is only available for re-injection and thus does not alter the sales gas rate.

Normally the group and field gas rate targets apply to the gross production rate of gas from the formation. But when the Gas Field Operations model is in use (triggered by the keyword SWINGFAC), these targets are automatically adjusted to allow for the specified consumption and, for ECLIPSE 100, import rates. (Gas injection is assumed not to occur, so no adjustment is made for it.) Thus the targets now apply to the sales gas rate, whether they are set manually using keyword GCONPROD or GCONPRI, or set automatically according to the DCQ.

ECLIPSE translates a sales gas target for a group into a gross gas target by adding the rate of gas consumption at and below the group and, for ECLIPSE 100, subtracting any gas import rate at and below the group. For ECLIPSE 100, the gas consumption rate may depend on whether or not the automatic compressors are operating. The gas consumption rate (with or without compression) must be a known fixed value. Thus the option to make the consumption rate a given fraction of the production rate (a negative value in item 2 of keyword GCONSUMP or a non-zero value in item 3 of keyword GRUPFUEL) is not allowed when running the Gas Field Operations model.

The production reports show the sales gas rate, and also the gross gas rate and consumption/import for any group that has gas consumption or import. The sales gas rates and totals can be output to the SUMMARY file using the mnemonics

FGSR	Field Gas Sales Rate
GGSR	Group Gas Sales Rate
FGST	Field Gas Sales Total (cumulative)
GGST	Group Gas Sales Total (cumulative)

The mnemonics FGPR, GGPR, FGPT, GGPT retain their original meanings as the field and group **gross** gas production rates and totals.

Do not use the keywords GCONSALE or GRUPSALE. These apply to the control of gas production from an oil field by re-injecting the surplus gas. It is not appropriate to do this when running the Gas Field Operations model.

Estimation of delivery capacity

At the start of each time step, ECLIPSE calculates the delivery capacity of the field. The delivery capacity is defined as the sales gas production rate that would ensue immediately after the removal of all group and field gas rate targets and limits, at the current grid block conditions. You can specify (with the keyword GASFDELC) whether any group or field oil or water rate limits, and any rate limits on the wells themselves, should remain in operation.

The delivery capacity is calculated at every time step after the first entry of the SWINGFAC keyword (which effectively turns this facility on), irrespective of whether GASYEAR, GASPERIO, DATES, TIME or TSTEP is used to advance the simulation. When the option for automatic DCQ adjustment is in use (see "Swing factor and DCQ adjustment" on page 362) the delivery capacity is calculated during the second pass of the contract period. In ECLIPSE 100 the delivery capacities of the field and each group are printed in the Production Reports in the right hand column. The well delivery capacities are also printed in ECLIPSE 100 if their

drainage radii are set negative in Item 7 of keyword WELSPECS (otherwise their productivity indices will be printed instead). The group and well delivery capacities are respectively the sales gas rate of each group and the gas production rate of each well when the field operates at its delivery capacity. The field, group and well delivery capacities can also be output to the summary file using the mnemonics FGDC, GGDC and WGDC.

The delivery capacity calculated at each time step is not exact; in fact it is somewhat optimistic. There are two reasons for this. Firstly, it is calculated at the start of the time step, using the grid block conditions existing at that time. It does not take into account the changes in grid block pressure and saturation that occur during the time step. Thus at the end of the time step (when the delivery capacity is in fact reported) it is actually smaller than the value calculated at the start of the time step. Secondly, the grid block conditions used in the calculations are those that exist when the field produces at the required offtake rate, not when it produces at full capacity. If the offtake rate were actually to increase to full capacity, the grid block pressures would decrease and thus decrease the delivery capacity. In fact, you may notice the delivery capacity increasing slightly when the offtake rate decreases in spring and summer. This is due to the build-up of grid block pressure that follows a decrease in production rate. The rate-dependence of the reported delivery capacity is more pronounced at low permeabilities, or if the grid blocks containing the well connections are small. The delivery capacity should therefore be regarded as an instantaneous transient value rather than a sustainable one.

ECLIPSE 100

When the Gas Field Operations model is used in conjunction with the Network option, the delivery capacity calculation takes account of the higher flows along the network branches when the field is producing at full capacity. Thus the network is balanced twice at the start of each time step: first with the field operating at full capacity, then with the field producing at the required offtake rate. If a balancing interval > 0.0 is entered in keyword NETBALAN it is ignored, because the network has to be balanced at each time step.

The delivery capacity should not be confused with the production potential. The potential of a production well is defined as the flow rate it would immediately have if, at the current grid block conditions, its only constraints were its THP and BHP limits. The field and group potentials are just the sum of the well potentials multiplied by their efficiency factors. The main differences between the delivery capacity and the potential are

- A well's production potential honors only its THP and BHP limits. The delivery capacity will honor all the well's rate constraints too, unless keyword GASFDELC has been used to disable them.
- The group production potential takes no account of any limits on group flow rate. The delivery capacity honors all group and field limits except on the gas production rate, unless keyword GASFDELC has been used to disable them.
- The delivery capacity is adjusted to allow for gas consumption and, in ECLIPSE 100, gas imports, while the potential is not.

ECLIPSE 100

- When using the Network option, the delivery capacity is calculated with the network balanced at the capacity flow rates. The potential is calculated using the THP limits obtained from the nodal pressures of the network balanced at the required offtake rate. Thus the potentials do not account for the increased wellhead pressures that would result when the flows increase to capacity.

Automatic compressors

ECLIPSE 100

The Network option can be used with the Gas Field Operations model to take account of the pipeline pressure losses in the gathering system (see "The Network Option" on page 681). The delivery capacity is determined by the need to supply the gas at a specified onshore pressure.

Compressors can be placed in the pipelines to boost the gas flow. In the Network option, the effects of turning on a compressor can be modeled either by switching to an alternative VFP table to calculate the pressure losses in the pipeline, or by changing the value of the Artificial Lift Quantity that is used to interpolate the table. One compressor is allowed in each branch of the network.

When the Network option and the Gas Field Operations model are used together, a class of compressors called "automatic compressors" can be declared. The basic compressor facility available with the Standard Network model (keyword GNETPUMP) is not versatile enough for the requirements of the Gas Field Operations model, and you must use the keyword GASFCOMP instead. This defines a more versatile compressor facility, which was designed specifically for the Gas Field Operations model. However, the automatic compressor facilities associated with the "Extended Network model" (keyword NETCOMPA) **are** suitable for use with the Gas Field Operations model. Thus, when using the Extended Network Model, automatic compressors should be defined with keyword NETCOMPA instead of GASFCOMP. In either of the two cases, ECLIPSE switches on compression whenever necessary to enable the field to meet its target rate. Compression is turned off when not needed (for example, during the summer "low" in the seasonality profile).

Each compressor, when operating, can consume a specified rate of gas for power. The gas consumption rate of the automatic compressors is set in the compressor data keyword (GASFCOMP or NETCOMPA). Whenever the automatic compressors are operating, their gas consumption rate is added to the fixed consumption rate (if any) specified in keyword GCONSUMP, and subtracted from the gross production to give the sales gas rate.

Simple and extended compressor options with GASFCOMP

This section describes the options available for compressors defined with keyword GASFCOMP when using the Standard Network model. If you are using the Extended Network model, you should define the compressors with keyword NETCOMPA instead. These compressors have common functionality with the extended compressor option described below. See the chapter on the Network option for details of the Extended Network model and its associated compressor keyword NETCOMPA.

The simple option for automatic compressors has just two states: "all compressors on" and "all compressors off". The compressors are turned on whenever the field cannot meet its target rate without them, and turned off whenever the target rate decreases to see if the field can meet it without them. The automatic compressors are also turned on while calculating the field's delivery capacity.

The extended compressor option offers a more versatile set of facilities, firstly by giving each compressor up to five distinct levels of compression, and secondly by allowing them to be turned on or switched to a higher compression level individually in a predefined order until the field can make its production target. The parameters of each compression level (VFP table number, ALQ value and gas consumption rate) for each compressor are entered in separate records of keyword GASFCOMP.

In the extended compressor option, individual compressors are turned on, or switched to a higher level of compression, in a predefined sequence whenever necessary to maintain the production target. Each compression level of each compressor is given an "action sequence number", specified by Item 6 in the GASFCOMP keyword. A particular sequence number can be assigned to the compression levels of more than one compressor, if required, but the levels of any one compressor must all have different sequence numbers. When the field cannot make its target, the actions with sequence number 1 are performed first, that is, all compression levels having an action sequence number 1 are turned on. If the target still cannot be met, compression

levels with action sequence number 2 are turned on, and so on until the target is met or no more compressors can be switched to a higher compression level. In common with the simple compressor option, all compressors are turned off whenever the target rate decreases, to see if the target can be met without them. Also in analogy with the simple option, all automatic compressors operate at their highest compression level while ECLIPSE is calculating the field's delivery capacity.

To keep track of the state of each compressor, there is a SUMMARY section mnemonic GMCPL that writes the current compression level of the compressor in each group's pipeline to the SUMMARY file. The compression level is represented in the SUMMARY file by a number between 0 (for no compression) and 5 (for the highest allowed level).

The extended compressor option requires a small amount of extra storage, which must be reserved by setting the first switch in the GASFIELD keyword in the RUNSPEC section to YES.

Specifying production rates for the second pass using GASFTARG and GASFDECR

ECLIPSE 100

The GASFTARG keyword can be used to specify a rate at which the field is to be produced during the second pass of the Gas Field Operations Model, independent of the calculated DCQ.

The GASFTARG keyword is used in conjunction with SWINGFAC keyword. During the first pass of the model the DCQ will be calculated using the swing factors provided. However, on the second pass the field is produced at the lesser of DCQ × PROFILE and the rate specified in the GASFTARG keyword. For months defaulted in the GASFTARG keyword the field is produced according to the PROFILE factors.

The GASFDECR keyword enables the field gas production rate for the second pass to be adjusted through monthly decrements. On the second pass the field is then produced at DCQ × PROFILE - DECREMENT.

The GASFTARG and GASFDECR keywords are restricted to single contract simulations.

Applying separate gas supply contracts to individual groups

The Gas Field Operations model by default applies a single set of swing factors and profile factors to the FIELD group at the top of the grouping hierarchy. There is, however, an option to allow individual groups to have their own independent gas supply contracts. Each "contract group" has its own set of swing factors and profile factors, and its own DCQ which ECLIPSE can adjust independently to meet its swing requirement.

Defining multiple contract groups

ECLIPSE 100

In order to use the Multiple Contract Groups facility, the flag 'YES' must be set in item 1 of the RUNSPEC section keyword GASFIELD. This reserves the extra storage needed by various extensions of the Gas Field Operations model. This step is not necessary in ECLIPSE 300.

To define multiple contract groups, use the keyword GSWINGF instead of the SWINGFAC keyword to activate the Gas Field Operations model. The GSWINGF keyword is used to identify the contract groups and to give each one its own set of swing and profile factors. (In comparison, the SWINGFAC keyword reads in a single set of swing and profile factors, and applies it to the FIELD group.)

The set of contract groups must obey the following restrictions:

- One contract group must not be subordinate to another contract group.
- The set of contract groups must cover the whole field, that is, every open production well must be subordinate to a contract group.
- There must be no gas consumption or, for ECLIPSE 100, import at any node downstream of a contract group (otherwise the contract group's sales gas targets would not be set correctly).

Simulating with multiple contract groups

The keyword GASYEAR or GASPERIO is still used to advance the simulator through an integral number of contract periods. ECLIPSE assumes that all contract groups start their contract periods at the same time, and the contract period length is the same for each group. However, each contract group generally has a different DCQ. So item 2 in keyword GASYEAR or item 3 in GASPERIO (the initial DCQ) is ignored, and instead the initial DCQ of each contract group must be entered with the keyword GDCQ. The remaining items in keyword GASYEAR or GASPERIO are applied to all contract groups.

When you set the initial DCQ of each contract group with keyword GDCQ, you should state whether the DCQ is variable or must remain fixed at the specified value. Variable DCQs are adjusted automatically as necessary to obey the group's swing requirement, depending on the option selected in item 3 of keyword GASYEAR or item 4 of GASPERIO. Fixed DCQs remain at their specified values until reset in a subsequent GDCQ keyword.

Fixed-DCQ contract groups

If gas produced by other operators is fed into the same network that you use, you may wish to determine the DCQ which your group is able to supply against assumed production profiles from the other operators' groups. In this situation, you could model the other operators' groups as fixed-DCQ contract groups, and your own group as a variable-DCQ contract group. The group DCQ values can be written to the SUMMARY file with the SUMMARY section mnemonic GGDCQ (Group Gas DCQ).

When there is only one variable-DCQ group in the problem, the simulation process is exactly the same as for the single contract group case described in Section 2, except that the swing requirement and DCQ adjustment is applied to the variable DCQ group instead of the FIELD, and gas from the other fixed-DCQ groups also enters the network. All the options in item 3 of the GASYEAR keyword or item 4 of GASPERIO are available for the variable-DCQ group.

The flow targets applied to the fixed-DCQ groups are equivalent to the target rate applied to the variable-DCQ group. In the first pass of the contract period each group has a target rate equal to its DCQ × Swing Factor; or its DCQ × Profile Factor if the 'PRO' option is selected. In the second pass of the contract period, the flow target is its DCQ × Profile Factor. But in contrast to the variable-DCQ group, if a fixed-DCQ group cannot achieve its flow target its DCQ is not reduced. A missed flow target indicates that its specified DCQ is too high, so you should either give it a lower DCQ or model it as a variable-DCQ group. The group sales gas rates and delivery capacities can be written to the SUMMARY file with the mnemonics GGSR and GGDC.

Fixed-DCQ groups have two choices for calculating delivery capacities. The first choice (FIX1) has each contract group operating at its full delivery capacity. The delivery capacity of the variable-DCQ contract group is therefore determined at a time when the other groups are also producing at their full capacities into the network. The second choice (FIX2) imposes a maximum rate of DCQ × Swing Factor on each fixed-DCQ group. Then the delivery capacity of the variable-DCQ contract group will be determined at a time when the other groups are producing at DCQ × Swing. The network pressures in the FIX2 case are lower than those in the FIX1 case, so the delivery capacity of the variable-DCQ group is higher. Again, you should ensure that the fixed-DCQ groups are able to make their delivery capacity targets, as otherwise it implies that their specified DCQs are too high.

Multiple variable-DCQ contract groups

When you wish to predict the future DCQs of two or more groups with independent gas supply contracts, you should model them as variable-DCQ contract groups. ECLIPSE will test each variable-DCQ contract group against its own swing requirement, and reduce its DCQ as necessary. The DCQ values of the contract groups can be written to the SUMMARY file with the SUMMARY section mnemonic GGDCQ (Group Gas DCQ).

The simulation method for multiple variable-DCQ contract groups is equivalent to the method for a single contract at the FIELD group. If automatic DCQ reduction is requested in keyword GASYEAR or GASPERIO, each contract period is solved in two passes. During the first pass, the contract groups aim to produce at their DCQ × Swing Factor. If a contract group cannot make its swing requirement, its DCQ is reduced accordingly. During the second pass of the contract period, the contract groups have their flow targets set equal to their DCQ values multiplied by their monthly profile factors. The group rate targets are met by Guide Rate Control, unless Prioritization is requested with the keyword GCONPRI.

When there is more than one contract group with variable DCQ, item 3 in keyword GASYEAR or item 4 in GASPERIO (the swing requirement) is restricted to two choices:

- YEAR or PER or YES

For automatic DCQ reduction, with the production target set at DCQ × swing factor for the whole contract period in the first pass

- NO

  For no automatic DCQ reduction.

The other options (ACQ, PRO etc.) are disabled to avoid problems with the contract groups interacting through the network. (The delivery capacity of each group varies with the flow rates of the other groups, so the flow rates of all the contract groups must be defined for the whole contract period while the swing requirement is being tested.) The YEAR or PER option is most suitable for cases where the monthly swing factors are the same as the profile factors. For cases where the swing factors exceed the profile factors, you could reduce them to the profile factor value in all but two months - at the end of the peak flow period and at the end of the contract period.

The delivery capacity is calculated in the usual way, by removing the constraints on gas production rate. The delivery capacity reported for the individual contract groups will be their production capacity while all other groups are producing at their full capacity. Because of the coupling through the network, a group may possibly achieve a higher rate than its reported delivery capacity when the others are producing at the smaller rate of DCQ × Profile.

Automatic compressors with multiple contract groups

ECLIPSE 100

When there are multiple contract groups, compressors may be situated either at (or subordinate to) specific contract groups, or further downstream at a node common to all contract groups. If a contract group G1 fails to meet its target, then turning on a compressor either at G1 or at a common node helps, but turning on a compressor located within a group that is still meeting its target does not help. Thus, when deciding which compressor to turn on next with the Extended Compressor option (keyword GASFCOMP), ECLIPSE ignores any compressors located within contract groups that are still meeting their target. Note that compressors located at a common node cannot have a gas consumption rate set, as gas consumption is not allowed downstream of a contract group.

Using the gas field operations model

RUNSPEC section

ECLIPSE 100

We recommend you use the accelerated iteration scheme for calculating the DCQ. Accordingly, in the RUNSPEC section enter the GASFIELD keyword with item 2 set to YES. Set item 1 to YES if you are using either multiple contract groups (keyword GSWINGF) or the extended compressor option in keyword GASFCOMP.

```
GASFIELD
 YES   YES /
```

SUMMARY section

Useful quantities to include in the SUMMARY section are:

Table 24.1 Useful gas field operations SUMMARY output controls

E100	E300	Field	Group	Other	Information
x	x	FGSR	GGSR		Field/Group Gas Sales Rate
x	x	FGST	GGST		Field/Group Gas Sales Total (cumulative)
x	x	FGDC	GGDC		Field/Group Gas Delivery Capacity
x	x	FGDCQ	GGDCQ		Field/Group Gas DCQ
x			GMCPL		Group Multi-level Compressor Level
x	x		GPR		Group or node PRessure in network
x			GPRDC		Group or node PRessure in network at Delivery Capacity
x	x			NGOPAS	Number of iterations to converge DCQ in first pass
x				WGPRFP	Well Gas Production Rate from end of First Pass
x				WTHPFP	Well Tubing Head Pressure from end of First Pass
x				WBHPFP	Well Bottom Hole Pressure from end of First Pass
x			GPRFP		Group or node PRessure in network from end of First Pass
x			GGPRNBFP		Gas flow rate along Group's or node's outlet branch in network, from end of First Pass

Remember that the mnemonics FGPR, GGPR, FGPT, GGPT refer to the **gross** gas produced from the formation, without allowing for consumption. Note also that the mnemonics WGPRFP, WTHPFP, WBHPFP, GPRFP, GGPRNBFP do not refer to continuous quantities but rather the values of these quantities at the end of the first pass.

Specifying the wells

Specify the wells in the usual way with WELSPECS, COMPDAT and WCONPROD (and WCONINJE for any water injectors). The optional keywords (for example WECON, WEFAC) are not mentioned explicitly in this example.

If there is a multi-level grouping hierarchy (more than one level of groups below the FIELD), set it up with GRUPTREE. For the purposes of this example, consider a grouping hierarchy as given by:

```
GRUPTREE
--  CHILD     PARENT
    PLAT-A    FIELD   /
    PLAT-B    FIELD   /
    G-B1      PLAT-B  /
    G-B2      PLAT-B  /
/
```

Setting up the network

If the Standard Network model is to be used, set up the network with GRUPNET. If using the Extended Network model, set up the network with keywords BRANPROP and NODEPROP instead. For this example we use:

```
GRUPNET
-- GROUP    FIXED VFP ALQ S-SEA LIFT
-- NAME     PRESS TAB     MAN   GAS
   FIELD    500                        /
   PLAT-A   1*    1                    /
   PLAT-B   1*    2                    /
   G-B1     1*    3                    /
   G-B2     1*    4                    /
/
```

NETBALAN may be used to alter the convergence tolerances and iteration limits, and set restrictions on the allowed balancing error at the end of each time step. But the balancing interval set here is ignored because when the Gas Field Operations model is in use, the network is balanced at the beginning of every time step.

ECLIPSE 100

Declare any automatic compressors with GASFCOMP (or NETCOMPA if using the Extended Network model). Do not use GNETPUMP. For example, to place a single-level compressor in the pipeline from PLAT-A in the Standard Network model:

```
GASFCOMP
-- GROUP      NEW VFP     NEW         GAS         COMP       SEQUENCE
-- NAME       TAB NO      ALQ       CONSUMP       LEVEL       NUMBER
   'PLAT-A'     1*         50         1000    /
/
```

ECLIPSE 100

Alternatively, to declare a pair of two-level compressors in the pipelines from PLAT-A and PLAT-B in the Standard Network model, and have them turned on in sequence:

```
GASFCOMP
-- GROUP      NEW VFP     NEW         GAS        COMP      SEQUENCE
-- NAME       TAB NO      ALQ       CONSUMP      LEVEL      NUMBER
  'PLAT-A'      1*         50        1000         1           1    /
  'PLAT-A'      1*        100        1500         2           2    /
  'PLAT-B'      1*         50        1000         1           3    /
  'PLAT-B'      1*        100        1500         2           4    /
/
```

This uses the extended compressor option, so remember to enter 'YES' in item 1 of keyword GASFIELD in the RUNSPEC section.

Group control instructions

If you are going to use GASYEAR or GASPERIO to advance the simulation, ECLIPSE will calculate the field's gas production target each month from the DCQ and the swing or profile factors. By default, ECLIPSE automatically places the FIELD under guide rate control to meet this target. You need to enter GCONPROD only if the field has limits on other phases, or if any groups have rate limits or guide rates. If on the other hand you wish the field's target rate to be met by prioritization instead of guide rate control, you must indicate this by entering the keyword GCONPRI. Place the FIELD under prioritization and give it a gas rate target. Any valid rate will do if you are going to use GASYEAR or GASPERIO, as the target is re-set at each month to the DCQ multiplied by the month's swing or profile factor. The default priority coefficients must also be set with keyword PRIORITY.

For example, to use prioritization where the lowest water-gas ratio wells have the highest priority, with well priorities being updated after 100 days:

```
PRIORITY
 100    0.0  0.0  0.0  1.0    0.0  0.0  1.0  0.0  /

GCONPRI
-- GROUP    OIL   PROC   WAT   PROC   GAS   PROC   LIQ   PROC   RESV
-- NAME     LIM   OIL    LIM   WAT    LIM   GAS    LIM   LIQ    LIM
   'FIELD'   4*                        1.0  'PRI'                  /
/
```

ECLIPSE 100

If any groups have a fixed rate of gas consumption or import (independent of the automatic compressors), this can be entered using GCONSUMP:

```
GCONSUMP
-- GROUP    CONS    IMPRT
-- NAME     RATE    RATE
  'PLAT-*'  5000          /
/
```

If any groups have a fixed rate of gas fuel usage, this can be entered using GRUPFUEL:

```
GRUPFUEL
-- GROUP     CONS
-- NAME      RATE
  'PLAT-*'   5000 /
/
```

Advancing the simulation

The Gas Field Operations model is turned on when the keyword SWINGFAC is entered. The keyword is used to set the monthly swing factors and profile factors for the FIELD. For example, consider a case where the swing factors are the same as the profile factors, which are in ratio 1.0, 1.5, 1.0 and 0.5 for Autumn, Winter, Spring and Summer respectively. Their normalized values are 1.0024, 1.5036, 1.0024 and 0.5012. Since the swing and profile factors are the same, the swing factor requirement is equivalent to checking that the seasonality profile fits inside the delivery capacity decline curve.

```
SWINGFAC
-- JAN   FEB   MAR   APR   MAY   JUN   JUL   AUG   SEP   OCT   NOV   DEC
   2*1.5036  3*1.0024        3*0.5012        3*1.0024        1.5036
   2*1.5036  3*1.0024        3*0.5012        3*1.0024        1.5036 /
```

Once the SWINGFAC keyword has been entered, the features labelled 3, 4 and 5 in the Introduction are activated. This means that all group and field gas targets will refer to sales gas, the delivery capacity will be calculated at each time step, and any automatic compressors are turned on and off as required.

If you wish to disable all rate limits for wells and groups when calculating the delivery capacity, the keyword GASFDELC must be used to indicate this requirement. The delivery capacity is then the sum of the well production rates when operating at their BHP or THP limits, minus any gas consumption.

```
GASFDELC
 2 /
```

To use features 1 and 2, that is, seasonality profile generation and DCQ adjustment, the keyword GASYEAR or GASPERIO must be used instead of TIME, TSTEP or DATES. This advances the simulation through an integral number of contract periods, starting at the current date (which must be the first day of a month). GASYEAR may be used if the contract period is a year, otherwise GASPERIO must be used. At the end of each contract period, reports are printed and restarts are written as requested with RPTSCHED or RPTPRINT. Reports can also be printed during the contract period if an Annual Scheduling File has been set up requesting them. For example, to request a printed report for the time step ending 1st March each year:

```
GASBEGIN
GASMONTH
-- MONTH  REPORT
   'MAR'   'YES'  /
GASEND
```

The Annual Scheduling File can also contain keywords defining well operations to be performed at the same time each year. More detailed examples can be found in the documentation for keywords GASBEGIN and GASMONTH.

With the first entry of the GASYEAR or GASPERIO keyword, the initial DCQ must be specified. For example, to advance the simulation through four contract years, with the DCQ initially 150,000 but reduced when necessary to obey the swing requirement:

```
GASYEAR
--
 No of   Initial  Swing  Monthly   Lim DCQ   Ant DCQ   Max no   Conv
--
 years     DCQ      req   t/steps    reduc     reduc    DCQ its  tol
      4     150000   'YEAR'   /
```

The interpretation of the swing requirement most appropriate for cases where the swing factors are the same as the profile factors has been chosen (this is in fact the default option). Defaulting the remaining items forces a new time step to start on the first of each month, applies no limit to the DCQ reduction in a single first pass, and applies no anticipated reduction factor to the DCQ at the start of a new contract year.

If you wish to model separate gas supply contracts for two or more groups, instead of a single contract for the FIELD, you must use the keyword GSWINGF instead of SWINGFAC. GSWINGF identifies the groups that have gas supply contracts, and sets their individual swing and profile factors. You must also set the initial DCQs of the contract groups individually, using keyword GDCQ. Keyword GASYEAR or GASPERIO is still used to advance the simulation through one or more contract periods, but the initial DCQ in item 2 or 3 will be ignored, so leave it defaulted. For example, to specify swing and profile factors and initial DCQs of two contract groups, PLAT-A and PLAT-B, and advance the simulation through four contract years, reducing their DCQs as necessary to obey their swing requirements:

```
GSWINGF
--
 GROUP    JAN  FEB  MAR  APR  MAY  JUN  JUL  AUG  SEP  OCT  NOV  DEC
  'PLAT-
A'  2*1.5036  3*1.0024       3*0.5012       3*1.0024       1.5036
            2*1.5036  3*1.0024       3*0.5012       3*1.0024       1
.5036 /
  'PLAT-
B'  2*1.6551  3*1.0031       3*0.3511       3*1.0031       1.6551
            2*1.6551  3*1.0031       3*0.3511       3*1.0031       1
.6551 /
/
GDCQ
-- GROUP     INITIAL   VAR/FIX
-- NAME        DCQ       DCQ
  'PLAT-A'    60000     'VAR'  /
  'PLAT-B'    40000     'VAR'  /
/
```

```
GASYEAR
--
 No of   Initial   Swing   Monthly   Lim DCQ   Ant DCQ   Max no    C
onv
--
 years     DCQ      req    t/steps    reduc     reduc    DCQ its   t
ol
      4       1*       'YEAR'   /
```

ECLIPSE 100

If there are multiple contract groups, remember to enter 'YES' in item 1 of keyword GASFIELD in the RUNSPEC section.

Examining the results

The set of output reports requested in RPTSCHED is written to the print file at the end of each contract period. Output reports can also be written in the middle of a contract period if requested in an Annual Scheduling File. Summary information is written to the SUMMARY file each time step as usual (only during the second pass, of course, if the automatic DCQ adjustment option is on).

The best way to plot the results in GRAF is in the form of histograms. Figure 24.1 shows the results of the example discussed above. The contract year starts on 1st September, and the initial DCQ is 150,000 Mscf/Day. With this DCQ, the annual production profile fits inside the delivery capacity decline curve for the first two years. During the third year, however, the field cannot deliver at the winter peak rate. ECLIPSE has to reduce the DCQ to 96,000 Mscf/Day to obey the swing factor requirement. For the fourth year, the DCQ is reduced to 59,000 Mscf/Day. The DCQ reduction in the third and fourth years turns out to be very substantial, and for greater accuracy the engineer would be recommended to apply a DCQ reduction limit or use the accelerated iteration scheme with a suitable convergence tolerance.

Figure 24.1 Sales Gas Production Rate and Delivery Capacity

Restrictions

- Do not use the keywords GCONSALE, GNETPUMP or GRUPSALE none are appropriate for use with the Gas Field Operations model.
- A variable consumption rate (a negative value in item 2 of keyword GCONSUMP or a non-zero value in item 3 of GRUPFUEL) is not allowed.
- Flux boundary conditions (see "Flux Boundary Conditions" on page 309) should not be used in conjunction with automatic DCQ reduction.
- The Gas Fields Operations model is not compatible with the Parallel option in ECLIPSE 100.

Summary of keywords

RUNSPEC section

The keywords are:

Table 24.2 RUNSPEC section

Simulator	Keyword	Description
ECLIPSE 100	GASFIELD	Enables special options in the Gas Field Operations model: the extended compressor option, multiple contract groups, and the accelerated iteration scheme for calculating the DCQ. The keyword may be omitted if the Gas Field Operations model is used without these options.

SUMMARY section

The following items are relevant to the Gas Field Operations model.

Table 24.3 Gas field operations SUMMARY output controls

E100	E300	Field	Group	Well	Information
x	x	FGSR	GGSR		Gas Sales Rate
x	x	FGST	GGST		Gas Sales Total (cumulative)
x	x	FGDC	GGDC	WGDC	Gas Delivery Capacity
x	x	FGDCQ	GGDCQ		Field/Group Gas DCQ
x			GMCPL		Group Multi-level Compressor Level
x	x		GPR		Group nodal PRessure in network When using the Extended Network model, the names must correspond to network nodes instead of groups
x			GPRDC		Group PRessure at Delivery Capacity When using the Extended Network model, the names must correspond to network nodes instead of groups
x		FGCR	GGCR		Gas consumption rate, at and below this group
x		FGCT	GGCT		Gas consumption cumulative total, at and below this group
	x	FFGR	GFGR		Fuel Gas rate, at and below this group
	x	FFGT	GFGT		Fuel Gas cumulative total, at and below this group
x		FGIMR	GGIMR		Gas import rate, at and below this group
x		FGIMT	GGIMT		Gas import cumulative total, at and below this group
x	x			NGOPAS	Number of iterations to converge DCQ in first pass
x				WGPRFP	Well Gas Production Rate from end of First Pass
x				WTHPFP	Well Tubing Head Pressure from end of First Pass

Table 24.3 Gas field operations SUMMARY output controls

E100	E300	Field	Group	Well	Information
x				WBHPFP	Well Bottom Hole Pressure from end of First Pass
x			GPRFP		Group or node PRessure in network from end of First Pass
x			GGPRNBFP		Gas flow rate along Group's or node's outlet branch in network, from end of First Pass

The mnemonics FGPR, GGPR, FGPT, GGPT refer to the **gross** gas produced from the formation, without allowing for consumption. GGDC and WGDC refer to the group and well gas sales rates when the field is operating at its delivery capacity.

GGDCQ should be used only for contract groups in cases where two or more groups have separate gas supply contracts (see keyword GSWINGF). FGDCQ should be used when there is a single contract applied to the FIELD (keyword SWINGFAC).

GPR provides the group nodal pressures when there is a production network. GPRDC may be used to obtain the group nodal pressures that exist then the field is operating at delivery capacity. When using the Extended Network model, the names must correspond to network nodes instead of groups.

The mnemonics WGPRFP, WTHPFP, WBHPFP, GPRFP, GGPRNBFP extract data from the first pass of the Gas field operations Model. However, these mnemonics do not refer to continuous quantities but rather the values of these quantities at the end of the first pass. Inclusion of any of these mnemonics in the summary section will force an additional iteration of the first pass so that converged data is recorded.

Simulator performance keyword

The keywords are:

Table 24.4 Simulator performance

Keyword	Description
NGOPAS	Outputs the number of iterations used to converge the DCQ in the first pass of the contract period.

SCHEDULE section

The keywords are:

Table 24.5 RUNSPEC section

Simulator	Keyword	Description
ECLIPSE 100	COMPOFF	Switches off automatic compressors defined by the GASFCOMP keyword (or NETCOMPA if you are using the Extended Network model).
	GASBEGIN	Denotes start of keywords to go into the Annual Scheduling File. This is required only if you want to change well controls or request printed reports in the middle of a contract period, when using GASYEAR or GASPERIO to advance the simulation.
	GASEND	Denotes end of keywords to go into the Annual Scheduling File. This is required only if there is an Annual Scheduling File.
ECLIPSE 100	GASFCOMP	Defines automatic compressors in network branches. Omit this keyword if you do not require automatic compressors. If you are using the Extended Network model (see "The extended network model" on page 693), automatic compressors must be defined with keyword NETCOMPA instead.
	GASFDELC	Specifies the definition of the delivery capacity. If this keyword is omitted, the default definition is used, in which all group and well limits are honored except those on the group and field gas production rate.
	GASMONTH	Denotes the times within each year that operations on the Annual Scheduling File should be performed, and requests printed reports. Omit this keyword if there is no Annual Scheduling File (see GASBEGIN and GASEND).
	GASPERIO	Advances the simulation through one or more complete contract periods, with the field sales gas target set equal to the DCQ multiplied by the monthly Profile Factor. Offers the option of automatic DCQ reduction to obey the Swing Requirement. TIME, TSTEP or DATES may be used instead, but the field target rate would have to be set manually each month and there would be no option of automatic DCQ reduction.
	GASYEAR	Advances the simulation through one or more complete contract years. This may be used instead of GASPERIO if the contract period is exactly one year.
	GDCQ	Sets the initial DCQs for individual contract groups. This keyword should only be used if there are multiple contract groups (keyword GSWINGF must be entered previously), in which case it must be entered before the first GASYEAR or GASPERIO keyword, and optionally again at later times if the initial DCQs need to be reset.
ECLIPSE 100	GDCQECON	Defines a minimum economic DCQ value for a contract group. All the producers in this group are shut when the DCQ falls below its minimum economic value. If the group is the FIELD, the run is terminated.
	GSWINGF	Turns on the Gas Field Operations model in the Multiple Contract Groups mode. It identifies the groups having gas supply contracts, and sets their swing and profile factors. Use this keyword only if you wish to apply separate gas supply contracts to two or more individual groups, in which case the keyword should be used in place of SWINGFAC.

Table 24.5 RUNSPEC section

Simulator	Keyword	Description
ECLIPSE 100	NETCOMPA	Defines automatic compressors in the Extended Network model. Use this instead of GASFCOMP if you are using the Extended Network model.
ECLIPSE 100	OPTIONS	Switch 34 prevents wells being drilled automatically in the 2nd pass of the contract period, in the same month that they were drilled in the first pass, if they are not needed to meet the 2nd pass production target.
	SWINGFAC	Specifies the Swing Factors and Profile Factors for each month, which are applied to the FIELD. Use this keyword if you wish to apply a single gas supply contract to the FIELD. (If you wish to apply separate gas supply contracts to two or more groups, use keyword GSWINGF instead.) Use of either SWINGFAC or GSWINGF is compulsory, as they both turn on the Gas Field Operations model.
		They activate the following features:
		• Calculation of the delivery capacity
ECLIPSE 100		• Automatic compressor switching
		• Control of sales gas rate instead of gross production rate.
ECLIPSE 100	GASFTARG	Specifies a gas rate at which the field is to be produced in the second pass of the Gas Field Operations model.
ECLIPSE 100	GASFDECR	Specifies a decrement to the gas rate at which the field is to be produced in the second pass of the Gas Field Operations model.

Gas Lift Optimization

Chapter 25

Introduction

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL

The Gas Lift Optimization facility is a Special Extension to both ECLIPSE 100 and ECLIPSE 300. It determines how much lift gas to allocate to each well in order to meet well, group or field production targets. If production targets cannot be met, it determines how to make the best use of the existing lift gas resources by allocating lift gas preferentially to the wells that can make the best use of it.

The Gas Lift Optimization facility can solve the following problems:

1 Optimizing gas lift to individual wells

How much lift gas does each well need in order to achieve its individual production target? If a well cannot achieve its production target, what is the optimum amount of lift gas to allocate to it?

2 Optimizing gas lift within a group of wells

How much lift gas does a group of wells need in order to achieve its group production target? If there is only a limited amount of lift gas available to the group, how should it best be distributed among the individual wells?

ECLIPSE 100

3 Optimizing gas lift within a simple network

If the wells flow into a sub-sea manifold connected to the surface by a flowline, how do the pressure losses in the flowline affect the optimum distribution of lift gas to the wells?

ECLIPSE optimizes the distribution of lift gas at each time step, or after user-defined intervals of time. The optimization is performed by dividing the total supply of lift gas into discrete user-defined increments, and allocating increments preferentially to the wells that can gain the most benefit from them.

The effects of gas lift are modeled by VFP tables (keyword VFPPROD). The tables must be prepared in advance with a suitable range of lift gas injection rates. The lift gas injection rate is equated with the Artificial Lift Quantity (ALQ value) in the tables. If the tables are generated by the VFP*i* program, it is essential to select the definition GRAT in the ALQ keyword in the input data file for VFP*i*. However, the VFP*i* program is also able to transform tables in which the ALQ is either IGLR (injected gas liquid ratio) or TGLR (total gas liquid ratio) into a table where the ALQ has the required definition of GRAT (lift gas injection rate). See the "VFPi User Guide" for more details.

Optimizing gas lift to individual wells

If you take a well under THP control and gradually increase the supply of lift gas to it, the production rate at first increases due to the reduced density of the mixture in the tubing. But as the lift gas supply is increased further, friction pressure losses in the tubing become more important, and the production rate peaks then starts to decrease (see Figure 25.1). For an individual well with no constraints other than a THP limit, with an unlimited free supply of lift gas, the optimum lift gas injection rate is the value at the peak (point A).

In reality, of course, lift gas injection is never free. Compression costs can be expressed as a cost per unit rate of lift gas injection (for example, dollars/day per Mscf/day). This must be balanced against the value of the extra amount of oil produced. Thus there is a “minimum economic gradient” of oil production rate versus lift gas injection rate, at which the value of the extra amount of oil produced by a small increase in the lift gas injection rate is equal to the cost of supplying the extra amount of lift gas. The optimum lift gas injection rate is then somewhat lower than the peak value, at the point on the curve where its gradient equals the minimum economic gradient (point B).

Figure 25.1 Optimum Lift Gas Injection Rate

For flexibility, each well can be given a “weighting factor” (which defaults to 1.0 if not specified). The local gradient of the oil production rate versus lift gas injection rate curve is multiplied by this weighting factor before being compared with the minimum economic gradient.

If the well is subject to additional constraints, the optimum lift gas injection rate may be lower still. For example, if the well has an oil rate target that is achievable with gas lift, it is pointless increasing the lift gas supply beyond the value that maintains the rate target.

To calculate the optimum lift gas injection rate, it is first necessary to divide the lift gas supply into discrete increments of uniform size.You must define the increment size in the keyword LIFTOPT. A typical increment size could be, say, a twentieth of the maximum allowed lift gas injection rate for each well. A smaller increment size would determine the optimum lift gas injection rate to a finer resolution, but at the expense of a longer computing time. (Generally, however, the computing time for gas lift optimization is only an important consideration when the Network option is being used.)

Each time ECLIPSE performs a gas lift optimization, it must decide whether lift gas increments must be added to, or subtracted from, each well. Taking the current lift gas injection rate of a well, ECLIPSE calculates what the well would produce if the lift gas injection rate were increased by one increment, and also what it would produce if the rate were reduced by one increment. The production rates are adjusted if necessary to obey any rate or BHP limits that the well may be subject to. From this information, ECLIPSE calculates the well's "weighted incremental gradient" and its "weighted decremental gradient" according to the formula:

$$Gradient = \left\{ \frac{\alpha_w \cdot \Delta Q_o}{(\Delta Q_{gli} + \alpha_g \cdot \Delta Q_g)} \right\} \qquad \text{[EQ 25.1]}$$

where:

α_w is the weighting factor for the preferential allocation of lift gas.

α_g is the gas production rate weighting factor.

ΔQ_o is the increment or decrement in oil production rate.

ΔQ_g is the increment or decrement in gas production rate.

ΔQ_{gli} is the gas lift increment or decrement.

By default, the gas production rate weighting factor α_g is defined to be 0.0 such that, in general, the gradients are calculated according to the reduced formula:

$$Gradient = \left\{ \frac{\alpha_w \cdot \Delta Q_o}{\Delta Q_{gli}} \right\} \qquad \text{[EQ 25.2]}$$

However, a positive value of α_g may be employed to discourage the allocation of lift gas to wells which are producing more gas. The gas rate term in the denominator is subject to the constraint $\alpha_g \cdot \Delta Q_g \geq 0.0$. If $\alpha_g \cdot \Delta Q_g \gg \Delta Q_{gli}$ the gradient becomes essentially an inverse GOR term. The weighting factors α_w and α_g are specified via items 4 and 6 respectively of the WLIFTOPT keyword.

These gradients are updated each time an increment is added or removed. ECLIPSE adds lift gas increments to the well as long as its weighted incremental gradient exceeds the minimum economic gradient, subject to any limits on the lift gas supply.

Optimizing gas lift within a group

If a group has any production rate constraints, and/or a limit on its total rate of lift gas supply, ECLIPSE allocates lift gas preferentially to the wells that gain the most benefit from it. Lift gas increments are allocated in turn to the well that currently has the largest weighted incremental gradient. The procedure takes account of any limits on the group production rate or lift gas supply applied to any level of group, including the Field.

The optimization procedure loops over all the groups on each level, firstly updating the distribution of the lift gas increments currently allocated to the group, then removing any surplus increments from the group. The currently allocated increments are redistributed by comparing the largest weighted incremental gradient and the smallest weighted decremental gradient of the subordinate wells. Consider for example that the largest weighted incremental gradient belongs to well W1, and exceeds the smallest weighted decremental gradient, which belongs to well W2. It would be more profitable to subtract an increment of lift gas from well W2 and give it to well W1. After the exchange, the incremental and decremental gradients of wells W1 and W2 are recalculated. The exchange of increments continues until no weighted incremental gradient exceeds a weighted decremental gradient, then any surplus lift gas increments are removed from the group. The group has surplus lift gas if it exceeds any production rate limits or a lift gas supply limit, or contains any wells that have a weighted decremental gradient less than the minimum economic gradient. Lift gas increments are removed in turn from the well that currently has the smallest weighted decremental gradient, until there is no surplus lift gas in the group.

The final stage of the optimization process involves looking to see whether any lift gas increments could profitably be added anywhere in the field. ECLIPSE looks for the well with the largest weighted incremental gradient that does not exceed its own lift gas supply limit, and does not belong to a group that exceeds a rate limit or a lift gas supply limit. (Note that this means that, in general, no further lift gas will be allocated once a field rate target is achieved. However, selected wells may be allocated further lift gas in order to continue optimising the oil rate if item 7 of the `WLIFTOPT` keyword is set to `YES`, for wells which belong to groups which are under gas target control.) If the well's weighted incremental gradient exceeds the minimum economic gradient, it receives another increment of lift gas. The process continues until no well can profitably receive any more lift gas increments.

The whole optimization process is efficient because it starts from the previous optimized distribution of lift gas. If the previous lift gas distribution is still the optimum, then no lift gas increments need be added or subtracted. This is more efficient than building the lift gas distribution up from zero at each time step.

If wells have efficiency factors to take account of regular downtime (see keyword `WEFAC`), their lift gas injection rates are multiplied by their efficiency factors when they are added to the group lift gas supply rate. This is consistent with the summation of flow rates for wells with downtime, and preserves the ratio of production rate to lift gas injection rate.

If groups have efficiency factors to model synchronized downtime of their subordinate wells (see keyword `GEFAC`), their lift gas injection rates are multiplied by their efficiency factors when they are added to the lift gas supply rate of the parent group.

Optimizing gas lift within a network

ECLIPSE 100

The problem becomes much more complicated when the group belongs to a production network, with a flowline to a higher level group (see "The Network Option" on page 681). Here, the wells' THP limits increase as the group's production rate increases. Adding a bit more lift gas to a particular well may increase its flow rate, but the consequent increase in the THP of all the other wells in the group will reduce the overall benefit of the extra lift gas. If the lift gas has to flow through the flowline along with the produced fluids, the extra friction pressure drop it creates could more than cancel out the extra production from the well. (Lift gas can be included in the network branch flows by setting item 6 in keyword `GRUPNET` or item 4 in keyword `NODEPROP`.)

The optimization procedure in a network is essentially the same as that described in the previous section, except that whenever a lift gas increment is added or subtracted, the incremental and decremental gradients must be recalculated for all the wells in the field. This is because a change in a well's production rate will affect the THP limits of the other wells in the network. Moreover, each time an incremental or decremental gradient is calculated, the whole network must be rebalanced with the appropriate lift gas rate in order to work out the consequent change in the field's oil production rate. This makes the computation time increase sharply according to the square of the number of wells in the field multiplied by the number of lift gas increments added and subtracted.

Gas Lift Optimization should therefore be used with care in conjunction with the Network option. It will be slow if applied to full-field studies with many wells. It is best applied to simple network systems, such as optimizing gas lift to a group of wells flowing into a common sub-sea manifold connected to the surface by a flowline. The choice of increment size is important also. A small increment size slows the calculation down as more increments need to be added and subtracted. Moreover, a small increment size may produce inaccurate estimates of the incremental and decremental gradients if the resulting change in the flow rate is small enough to be vulnerable to the allowed tolerance in the network balancing iterations. We recommend you use the `NETBALAN` keyword to decrease the network convergence tolerance by an factor of 10 (item 2), and allow a few more iterations in the balancing calculation (item 3).

Treatment of wells that can only flow under gas lift

Later in the life of the field, it may happen that some wells can only flow under gas lift. If ECLIPSE decides it is not economic to allocate lift gas to such a well, it dies. Once a well has died, it remains shut or stopped and is not considered any longer as a candidate for gas lift. However, if you want to keep the well as a candidate for gas lift, with the possibility of allocating lift gas to it at a later time if it becomes economical to do so, you can use the `WTEST` keyword to revive the well periodically for testing. When the well is revived, the optimization process considers it along with the other wells for allocation of lift gas increments. If it is not economic to allocate any lift gas to the well, it dies again. But if it is economic to allocate lift gas to the well, it will resume production.

To give the well a fair chance of revival, ECLIPSE works out the number of lift gas increments that would give the well the largest ratio of oil production rate to lift gas injection rate. For example, the well may require three increments just to get it to flow, but five increments would give it the best ratio of oil production to lift gas injection (see Figure 25.2). The well's incremental gradient would be set equal to this ratio (line O - A), to be compared with the incremental and decremental gradients of the other wells; and if this gradient is large enough, the well is given five increments of lift gas to start it flowing (position A).

Figure 25.2 Incremental gradient calculation

Thereafter the well is treated as a normal well for optimization, with single increments being added or subtracted as necessary. But if the well is only just able to flow at a particular rate of gas lift (that is the removal of another increment would cause the well to die), the decremental gradient is calculated in a special way. In our example, the well can flow with three lift gas increments, but not with two. If the well is currently on three increments of lift (position B), what is the decremental gradient? If we subtract one increment the well does not flow, but it still has two increments which are wasted. So if we decrease its lift gas rate, we may as well remove all three increments at once. The well's decremental gradient at that point is calculated as its oil rate divided by three increments of lift gas (line O - B), and if this gradient compares unfavorably with the other wells, all three increments are removed and the well dies.

Keeping wells alive with a minimum lift gas rate

Wells can be prevented from dying by ensuring they are always allocated at least enough lift gas to keep them flowing. This can be done by setting a minimum lift gas injection rate for each well in keyword `WLIFTOPT`. If a positive value is specified, the well is allocated at least that amount of lift gas, unless it is unable to flow with that rate of lift gas injection, or unless it can already meet one of its own rate limits before receiving its minimum lift gas rate. Alternatively a negative number can be specified for the minimum lift gas rate. In this case, ECLIPSE allocates at least enough lift gas to enable the well to flow, provided that it can flow at a lift gas injection rate within the maximum value specified for the well. If there is not enough lift gas available to satisfy all the wells' minimum requirements, the wells are allocated their minimum requirements in decreasing order of their weighting factor.

If a well with a minimum lift gas requirement belongs to a group (including the FIELD) having a production rate target that can be achieved without any lift gas, by default no lift gas is allocated to the well, regardless of any minimum requirement. However, if the well is given a weighting factor greater than 1.0, the well will receive its minimum lift gas requirement even if the group could achieve its production rate target without any gas lift. Thus, if you wish to allocate lift gas to keep each well flowing, regardless of whether they are needed to reach a group or field production rate target, then give them a weighting factor greater than 1.0 (for example 1.01) and specify a negative number for their minimum lift gas rate.

Note There is a possible conflict when an economic limit is applied to a well (for example using the `WECON` keyword) that has been instructed to be kept flowing in this way by gas lift optimization. When the well's flow becomes too low, it may get shut and hence will not be available to take part in the allocation of gas lift. For this reason, you should not specify conflicting economic limits if you want a well to be kept alive by gas lift optimization.

Using the gas lift optimization facility

Preparing the VFP tables

The VFP tables (keyword VFPPROD) must first be prepared for the wells (and network branches, if required). The effects of lift gas injection are handled by the Artificial Lift Quantity (ALQ value), which is equated with the lift gas injection rate. If the tables are generated by the VFP*i* program, it is essential to select the definition GRAT (lift gas injection rate) for the artificial lift quantity. However, the VFP*i* program is also able to transform tables in which the ALQ is either IGLR (injected gas liquid ratio) or TGLR (total gas liquid ratio) into a table where the ALQ has the required definition of GRAT. See the "VFPi User Guide" for more details.

Gas lift effects are modeled by interpolating the VFP table with an ALQ value equal to the current lift gas injection rate. The ALQ values in each table must span the expected range of lift gas injection rates for the well, as extrapolation of the tables can give unrealistic behavior.

ECLIPSE 100

In ECLIPSE 100, lift gas injection rates lying in between tabulated ALQ values are handled by linear interpolation, by default, like the other parameters in the table. However, there is an option to apply cubic spline interpolation to the ALQ values. This has the advantage of making the gradient of well performance with respect to the lift gas injection rate vary smoothly across the range of ALQ values. With linear interpolation, the gradient can change abruptly at the tabulated ALQ points, which may bias the optimized lift gas distribution towards these values. Spline interpolation is enabled with the keyword VFPTABL.

LIFTOPT keyword

The Gas Lift Optimization facility is activated with the keyword LIFTOPT, where the lift gas increment size and the minimum economic gradient are specified. The keyword can also be used to adjust the frequency with which the gas lift is optimized. The default is every time step. In fact, when there is no network, the optimization is generally fast enough to be performed in each of the first NUPCOL iterations of the time step, so that production targets and limits are observed more accurately. The value of NUPCOL is set using keyword NUPCOL or GCONTOL (see these keywords for a discussion of the use of NUPCOL in group control).

WLIFTOPT keyword

The keyword WLIFTOPT must be used to tell ECLIPSE which wells should have their lift gas rates calculated by optimization. It is possible to operate some wells on fixed lift gas rates, if required, while other wells have their lift gas rates optimized. The maximum lift gas injection rates for each well are also set here. To prevent extrapolation of the VFP tables, their maximum lift gas rates are defaulted to the highest ALQ value in their respective VFP tables, and they must not be given a higher value. The well weighting factors are also set here, and they default to 1.0. A minimum lift gas rate can be set for each well. If a negative value is specified, ECLIPSE allocates at least enough lift gas to keep the well flowing, if it will die without gas lift. If you wish the wells to receive their minimum lift gas requirement even if no gas lift is necessary to

achieve a group or field production rate target, you must set their weighting factors to a value greater than 1.0. A weighted gas production rate term may be included in the denominator of the gradient calculation by setting the gas rate weighting factor to a non-zero value and, if required, selected wells may be designated to continue receiving lift gas after a gas rate group target has been achieved in order to continue optimisation of the oil rate.

GLIFTOPT keyword

The keyword GLIFTOPT can optionally be used to set limits on the lift gas supply and the total gas rate (produced gas + lift gas) for individual groups. Remember that the lift gas supply to a group equals the sum of the lift gas injection rates of its subordinate wells or groups multiplied by their efficiency factors.

Group production rate limits

Group production rate limits are set with either keyword GCONPROD or GCONPRI as usual, depending on which method of group control is required. (Note that any gas rate limits set with these keywords refer to gas produced from the reservoir, and do not include the lift gas.) Lift gas is allocated if a group or the field cannot reach its oil production rate target without it. Subject to any limits on the lift gas supply or total gas rate set in keyword GLIFTOPT, increments are added until the oil production rate target has just been passed, then the group control method adjusts the well rates to meet the rate target. Increments are not added if they would cause a group rate limit for any other phase (water, gas, liquid) to be exceeded (unless increments are necessary to make up a well's minimum lift requirement and the well has a weighting factor greater than 1.0).

If gas lift is optimized in only the first iteration of a long time step, the reservoir conditions may change sufficiently over the time step so that rate targets and limits are no longer honored exactly. For example, a lift gas supply calculated to achieve the field's oil rate target at the beginning of the time step may fall somewhat short of this by the end of the time step. Conversely, a group's water rate limit honored in the first iteration may be exceeded by the final iteration of the time step. If the action on exceeding this limit is to work over the worst-offending well in the group, an un-necessary workover will be performed. It is therefore best to set the actions for rate limit violations in GCONPROD or GCONPRI to RATE or NONE. An action of NONE still makes the rate limit be honored by the gas lift optimization process. However, rate limits are honored more exactly if optimizations are performed at each of the first NUPCOL iterations of every time step, which is allowed unless there is also a network that is balanced less frequently.

Use with group production rules

ECLIPSE 100

In ECLIPSE 100, gas lift optimization can be used in combination with the group production rules (see "Group Production Rules" on page 505 and keyword PRORDER) to define a sequence of actions to be performed if a group fails to meet its production rate target set in keyword GCONPROD. These include drilling new wells, opening new connections in existing wells, reducing the THP limit or changing the well's VFP table number. By default, all the requested production rule actions are attempted **before** gas lift is allocated. However, if the mnemonic LIFT is entered in the PRORDER keyword, gas lift allocation takes precedence over any production rule actions whose mnemonics are entered after LIFT. Thus, for example, it is possible to specify that drilling new wells should only be attempted if group targets cannot be

met by applying gas lift. Essentially, all production rule actions entered after LIFT are only be performed immediately after a gas lift optimization calculation, and not at any other time or Newton iteration, if a group production target still cannot be met. In this case it is best to configure the keyword LIFTOPT so that gas lift is optimized during each of the first NUPCOL iterations of every time step.

Use with the network option

ECLIPSE 100

Gas Lift Optimization can be used with standard or extended networks in ECLIPSE 100 (see "The Network Option" on page 681). When there is a production network, gas lift can only be optimized when the network is being balanced. If the network is only balanced in the first iteration of the time step, gas lift can only be optimized for conditions at the beginning of each time step. However, if the network is balanced in each of the first NUPCOL Newton iterations of every time step (by setting a negative balancing interval in keyword NETBALAN), then the gas lift distribution can be optimized with this frequency also. Note that the computing time taken by each optimization is much longer than it is without a network, and increases sharply with the square of the number of wells in the simulation. You are also recommended to use the NETBALAN keyword to decrease the convergence tolerance in Item 2 by an order of magnitude, and increase the maximum number of iterations in Item 3.

When setting up the network with keyword GRUPNET, use Item 6 to indicate whether or not lift gas should be included in the flow along each of the branches. Lift gas from the subordinate wells can be included in either of two ways. The FLO option adds the wells' lift gas rates (multiplied by their efficiency factors) to the gas flow rate along the branch. The branch's ALQ value in Item 4 should be kept at zero. Ensure that the range of gas fraction (GFR) values in the branch's VFP table is wide enough to include the expected lift gas rates along with the produced gas. Alternatively, the ALQ option automatically sets the ALQ value of the branch to the sum of the subordinate wells' lift gas rates, multiplied by their efficiency factors. When using this option, ensure the VFP table for the branch has ALQ values spanning the expected range, to prevent the need for extrapolation.

If the Extended Network option is used, the network is constructed with keywords BRANPROP and NODEPROP instead of GRUPNET. Set item 4 of keyword NODEPROP to YES to include lift gas flows in the branch pressure drop calculation. This is equivalent to the FLO option in item 6 of GRUPNET.

Output

The current lift gas injection rates to each well, each group, and the whole field are printed in the Production Report at each Report time. These rates can also be written to the SUMMARY File, if requested with the SUMMARY section mnemonics WGLIR, GGLIR, FGLIR (Well/Group/Field Gas Lift Injection Rate).

The incremental gradients of selected wells (that is the increase in oil production rate achieved by allocating an additional increment of gas lift to the well, divided by the sum of increment size and a weighted gas production rate term) can also be written to the SUMMARY file by using the mnemonic WOGLR (Well Oil Gas Lift Ratio).

Restrictions

ECLIPSE 100

Do not use the keyword GLIFTLIM or the lift switching option in keyword WLIFT (item 5). These are meant for use with the ECLIPSE 100 Automatic Lift Switching facility, which is not as comprehensive and as flexible as Gas Lift Optimization. The equivalent keywords for Gas Lift Optimization are GLIFTOPT and WLIFTOPT

ECLIPSE 300

Gas Lift Optimization is not compatible with networks in ECLIPSE 300 (see "The Network Option" on page 681), and will be disabled if a standard or extended network is detected.

The Flux Boundary Conditions facility should not be used with Gas Lift Optimization, as no account will be taken of lift gas supplied to wells outside the flux boundary.

Summary of keywords

SUMMARY section

The following keywords control output of data specific to the Gas Lift Optimization facility:

Table 25.1 Gas lift optimization SUMMARY output controls

Field	Group	Well	Information
FGLIR	GGLIR	WGLIR	Gas Lift Injection Rate
		WOGLR	Well Oil Gas Lift Ratio (This is the incremental change in the field's production rate with respect to the addition of another gas lift increment to that well. The denominator in this expression may optionally include a weighted version of the incremental change in gas production rate.)

Note In the presence of gas lift optimization, the produced gas reported (e.g. using the FGPR, GGPR and WGPR summary section mnemonics) does not include produced lift gas, only produced formation gas.

SCHEDULE section

The keywords are:

Table 25.2 SCHEDULE section

Simulator	Keyword	Description
	GLIFTOPT	Sets limits on group lift gas supply.
	LIFTOPT	Activates Gas Lift Optimization and sets some governing parameters.
	VFPPROD	Inputs VFP tables.
ECLIPSE 100	VFPTABL	Selects the method of interpolating the ALQ variable in VFP tables.
	WLIFTOPT	Sets gas lift optimization data for wells.

Gas Plants and NGLs

Chapter 26

Gas plant tables

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

A gas plant table may be used instead of a flash calculation for a separator stage (see item 9 of keyword SEPCOND). In almost all cases, a single stage only will be required. Gas plant tables are defined with keyword GPTABLE, GPTABLEN or GPTABLE3. The table defined with GPTABLE specifies the oil recovery fraction (the number of moles of liquid output per mole of feed) for each component as a function of the 'heavy' mole fraction of the feed. The components that comprise the heavy fraction are also defined in the table.

The GPTABLEN keyword should be used instead of GPTABLE if NGL (natural gas liquid) output is also required. The table defined with GPTABLEN specifies both the oil recovery fractions and the NGL recovery fractions for each component as a function of the heavy mole fraction of the feed. The DNGL keyword should also be used to set the partial densities in the NGL stream.

The GPTABLE3 keyword defines a table with recovery fractions for oil, NGL and gas for each component as a function of the heavy mole fraction of the feed. Use this keyword instead of GPTABLE or GPTABLEN if it is required that, for one or more components, the sum of the oil, NGL and gas recovery fractions is not unity. You can use this keyword to reduce or exclude one or more components in the post separator oil, NGL or gas streams. The DNGL keyword should also be used to set the partial densities in the NGL stream.

The way in which the gas plant table splits the input streams is described below:

Suppose the input composition to the gas plant is z, and the oil, NGL and gas recovery factors are R_o, R_{NGL} and R_g. Then the moles of each component in the oil, NGL and gas streams are:

$$m_{o_i} = R_{o_i} z_i \qquad \text{[EQ 26.1]}$$

$$m_{\mathrm{NGL}_i} = R_{\mathrm{NGL}_i} z_i \qquad \text{[EQ 26.2]}$$

$$m_{g_i} = R_{g_i} z_i \qquad \text{[EQ 26.3]}$$

Where R_g is either taken directly from GPTABLE3, or calculated from the requirement that the factors sum to unity when using GPTABLE or GPTABLEN.

To obtain the oil, NGL and gas compositions, the streams are normalized:

$$x_i = \frac{m_{o_i}}{\Sigma m_{o_i}} \qquad \text{[EQ 26.4]}$$

$$n_i = \frac{m_{\mathrm{NGL}_i}}{\Sigma m_{\mathrm{NGL}_i}} \qquad \text{[EQ 26.5]}$$

$$y_i = \frac{m_{g_i}}{\Sigma m_{g_i}} \qquad \text{[EQ 26.6]}$$

The molar output rates of oil, NGLs and gas per mole of feed are given by:

$$L = \sum_i m_{o_i} \qquad \text{[EQ 26.7]}$$

$$N = \sum_i m_{\mathrm{NGL}_i} \qquad \text{[EQ 26.8]}$$

$$V = \sum_i m_{g_i} \qquad \text{[EQ 26.9]}$$

In order to define a volume rate of NGLs, the molar density must be obtained. For this we use Amagat's law - that the mixture molar volume is the mole-fraction weighted sum of the component molar volumes:

$$V_{mix} = \sum_i n_i V_i \qquad \text{[EQ 26.10]}$$

The partial molar volumes are obtained from the density values entered with keyword `DNGL`, using

$$V_i = \frac{Mw_i}{\rho_i} \qquad \text{[EQ 26.11]}$$

The recovery plant option

The Recovery Plant option takes all the gas exported from the field and processes this into NGLs and remaining gas. All other field quantities, such as GOR and gas-injection are reported prior to this processing. The aim of this facility is to model processing of the gas produced by the reservoir through facilities such as coolers, which can extract additional liquids from the stock tank gas.

Recovery plant calculations are turned on if the RECOVERY keyword is encountered. The recovery plant table may be specified in the SOLUTION or SCHEDULE sections, and may be re-specified during the SCHEDULE section.

The RECOVERY keyword has two arguments, followed by a table of one or more rows of data.

The first argument specifies the lower component index used to define the 'heavy' fraction. This defaults to the last component.

The second argument specifies the upper component index used to define the 'heavy' fraction. This defaults to the last component.

The table consists of $N_c + 1$ columns, where N_c is the number of components in the run, listing the NGL recovery fractions of each component as a function of the 'heavy' mole fraction. The recovery fraction is the number of moles of NGL per mole of feed for each component.

The defaults for items 1 and 2 imply that the recovery plant table is normally of recovery fractions against the mole fraction of the last component in the feed.

An example would be:

```
RECOVERY
6 9
--Z9  CO2 N2    C1    C2      C3   C4-6 CZ7 CZ8 CZ9
  0.0 0.0 0.002 0.001 0.45000 0.65 0.98 1.0 1.0 1.0
  0.2 0.0 0.002 0.001 0.48000 0.68 0.99 1.0 1.0 1.0
/
```

The feed used by the recovery table is the export gas from the field, which is the total gas production minus gas used for fuel and for re-injection.

The volume of NGL produced is obtained using Amagat's law, adding partial molar volumes. The molar density used is that entered with DNGL.

The rate of production of recovered NGLs and of the remaining gas is reported in the printed output, and may be obtained from the SUMMARY section using the following keywords:

Table 26.1 Keywords to obtain reports of recovered NGLs and gas

Keyword	Description
FNRR	NGL recovery volume rate
FNRT	NGL recovery volume total
FNRM	NGL recovery molar rate
FGRR	gas remaining volume rate
FGRT	gas remaining volume total

Table 26.1 Keywords to obtain reports of recovered NGLs and gas

Keyword	Description
FGRM	gas remaining molar rate
FXNR	NGL recovery composition
FYGR	gas remaining composition

The rates and totals for the recovery process are also output to the `PRINT` file under the `REPORT` header, in the table headed 'NGL recovery'. The compositions are reported in the same section in the table headed 'Recovered NGLs and remaining gas'.

GASSOL Solution in the Aqueous Phase

Chapter 27

Introduction

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

The GASSOL solution algorithm allows a specified number of gas components to dissolve in the aqueous phase. This is switched on using the GASSOL keyword in the RUNSPEC section. The basic model consists of fugacity functions for dissolving hydrocarbons in the aqueous phase, which are constructed to match solubility data and that take the form:

$$f_i^A = x_i^A P \phi_i^A(P, T) \qquad \text{[EQ 27.1]}$$

where $\phi_i^A(P, T)$ is the fugacity coefficient for component i in the aqua phase. The phase equilibrium between the aqueous hydrocarbons and the hydrocarbon phases is then defined by the conditions that the fugacity values are equal. Using the equation of state with a sample pure "gas" composition, the corresponding vapor fugacity f_i^V is computed. The aqua fugacity coefficient is then set to $\phi_i^A(P, T) = \dfrac{f_i^V}{P x_i^A}$, where x_i^A is the saturated mole fraction from the solubility table for component i.

It is also possible to construct the aqueous hydrocarbons' Gibbs energy contribution.

The solubility data may be entered using the SOLUAQA keyword. Also, a default set of data can be provided by the program for selected dissolving hydrocarbons in aqueous phase. To obtain details of the solubility data used in a simulation, the SOLU mnemonic may be specified with the RPTPROPS keyword.

The initial dissolved hydrocarbons solubility as a function of depth may be specified using the RSWVD keyword.

Notes

1 The initial dissolved hydrocarbons' concentration may not be in exact equilibrium with the hydrocarbon oil and gas phases. Also, initial flash modifies the input values slightly.

2 The first line of the solubility table defined using the SOLUAQA keyword represents the dissolving component index defined by the CNAMES keyword.

3 Where the SOLUAQA keyword is used, the solubilities of any dissolving hydrocarbon component may be defaulted, if desired. In such cases, a default solubility table is generated by applying the GASWAT framework with the component properties specified in the PROPS section. The calculations assume a feed composition that comprises a 50-50 split between water and the component in question, covering a range of pressures that is defined internally. If all dissolving components have their solubilities defaulted, then twenty pressures are considered. When some, but not all, dissolving components are assigned default solubilities, then the number of default pressures considered is equal to the maximum number of pressures observed throughout the specified solubility tables. It is likely that non-physical default solubilities will be generated if the data on which this calculation depends, notably the component properties, is not accurate enough. Henceforth, the user is strongly encouraged to enter the solubilities manually.

Aqueous phase properties

After computing the dissolving hydrocarbons' mole fraction in the aqueous phase using the phase equilibrium algorithm mentioned previously, the water density and the water viscosity μ_{sat} can be calculated using the Ezrokhi's data given by either the DENAQA or the VISCAQA keyword.

Density

The water density can be computed from a correlation of the form (Ezrokhi's method):

$$log10(\rho) = log10(\rho_0(P,T)) + \sum_i A_i(T) w_i \qquad \text{[EQ 27.2]}$$

where w_i is the weight fraction of the non-water component i and $\rho_0(P,T)$ is the density of pure water. The coefficients are a polynomial in temperature $A_i(T) = a_{0.i} + a_{1,i}T + a_{2,i}T^2$, where temperature is specified in ^{o}C. The model hence requires three regression parameters for each component specified by DENAQA keyword. The reservoir density of aqueous phase can be output by using the mnemonic DENW of RPTRST or RPTSOL.

Viscosity

The viscosity of water can be computed also from a correlation of the form (Ezrokhi's method):

$$log10(\mu) = log10(\mu_0(P,T)) + \sum_i B_i(T) w_i \qquad \text{[EQ 27.3]}$$

where w_i is the weight fraction of the non-water component i and $\mu_0(P,T)$ is the viscosity of pure water. The coefficients are a polynomial in temperature $B_i(T) = b_{0,i} + b_{1,i}T + b_{2,i}T^2$, where temperature is specified in ^{o}C. The method hence requires three regression parameters for each component specified by VISCAQA.

Aquifer properties

An analytic water aquifer defined by AQUCHWAT, AQUCT, AQUFETP or AQUFLUX contains only the "H2O" component. Any water component is allowed to leave the reservoir. Additionally, the water properties of the analytic aquifers are calculated as described above.

The GASWAT Option

Chapter 28

Introduction

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

The GASWAT option in ECLIPSE 300 provides a method of modeling gas phase/aqueous phase equilibria using an equation of state. The Peng Robinson equation of state is modified following the suggestions of Soreide and Whitson to obtain accurate gas solubilities in the aqueous phase.

Note The GASWAT option is two-phase only. By using it, the gas and water phases are implicitly requested. It is not possible to model the oil phase while using this option. If the GASWAT keyword is present, then the OIL, WATER and GAS keywords are not necessary and will be ignored.

The temperature of the reservoir may vary with depth, and this is reflected in the equation of state treatment, in which temperature coefficients are stored to be combined with the temperature of each cell. A global salinity value may be entered to correct the default equation of state modifications for brine concentration. The modified equation of state coefficients may also be over-ridden by the user.

The GASWAT option also allows molar calorific values to be entered under the CALVAL or CALVALR keywords. Gas qualities, calorific production rates and calorific production totals for wells and groups are calculated and reported. A group can be given a quality target using the GQUALITY keyword - quality is here defined as calorific value per mole of gas. The group control algorithm will attempt to meet this quality target by distributing production between wells of better and worse quality.

The reservoir may be equilibrated with a single phase present or with an initial gas-water contact. In the latter case the condition that the two phases are in equilibrium at the contact is honored - you specify the composition of one of the phases and the program obtains the composition of the other from the equilibrium condition.

A number of new summary items have been added so that calorific production rates, calorific production totals and quality values may be obtained for wells, groups and the field.

The water analytic aquifers are dealt with in the same way as in the CO2STORE option. Please refer to "Analytic water aquifers" on page 138.

The determination of single phase as gas or water use a simplified criteria. If the temperature is less than the pseudo critical temperature, as computed by the Li criteria, see keyword FACTLI, and the phase contains mostly H_2O, the phase is labeled water otherwise gas.It is possible to activate the pre-2008.1 behavior by putting OPTIONS3 switch 146 to 6. Also the flash will treat H_2O as the most heavy component unless OPTIONS3 switch 125 is put to -1.

Note that the default density and viscosity calculations for water with solutes may be replaced by Ezrokhi's method, the latter also used in conjunction with CO2STORE and GASSOL. This is done by adding the keywords DENAQA and VISCAQA respectively for density and viscosity, and specifying the Ezrokhi temperature coefficients for each component.

Equation of state modifications

The conventional Peng-Robinson equation of state

The A -coefficient in the Peng-Robinson equation of state is given by:

$$A = \sum_{jk} x_j x_k A_{jk} \qquad \text{[EQ 28.1]}$$

where x are component mole fractions and

$$A_{jk} = (1 - k_{jk})(A_j A_k)^{1/2} \qquad \text{[EQ 28.2]}$$

k_{jk} are binary interaction coefficients. A_{jk} is defined as:

$$A_j = \frac{\Omega_j P_{rj}}{T_{rj}^2} \qquad \text{[EQ 28.3]}$$

The reduced critical temperatures and pressures are $T_r = T/T_c$ and $P_r = P/P_c$. We define reduced A-coefficients by extracting this temperature and pressure dependence:

$$A_j = \frac{A_j^r \cdot P}{T^2} \qquad \text{[EQ 28.4]}$$

The Peng-Robinson equation of state then defines A_j^r as:

$$A_j^r = \frac{\Omega_{oj} T_{cj}^2 \alpha_j}{P_{cj}} \qquad \text{[EQ 28.5]}$$

with

$$\alpha_j^{1/2} = 1 + f(w_j)(1 - T_{rj}^{1/2}) \qquad \text{[EQ 28.6]}$$

$f(w_j)$ is a polynomial in the acentric factor, given by:

$$f(w_j) = 0.37464 + 1.54226 w_j - 0.26996 w_j^2 \qquad \text{[EQ 28.7]}$$

To account for the temperature dependence in this expression we write it in the form:

$$A_j^r = \frac{\Omega_{oj} T_{cj}^2 (p_j + q_j T^{1/2})^2}{P_{cj}} \qquad \text{[EQ 28.8]}$$

where $p_j = 1 + f(w_j)$ and $q_j = -f(w_j)/T_c^{1/2}$.

Then:

$$A_{rjk} = [1 - k_{jk}] \left[\frac{\Omega_{oj} T_{cj}^2}{P_{cj}}\right]^{1/2} \left[\frac{\Omega_{ok} T_{ck}^2}{P_{ck}}\right]^{1/2} [p_j p_k + (p_j q_k + p_k q_j) T^{1/2} + q_j q_k T] \qquad \text{[EQ 28.9]}$$

In ECLIPSE 300 we store the three coefficients A_{1jk}, A_{2jk} and A_{3jk}, where:

$$A_{1jk} = (1-k_{jk})\left(\frac{\Omega_{oj}T_{cj}^2}{P_{cj}}\right)^{1/2}\left(\frac{\Omega_{ok}T_{ck}^2}{P_{ck}}\right)^{1/2}(p_j p_k) \qquad \text{[EQ 28.10]}$$

$$A_{2jk} = (1-k_{jk})\left(\frac{\Omega_{oj}T_{cj}^2}{P_{cj}}\right)^{1/2}\left(\frac{\Omega_{ok}T_{ck}^2}{P_{ck}}\right)^{1/2}(p_j q_k + p_k q_j) \qquad \text{[EQ 28.11]}$$

$$A_{3jk} = (1-k_{jk})\left(\frac{\Omega_{oj}T_{cj}^2}{P_{cj}}\right)^{1/2}\left(\frac{\Omega_{ok}T_{ck}^2}{P_{ck}}\right)^{1/2}[q_j q_k] \qquad \text{[EQ 28.12]}$$

This enables the equation of state coefficient for a cell of temperature T to be simply assembled as:

$$A_{jk} = (A_{1jk} + A_{2jk}T^{1/2} + A_{3jk}T)\frac{P}{T^2} \qquad \text{[EQ 28.13]}$$

Modifications for the vapor-aqueous system

If the Peng-Robinson equation is used in the above form for a gas-aqueous system the predicted saturated mole fractions of gases dissolved in water are hopelessly wrong - predicted solubilities are typically 10^{-7} mole/mole, whilst experimental values are nearer 10^{-3}mole/mole.

Soreide and Whitson propose two changes:

- The expression for $\alpha^{1/2}$ above is modified for the water component. It is replaced by an expression with a more complex temperature dependence:

$$\alpha^{1/2} = 1 + aq_1(1 - aq_2 T_r) + aq_3(T_r^{-3} - 1) \qquad \text{[EQ 28.14]}$$

  with default values:

  $aq_1 = 0.4530$

  $aq_2 = 1 - 0.0103c_s^{1.1}$

  $aq_3 = 0.0034$

  c_s is the brine concentration expressed as a molality. (Molality is given by $[1000 \times (\text{moles of salt})/(\text{mass of water})]$. This is discussed further below).

  In ECLIPSE 300 you may overwrite the defaults for aq_1, aq_2 and aq_3 by entering the `AQCOEF` keyword.

  This modification is only done for the water component, and the remainder take the usual form. A general form for all the components is thus:

$$\alpha^{1/2} = p + qTr^{1/2} + rTr + sTr^{(-3)} \qquad \text{[EQ 28.15]}$$

- Different binary interaction coefficients are used for the gas and aqueous phases. This would cause problems near the critical point, but for a water-dominated system the critical temperature is in the region of $T_{cw} = 647\text{K}$, which should be well above the area of interest.

  Soreide and Whitson propose adding a temperature dependence to the aqueous phase binary interaction coefficients:

$$k_{jw}^{a} = bq_1 + bq_2 T_{rj} + bq_3 T_{rj}^2 \qquad \text{[EQ 28.16]}$$

with default values:

$$bq_1 = A_0(1 + S_0 c_s)$$

$$bq_2 = A_1(1 + S_1 c_s)$$

$$bq_3 = A_2(1 + S_2 c_s)$$

where:

$$A_0 = 1.112 - 1.7369 w_j^{-0.1}$$

$$A_1 = 1.1001 + 0.836 w_j$$

$$A_2 = -0.15742 - 1.0988 w_j$$

and:

$$S_0 = 0.017407$$

$$S_1 = 0.033516$$

$$S_2 = 0.011478$$

Special values are used for binary interaction coefficients between the aqueous phase and N_2, CO_2 and H_2S:

$$k_{N_2w}^{a} = -1.70235(1 + 0.025587 c_s^{3/4}) + 0.44338(1 + 0.08126 c_s^{3/4}) T_{rN_2} \qquad \text{[EQ 28.17]}$$

$$k_{CO_2w}^{a} = -0.31092(1 + 0.15587 c_s^{0.7505}) + (0.23580)(1 + 0.17837 c_s^{0.979}) T_{rCO_2} - 21.2566 \exp(-6.7222 T_{rCO_2} - c_s) \qquad \text{[EQ 28.18]}$$

$$k_{H_2Sw}^{a} = -0.20441 + 0.23426 T_{rH_2S} \qquad \text{[EQ 28.19]}$$

These fit into the general quadratic form, apart from the exponential term in the CO_2 case: this is evaluated at the reservoir temperature or surface standard conditions and added to the bq_1 term.

In ECLIPSE 300 you may overwrite these defaults for bq_1, bq_2 and bq_3 using the `BICAQ1`, `BICAQ2` and `BICAQ3` keywords.

The above forms for α and k_{ajk} may then be simply assembled with the cell temperature to build up the equation of state A-coefficient.

The program automatically uses both these modifications in GASWAT mode. The water component is identified as that with name ‘H2O’ - this must be present. Nitrogen, carbon dioxide and hydrogen sulfide, if present, should be identified with the names ‘N2’, ‘CO2’ and ‘H2S’ respectively. Keywords such as SALINITY, SALINITR, AQCOEF, BICAQ1, BICAQ2 and BICAQ3 are entered once, and need not be specified for each equation of state region or separately for reservoir and surface conditions. The gas binary interaction coefficients are obtained from the usual user-input data.

Calorific values

Units

These are entered in the PROPS section under the CALVAL or CALVALR keywords. The metric units are kJ/kg-mol. For conversion purposes 1 M^3=0.04165935 kg-mol at normal standard conditions. Well and group quality is reported in kJ/kg-mol. Calorific rates and totals are in kJ/D and kJ respectively. The equivalent field units are BTU/lb-mol etc.

Calorific well and group production rates and totals are reported in the PRINT file under the REPORT, FIELD and WELLS headers. The gas quality is also reported - this is the mixture calorific value per mole.

Specifying group production quality

Where a group target is set - for example a required group gas production rate - the group distributes this target amongst the wells in the group using guide rates. By modifying these guide rates we can weight the production between low and high quality wells so that a required group quality is obtained. This required quality can be set using the GQUALITY keyword.

This option applies only if:

1. The group has more than one son under group control
2. At least one of the group controlled sons has a quality better than the required target and at least one has a quality worse than the required target.

Generally a point is ultimately reached when the required calorific value can no longer be met (for example if all the wells in the group are producing gas of too low a quality). The quality does not then honor the required target.

Salinity

In the above, salinity is defined by molality: $c_s = 1000 \cdot n_s / m_w$, where n_s is the number of moles of salt and m_w the mass of the water. This may be related to the usual definition in terms of parts per million by mass:

$$c_s^{ppm} = A \cdot c_s / (1 + B \cdot c_s) \qquad \text{[EQ 28.20]}$$

$$c_s = c_s^{ppm} / (A - B \cdot c_s^{ppm}) \qquad \text{[EQ 28.21]}$$

with $A = 58440.0, B = 0.05844$

The salinity variation of the equation of state coefficients is not treated on a cell by cell basis - a single value is entered using the `SALINITY` keyword in the `PROPS` section. If it is not entered the salinity value is assumed to be zero.

Relative permeability tables

The water relative permeability is input using the WSF keyword, while the gas relative permeability and the gas-water capillary pressure are entered using the GSF keyword. These keywords are analogous to the SOF2 and SGFN keywords.

The saturation tables can be output to the PRINT file by entering the WSF/GSF mnemonics in the RPTPROPS keyword.

Initialization

The equilibration of the reservoir is straightforward if only a single phase is present: the composition may be entered using ZI or with respect to depth using ZMFVD.

The situation is slightly more complicated if an initial gas-water contact exists within the reservoir. The need for equilibrium at the contact means that the two phase compositions cannot be specified independently. You may specify either the vapor or aqueous phase composition at the contact (ZMFVD or COMPVD may be used). The ninth argument of the EQUIL keyword specifies which is the phase specified. The other phase composition is then defined by the bubble or dew point condition. Strictly, this also defines the pressure and by default this is reset to the equilibrium value. However, this resetting of the pressure may be over-ridden using the tenth argument of EQUIL.

Note Note that the phase equilibrium is only required at the contact: COMPVD allows arbitrary aqueous phase compositions below the water-gas contact and arbitrary vapor compositions above it.

Summary items

The following SUMMARY section keywords are part of the GASWAT development:

W

- WEPR Well energy production rate
- WEPT Well energy production rate
- WEIR Well energy injection rate
- WEIT Well energy injection rate
- WGQ Well gas quality

G

- GEPR Group energy production rate
- GEPT Group energy production rate
- GEIR Group energy injection rate
- GEIT Group energy injection rate
- GGQ Group gas quality

F

- FEPR Field energy production rate
- FEPT Field energy production rate
- FEIR Field energy injection rate
- FEIT Field energy injection rate
- FGQ Field gas quality

DENSITY values are not used in the GASWAT option: both gas and water density values are obtained from the equation of state.

Note as well that the water phase is considered as “hydrocarbon” when calculating the “hydrocarbon weighted” field pressure reported by FPR and FPRH. For a comparison with a simulation involving an oil phase such as CO2SOL, the “pore volume weighted” pressure returned by FPRP can be used. Additionally, setting the 151st item of OPTIONS3 to 1 ensures that only the gas phase is treated as “hydrocarbon”.

Gas Calorific Value Control

Chapter 29

Introduction

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL

The Gas Calorific Value Control option is a Special Extension to ECLIPSE 100; it is not available in ECLIPSE 300, but the GASWAT option in that simulator (see "The GASWAT Option" on page 405) offers some of the same functionality. The purpose of this option is to enable you to control the mean calorific value of gas produced from the field, while also controlling the gas production rate. Production rate targets are handled by guide rate group control (keyword GCONPROD), and ECLIPSE automatically adjusts the guide rates so that the produced mixture has, whenever possible, the required mean calorific value. As an alternative to controlling the gas production rate, you may elect instead to control the overall energy produced (that is the calorific value times volumetric production rate).

Calorific value

The calorific value is a measure of the energy content of gas, that is the energy released when a given volume of the gas is burned. ECLIPSE measures the calorific value in units of kJ/sm^3 (METRIC), Btu/Mscf (FIELD), J/scc (LAB).

The term ‘quality’ is also used to denote the energy content of gas. However, we must distinguish between the volumetric quality and the molar quality. Volumetric quality is the energy content per unit volume, while molar quality is the energy content per mole of gas. Molar quality has units of KJ/Kg-mol (METRIC), Btu/lb-mol (FIELD), J/gm-mol (LAB). Conversion between the two definitions of quality can be performed assuming ideal gas behavior:

- 23.6903 sm^3/Kg-mol (METRIC)
- 0.379506 Mscf/lb-mol (FIELD)
- 23690.3 scc/gm-mol (LAB).

As ECLIPSE 100 is not a compositional simulator, the calorific value of reservoir gas cannot be computed from its composition; you must enter it. The calorific value of the produced gas may either be specified independently for each well, or equated to the concentration of a gas phase tracer.

Controlling the mean calorific value

When the production rate of a group (or the field) is under guide rate control (keyword GCONPROD), its rate target is shared between the subordinate wells in proportion to their guide rates. It is also possible to control simultaneously the mean calorific value of the gas produced by the group, by automatically adjusting the well guide rates to give an appropriate mixture of high and low calorific value gas. We divide the set of subordinate wells currently under group control into two categories: those producing gas with higher calorific value than the target, and those with a calorific value lower than the target. All the wells in the second category have their guide rates multiplied by the same adjustment factor, calculated during each of the first NUPCOL iterations of the time step to give the required overall calorific value. This method works provided that there is at least one subordinate well under group control in each of the two categories.

There could ultimately come a point when the required calorific value cannot be met, when there are no group controlled wells left in one or other of the two categories. It may be possible to restore the required mean calorific value by drilling a new well or by performing one of the other actions associated with the group production rules (keyword PRORDER). Otherwise there is an option to reduce the group's flow target in stages until wells straddling the required calorific value are brought back under group control. This option would therefore attempt to honor the calorific value target at the expense of the flow rate target.

Controlling the energy production rate

Instead of supplying a target for the group's flow rate, you may supply a target for its energy production rate (that is the gas production rate multiplied by its calorific value). ECLIPSE shares a group's energy target among its subordinate wells in proportion to their energy guide rates (that is the gas guide rates multiplied by their latest calorific values).

This facility may be used with or without a control target for the mean calorific value. If there is no such control target, the mean calorific value depends on the guide rates of the wells.

Adjusting guide rates to control mean calorific value

Wells under group control have their production rate targets set equal to their guide rates for the controlled phase multiplied by the controlling group's guide rate multiplier. But if the group's mean calorific value is also controlled to meet a specified target, those wells under group control whose calorific values are less than the target will have their guide rates multiplied by an adjustment factor α. Thus the group's overall production rate of the phase p under guide rate group control is given by

$$Q_{pG} = \sum_{W \in i} Q_{pW} + M_G \sum_{W \in g^+} G_{pW} + M_G \, \alpha \sum_{W \in g^-} G_{pW} \qquad \text{[EQ 29.1]}$$

where:

Q_{pG} is group G's production rate of the controlled phase p

Q_{pW} is well W's production rate of the controlled phase p

G_{pW} is well W's guide rate of the controlled phase p

M_G is the guide rate multiplier for group G

α is the guide rate adjustment factor for the set $W \in g^-$

$W \in i$ represents the set of subordinate wells operating under their individual control modes

$W \in g^+$ represents the set of subordinate wells on group control whose calorific values exceed the target

$W \in g^-$ represents the set of subordinate wells on group control whose calorific values are less than the target.

The group's guide rate multiplier is obtained from [EQ 29.1] by setting

$$Q_{pG} = Q_{pG}^{\text{targ}} \qquad \text{[EQ 29.2]}$$

where

Q_{pG}^{targ} is group G's target production rate of the controlled phase p.

This gives

$$M_G = \frac{Q_{pG}^{\text{targ}} - \sum_{W \in i} Q_{pW}}{\sum_{W \in g^+} G_{pW} + \alpha \sum_{W \in g^-} G_{pW}} \qquad \text{[EQ 29.3]}$$

The guide rate adjustment factor α is calculated from the condition that the mean calorific value of the group's gas production must be equal to a specified target,

$$\frac{\sum_W C_{gW} Q_{gW}}{\sum_W Q_{gW}} = C_{gG}^{\text{targ}} \qquad \text{[EQ 29.4]}$$

where:

Q_{gW} is well W's gas production rate

C_{gW} is the calorific value of the gas produced by well W

C_{gG}^{targ} is the target mean calorific value for group G.

The flow rate of a well under group control depends upon its guide rate and the group's guide rate multiplier,

$$Q_{gW} = M_G G_{pW} (Q_g/Q_p)_W \quad \text{for } (W \in g^+)$$
$$Q_{gW} = \alpha M_G G_{pW} (Q_g/Q_p)_W \quad \text{for } (W \in g^-) \qquad \text{[EQ 29.5]}$$

where

$(Q_g/Q_p)_W$ is the well's latest known phase production ratio of gas to the controlled phase p.

Substituting [EQ 29.5] into [EQ 29.4] yields

$$M_G \sum_{W \in g^+} (C_{gW} - C_{gG}^{\text{targ}})\, G_{pW}\, (Q_g/Q_p)_W - \alpha M_G \sum_{W \in g^-} (C_{gG}^{\text{targ}} - C_{gW}) G_{pW} (Q_g/Q_p)_W = \sum_{W \in i} (C_{gG}^{\text{targ}} - C_{gW}) Q_{gW} \qquad \text{[EQ 29.6]}$$

Simultaneous solution of [EQ 29.1] and [EQ 29.6] yields the guide rate adjustment factor α

$$\alpha = \frac{a \sum_{W \in g^+} (C_{gW} - C_{gG}^{\text{targ}}) G_{pW} \left(\frac{Q_g}{Q_p}\right)_W - b \sum_{W \in g^+} G_{pW}}{a \sum_{W \in g^-} (C_{gG}^{\text{targ}} - C_{gW}) G_{pW} \left(\frac{Q_g}{Q_p}\right)_W + b \sum_{W \in g^-} G_{pW}} \qquad \text{[EQ 29.7]}$$

where:

$$a = Q_{pG}^{\text{targ}} - \sum_{W \in i} Q_{pW}$$
$$b = \sum_{W \in i} (C_{gG}^{\text{targ}} - C_{gW}) Q_{gW} \qquad \text{[EQ 29.8]}$$

Once α has been obtained from [EQ 29.7], the group's guide rate multiplier M_G can be obtained from [EQ 29.3]. α of course must be constrained to be positive. A negative solution for α implies that the target mean calorific value cannot be achieved at the target flow rate. When both M_G and α are known, the group controlled wells can be given their production rate targets. If a well cannot achieve its rate target, it drops out of group control and becomes independently controlled (at its own rate or pressure limit), and the calculations for M_G and α must be repeated.

If the group G has subordinate groups that have been given group guide rates, then these groups and their guide rates will be dealt with as if they were wells in the above equations. The wells subordinate to these groups with guide rates are not processed as individual entities. Essentially, the subscript W in the above equations refers to the set of wells or groups with guide rates that can come under direct control of group G.

Action when calorific value target cannot be met

In order to meet the mean calorific value target, the set of wells under group control must include at least one well producing gas with a calorific value greater than the target, and at least one producing with less than the target calorific value. ECLIPSE can take certain remedial actions when there is insufficient high or low calorific value gas available.

Not enough high calorific value gas

If the group is meeting its production rate target, but the numerator of [EQ 29.7] is zero or negative, there are two actions that ECLIPSE may take:

1 Increase the productivity of high calorific value gas sources.

If group production rules have been specified (keyword PRORDER), ECLIPSE performs the requested actions with the aim of obtaining more high quality gas. The DRILL action opens the next suitable well from the drilling queue that has a calorific value greater than the target value. The REPERF action selects the well with the largest sum of {connection factor * gas phase mobility * $(C_{gW} - C_{gG}^{\text{targ}})$ } of its remaining connections on AUTO, and opens the next connection in its AUTO queue. The THP, RETUBE and LIFT actions select the well with the largest value of $\Delta Q_{gW}\,(C_{gW} - C_{gG}^{\text{targ}})$, where ΔQ_{gW} is the increase in gas production rate that would result from the action. COMP actions are not triggered by a calorific value target. If the PRORDER keyword is not present, only the DRILL action will be performed, if there are wells in the drilling queue.

2 Cut back the group's gas rate target to bring more high calorific value gas wells under group control.

If the productivity of high calorific value gas sources cannot be enhanced by the production rule actions described above, but there are some high calorific value gas sources that have dropped out of group control because they are unable to meet their share of the target, it could be possible to bring them back under group control by decreasing the group's gas rate target. If this action is requested, ECLIPSE cuts back the group's gas rate target by a specified fraction each time this situation arises, until the rate target is low enough to bring them back under group control. The need to maintain the target calorific value takes precedence over the group's rate target.

Not enough low calorific value gas

If the group is meeting its production rate target, but the denominator of [EQ 29.7] is zero or negative, there are two actions that ECLIPSE may take:

1 Increase the productivity of low calorific value gas sources.

The production rule actions as described above are invoked, but this time with the aim of making more low calorific value gas available. The DRILL action looks for a well with a calorific value less than the target. The REPERF action selects he well with the largest sum of {connection factor * gas phase mobility * $(C_{gG}^{\text{targ}} - C_{gW})$ } of its remaining connections on AUTO. The THP, RETUBE and LIFT actions select the well with the largest value of $\Delta Q_{gW}\,(C_{gG}^{\text{targ}} - C_{gW})$.

2 Cut back the group's gas rate target to bring more low calorific value gas wells under group control. This is performed as described above, if the production rule actions are not able to restore the group's mean calorific value.

Not enough gas to meet the group's rate target

If there are no wells at all under group control, the group's rate target will no longer be met. ECLIPSE may take the following actions:

1 Apply the production rules to enhance gas production.

The production rules are applied in the usual way to maintain the group's production target (as described in keyword `PRORDER`), but with the following difference. If the group's mean calorific value is below the target value, ECLIPSE initially only considers the wells that are producing gas with calorific value higher than the group's target; if no such wells are available for action then the low quality wells are examined also. Conversely if the group's mean calorific value is above the target value, ECLIPSE initially only considers the wells that are producing gas with calorific value lower than the group's target; if no such wells are available for action then the high quality wells are examined also.

2 Cut back the group's gas rate target to bring some high and low calorific value gas wells under group control.

This is performed as described above, if the production rule actions are not able to restore the group's mean calorific value.

Controlling the energy production rate

The energy production rate of the group is given by

$$E_G = \sum_{W \in i} E_W + M_G \sum_{W \in g^+} G_{eW} + M_G \, \alpha \sum_{W \in g^-} G_{eW} \qquad \text{[EQ 29.9]}$$

where:

E_W is $C_{gW} \, Q_{gW}$, the energy production rate from the well

G_{eW} is $C_{gW} \, G_{gW}$, the well's energy guide rate, which is set equal to its gas phase guide rate multiplied by its calorific value at the beginning of the time step.

When the control target refers to the overall energy produced rather than the gas flow rate, the group's guide rate multiplier is obtained from [EQ 29.9] by equating E_G with the target value. If there is **no** simultaneous control of the mean calorific value, the group's guide rate multiplier is

$$M_G = \frac{E_G^{\text{targ}} - \sum_{W \in i} E_W}{\sum_{W \in g} G_{eW}} \qquad \text{[EQ 29.10]}$$

where:

E_G^{targ} is group G's target energy rate

$W \in g$ represents the set of subordinate wells on group control.

If in addition there is also a target mean calorific value for the group, its guide rate multiplier is

$$M_G = \frac{E_G^{\text{targ}} - \sum_{W \in i} E_W}{\sum_{W \in g^+} G_{eW} + \alpha \sum_{W \in g^-} G_{eW}} \qquad \text{[EQ 29.11]}$$

When the controlled quantity is energy, [EQ 29.6] becomes

$$M_G \sum_{W \in g^+} \left(1 - \frac{C_{gG}^{\text{t arg}}}{C_{gW}} \right) G_{eW} - \alpha \, M_G \sum_{W \in g^-} \left(\frac{C_{gG}^{\text{targ}}}{C_{gW}} - 1 \right) G_{eW} \qquad \text{[EQ 29.12]}$$

$$= \sum_{W \in i} (C_{gG}^{\text{targ}} - C_{gW}) Q_{gW}$$

Simultaneous solution of [EQ 29.9] and [EQ 29.12] gives the guide rate adjustment factor required to meet the target mean calorific value

$$\alpha = \frac{e \sum_{W \in g^+} \left(1 - \frac{C_{gG}^{\text{t arg}}}{C_{gW}}\right) G_{eW} - b \sum_{W \in g^+} G_{eW}}{e \sum_{W \in g^-} \left(\frac{C_{gG}^{\text{targ}}}{C_{gW}} - 1\right) G_{eW} + b \sum_{W \in g^-} G_{eW}} \quad \text{[EQ 29.13]}$$

where:

$$e = E_G^{\text{targ}} - \sum_{W \in i} E_W$$

$$b = \sum_{W \in i} (C_{gG}^{\text{targ}} - C_{gW}) Q_{gW} \quad \text{[EQ 29.14]}$$

Action when energy target cannot be met

When the group's energy target cannot be met, ECLIPSE applies the production rules in the usual way to maintain the target (see keyword PRORDER). However, since the control is on energy rather than gas, the rules are applied to enhance energy production rather than gas production. The DRILL action opens the next suitable well from the drilling queue, as usual. The REPERF action selects the well with the largest sum of {connection factor * gas phase mobility * calorific value} of its remaining connections on AUTO, and opens the next connection in its AUTO queue. The THP, RETUBE and LIFT actions select the well with the largest value of $\Delta Q_{gW} C_{gW}$, where ΔQ_{gW} is the increase in gas production rate that would result from the action. COMP actions are triggered in the usual way. If the PRORDER keyword is not present, only the DRILL action is performed, if there are wells in the drilling queue.

When the group also has a calorific value target, the production rule actions described in "Not enough high calorific value gas" on page 421 are applied in order to maintain it.

Using the facility

Defining the calorific value

The calorific value of the produced gas may be defined in either of two ways:

- it may be set independently for each well
- it may be equated to the concentration of a nominated gas phase tracer.

Defining it for each well provides a simple approach that is suitable when gases of different calorific value do not mix within the reservoir, for example when the field consists of a collection of isolated formations containing gases of different calorific value. The value for each well is specified in the SCHEDULE section with keyword WCALVAL, for example:

```
WCALVAL
-- Well       Calorific
-- name       value
 'PRODA*'     30000.0 /
 'PRODB*'     26000.0 /
/
```

Changes in calorific value with time can be accommodated 'manually' by re-entering the WCALVAL keyword.

Equating the calorific value to the concentration of a gas phase tracer provides a more general approach, which is recommended when gases of different calorific values may mix within the reservoir. A gas phase passive tracer may be defined (keyword TRACER in the PROPS section), and initialized to equal the calorific value in units of kJ/sm^3 (METRIC), Btu/Mscf (FIELD), J/scc (LAB). (See "Tracer Tracking" on page 1117 for an account of how to define tracers.) ECLIPSE should then be told which tracer to use for the calorific value, with the keyword CALTRAC, for example:

```
CALTRAC
 CAL    /
```

ECLIPSE then takes each well's calorific value from its produced concentration of this named tracer, unless the well's calorific value has already been defined with keyword WCALVAL. If keyword CALTRAC is **not** used to nominate a gas phase tracer, all production wells should have their calorific values set with WCALVAL.

Controlling the mean calorific value

The target mean calorific value for a group (or the field) is set with keyword GCONCAL. The keyword also contains a flag to specify the action to be taken when this target cannot be met. The options are either to take no action ('NONE'), or to cut back the group's gas rate target successively until the mean calorific value target is honored ('RATE'). The fraction by which the rate target is reduced each time is set by the user.

```
GCONCAL
-- Group        Target        Action     Rate reduction
-- name         Cal Val                    fraction
  FIELD         29000.0        RATE          0.9  /
/
```

The facility can only work when the group is operating under guide rate control. Thus it must have a target gas production rate set for it with keyword GCONPROD, or alternatively a target energy rate set with keyword GCONENG (see below).

Calculating guide rates on the basis of calorific value

Another way of influencing the mean calorific value is to calculate the individual well guide rates (and any group guide rates) as a function of their calorific values. Well guide rates that are not set directly with keyword WGRUPCON are calculated at the start of each time step as a function of their production potentials. By default the guide rates are set equal to their production potentials, but the keyword GUIDERAT can be used to specify the coefficients of a more general function of production potentials.

The keyword GUIDECAL can be used to make the guide rate calculation take account of calorific values also. It sets the coefficients A and B of a function β_W which depends on the calorific value C_{gW}

$$\beta_W = (1 + AC_{gW})^B \qquad \text{[EQ 29.15]}$$

When a well W has its guide rate calculated from its production potentials, the resulting guide rate will be multiplied by β_W. Thus by setting appropriate values for the coefficients A and B in keyword GUIDECAL, it is possible to either increase or decrease the well guide rates with increasing calorific value. For example, setting a positive value for B will bias the production towards wells with higher calorific values. The coefficient A cannot be negative.

If **groups** are given guide rates that are calculated from the guide rate formula in keyword GUIDERAT, their guide rates will also be multiplied by β_G, which is calculated as for β_W but using the groups' mean calorific values.

Controlling the energy production rate

As an alternative to controlling the gas production rate with GCONPROD, a target may be supplied for the overall energy production rate. The energy rate is simply the gas production rate multiplied by its calorific value, and is measured in units of kJ/day (METRIC), Btu/day (FIELD), J/hr (LAB). The energy rate target for a group (or the field) is set with the keyword GCONENG,

```
GCONENG
-- Group        Target
-- name         energy rate
  FIELD            1.0E9 /
/
```

The GCONENG keyword may be used with or without a GCONCAL target for the mean calorific value. If there is no GCONCAL target the mean calorific value will depend on the well guide rates. Note that a GCONENG target requires the gas production rate to increase as the mean calorific value falls. Consequently if the GCONCAL keyword is also used, the action flag in item 3 should normally be set to NONE to leave ECLIPSE free to increase the flow rate as necessary.

Economic limits

The keyword GCALECON allows minimum economic limits to be applied to a group's energy rate and to its mean calorific value. If a group violates either of these limits its subordinate producers will be closed, and optionally the run may be ended.

```
GCALECON
-- Group        Min Economic     Min Economic      End run flag
-- name         energy rate     calorific value
  FIELD            1.0E9 /
  'G*'             5.0E7            20000.0             YES   /
/
```

Use with the gas field operations model

The GCONCAL keyword may be used in combination with the Gas Field Operations Model, if required (see "Gas Field Operations Model" on page 359). It should only be applied to contract groups, and the contract groups must be under guide rate control rather than prioritization. There is no need to use GCONPROD to place contract groups under guide rate control; this is done automatically when their swing and profile factors are entered, as long as GCONPRI is not used.

If the delivery contract is expressed in terms of energy production rate, instead of gas flow rate, use the DCQDEFN keyword in the SCHEDULE section.

```
DCQDEFN
  ENERGY /
```

This tells ECLIPSE that the DCQ values entered in keyword GASYEAR, GASPERIO or GDCQ refer to a daily contracted quantity of **energy** instead of flow rate. You may then enter the energy rate DCQ values in place of the gas rate DCQ values, in the units appropriate to energy production rate: kJ/day (METRIC), Btu/day (FIELD), J/hr (LAB). If the DCQDEFN keyword is used it **must** be entered before any other Gas Field Operations keyword, including the SWINGFAC and GSWINGF keywords. ECLIPSE gives the contract groups a GCONENG target equal to the energy DCQ multiplied by the month's swing or profile factor.

Satellite groups

Energy can be produced or injected by satellite groups by specifying a gas production or injection rate **and** defining a non-zero calorific value for the gas. See the keywords GSATPROD and GSATINJE for more details.

Energy injection

ECLIPSE does not allow direct control over injected energy rates or net (that is the produced minus injected) energy rates. However, energy sales rates can be reported in the SUMMARY file, which represent the energy produced minus energy injected minus the energy content of any consumed gas. Injected energy can be taken into account by giving the gas injection wells a specified calorific value with the WCALVAL keyword. Alternatively, if a tracer is used to describe the energy content of the gas, the WTRACER keyword can be used to set the tracer value in the injection wells. Gas injection should not be applied when using the Gas Field Operations model.

Output

An additional table is printed in the Well Reports showing the calorific value, energy production rate and cumulative energy produced for each well and group. Note that if the calorific value is equated with the tracer 'CAL', these quantities may also be obtained from the Tracer Reports.

The following SUMMARY section keywords may also be used to provide appropriate output to the Summary file:

Table 29.1 Gas Calorific Value Control SUMMARY output

Keywords			Description
FGCV	GGCV	WGCV	Gas calorific value
FGGQ	GGGQ	WGQ	Gas molar quality
FEPR	GEPR	WEPR	Energy production rate
FEPT	GEPT	WEPT	Cumulative energy production total
FESR	GESR		Energy sales rate
FEST	GEST		Cumulative energy sales total
FEDC	GEDC	WEDC	Energy delivery capacity
FEDCQ	GEDCQ		Field/Group Energy DCQ

The energy delivery capacity and energy DCQ are available only in conjunction with the Gas Field Operations model.

Restrictions

- If `GCONCAL` is used in a Reservoir Coupling master run (see "Reservoir Coupling" on page 811), the master cannot control the production of individual wells in the slave reservoirs. Instead it will apply adjustment factors to the guide rates of the master groups. Also, production rules applied in the master reservoir cannot affect individual wells in the slave reservoirs.

Summary of keywords

SUMMARY section

The following items are relevant to the Gas Calorific Value Control option:

Table 29.2 Gas calorific value control SUMMARY output controls

Field	Group	Well	Information
FGCV	GGCV	WGCV	Gas Calorific Value
FGQ	GGQ	WGQ	Gas molar Quality
FEPR	GEPR	WEPR	Energy Production Rate
FEPT	GEPT	WEPT	Energy Production Total (cumulative)
FESR	GESR		Energy Sales Rate
FEST	GEST		Energy Sales Total (cumulative)
FEDC	GEDC	WEDC	Energy Delivery Capacity
FEDCQ	GEDCQ		Energy DCQ

The energy production rates and totals (FEPR, GEPR, FEPT, GEPT) refer to the **gross** energy produced from the formation, without allowing for consumption. The energy sales rates and totals (FESR, GESR, FEST, GEST) refer to the energy produced, minus the energy injected, minus the energy of any consumed gas.

The energy delivery capacity and DCQ are available only when using the Gas Field Operations model. GEDC and WEDC refer to the group and well energy sales rates when the field is operating at its delivery capacity. GEDCQ should be used only for contract groups in cases where two or more groups have separate gas supply contracts (see keyword GSWINGF). FEDCQ should be used when there is a single contract applied to the FIELD (keyword SWINGFAC).

SCHEDULE section

The keywords are:

Table 29.3 SCHEDULE section

Keyword	Description
CALTRAC	Specifies which tracer is to be equated with the calorific value.
DCQDEFN	When using the Gas Field Operations model, use this to tell ECLIPSE whether the DCQ targets refer to energy or gas rate.
GCALECON	Allows minimum economic energy rates and mean calorific values to be defined for groups and the field.
GCONCAL	Specifies the target calorific value for a group or the field.

Table 29.3 SCHEDULE section

Keyword	Description
GCONENG	Specifies the target energy rate for a group or the field.
GUIDECAL	Scales the guide rates calculated for each well by a function of the well's calorific value.
WCALVAL	Specifies the calorific value of gas at a well.

Geomechanics

Chapter 30

Introduction

Geomechanics provides improved ability to simulate compaction processes that are not available with the standard rock compaction option (see "Rock Compaction" on page 859 of this manual). This gives improved ability to predict subsidence and compaction in a reservoir, and also improved modeling of fluid flow through stress dependent permeability and Biot's constant.

Two separate geomechanics workflows are available:

- "Geomechanics external to ECLIPSE" on page 434
- "Geomechanics internally within ECLIPSE Compositional" on page 437

Geomechanics external to ECLIPSE

The geomechanics workflow which is external to ECLIPSE uses the Schlumberger VISAGE rock mechanics simulator [Ref. 128] to process the results of ECLIPSE, together with rock and fluid elastic and plastic properties, in order to calculate the stresses and strains which will occur in the reservoir due to the changes in pressure, saturation (or composition) and temperature. It can modify the pore volume and permeability (or transmissibility) of the model due to these calculated stresses and strains.

In the VISAGE workflow, there are three main styles of interaction with ECLIPSE:

1 One-way coupling

The results of an ECLIPSE simulation (as stored in the restart files) are used to calculate the stresses and strains. There is no feedback from the computations in VISAGE to the inputs of the ECLIPSE simulation, so the ECLIPSE simulation can be run completely independently of the VISAGE simulation.

2 Explicit coupling

At certain times in the simulation (known as stress times), the ECLIPSE run is stopped, and the results from the restart files are processed within the VISAGE simulator. After the VISAGE simulation at the current stress time is complete, the ECLIPSE run is restarted from this time but with certain properties such as pore volume and permeability (or transmissibility) modified based on the stresses and strains computed by VISAGE.

3 Implicit coupling

The ECLIPSE run is stopped at a stress step report time, and the results in the restart files are processed in VISAGE. The ECLIPSE run is restarted from the previous stress step report time with modified pore volume and permeability (or transmissibility) and the process repeated until the change in pore volume calculated by ECLIPSE over the period agrees with that computed by VISAGE.

The VISAGE workflow in ECLIPSE

From 2008.2 onwards, features have been added to the ECLIPSE 100 and ECLIPSE 300 simulators to assist the VISAGE application in this external geomechanics workflow. The `VISAGE` keyword activates these workflow features.

1 The internal geomechanics workflow is not compatible with the VISAGE workflow. For this reason, all internal geomechanics keywords and restart summary mnemonics are ignored by ECLIPSE if the `VISAGE` keyword is present.

2 The list of report dates at which external geomechanics calculations are required is specified in the `VISDATES` keyword. A list of available report dates from a simulation is requested by running the VISAGE workflow in initialization mode, this is achieved by setting item 1 of the `VISOPTS` keyword to `YES`.

   a In initialization mode, the model is run as though `NOSIM` is present and ECLIPSE writes out a file containing information about all report steps that would be performed in the simulation. This file has the same root name as the simulation, but with an EDESC extension (for example, when running a simulation model NAME.DATA, this file will be NAME.EDESC). Also in initialization mode, all requests for stress steps are ignored; and GRID, INIT and time zero restart files are output.

3 ECLIPSE can be made to exit at a stress step report time (once the restart files have been written) by setting item 2 of the VISOPTS keyword to YES. This feature is turned off by default but it must be turned on in explicit or implicit coupling in the VISAGE workflow.

4 When the field pressure at a report time has changed by more than the relative tolerance (specified in item 4 of the VISOPTS keyword) since the last stress step report time, then an additional stress step will be performed at that report time - this is called a dynamic stress step. Note that this feature must be turned off when repeating the calculations over a stress step interval in the implicit coupling, and any dynamic stress steps which have been performed must be added to the VISDATES list. The dynamic stress step mechanism is turned off by default, but is turned on by setting item 3 of the VISOPTS keyword to YES.

Note The VISAGE suite of products can only handle a maximum of 99 stress steps in its workflow. If you request more than 99 stress steps in the VISDATES keyword, or if a dynamic stress step would be triggered after 99 stress steps have already occurred, then ECLIPSE will exit with an error.

---

5 In order to guarantee that restart files are generated at all of the stress step report times, any FREQ mnemonic specification in the RPTRST keyword is ignored and FREQ=1 (that is, output a restart file at every report step) is enforced by default. If a smaller amount of data is required in the restart files, the frequency of output can be reduced by

a specifying the RPTRST BASIC mnemonic as 3, 4 or 5,

b setting the FREQ mnemonic to the desired output frequency and

c setting VISOPTS keyword item 6 to YES.

Note When modifying the restart output frequency, you must ensure that there will still be restart information available at all required stress step dates.

---

6 In the VISAGE workflow, the minimum requirement for restart file content is that it can be used to restart the run. For this reason, the RPTRST BASIC=2 mnemonic is enforced when the VISAGE keyword is present. In addition, the RPTRST ROCKC mnemonic is enforced so that the modified pore volume and transmissibility multiplier are present in the restart files. By default, any other property requests will be ignored. If additional properties are required in the restart file, these must be specified in the RPTRST keyword, and item 7 of the VISOPTS keyword must be set to YES.

7 Whenever a new stress step is encountered, ECLIPSE writes a list of all stress steps performed to a small file with the same root name as the simulation, but with a VDESC extension (for example, when running a simulation model NAME.DATA, this file will be NAME.VDESC). This information allows the external geomechanics simulator to know which of the requested stress steps have been performed, and also whether any dynamic stress steps have been performed.

a At the beginning of the workflow, this VDESC file needs to be emptied (if it already exists from a previous run). ECLIPSE is instructed to empty this file when the VISAGE initialization is requested by setting item 1 of the VISOPTS keyword to YES.

Summary of keywords

RUNSPEC section

The keywords are:

Table 30.1 RUNSPEC section

Keyword	Description
VISAGE	Activate the VISAGE geomechanics workflow

SOLUTION section

The keywords are:

Table 30.2 SOLUTION section

Keyword	Description
VISDATES	Specify VISAGE geomechanics stress step dates
VISOPTS	Specify VISAGE geomechanics workflow options

Geomechanics internally within ECLIPSE Compositional

Some features of coupling the geomechanics rock stress calculation with the fluid flow include:

1 Flexibility

   a Stress, traction and displacement boundary conditions can be set on external as well as internal surfaces.

   b The grid for the stress calculation is the same grid that is used when calculating fluid flow.

   c Rock mechanical properties are input on a grid by grid basis.

2 Plastic yield function parameters and permeability/Biot constant versus stress tables are set in user-defined geomechanical regions of the grid. The user controls the number of regions and the definition of these regions within the grid.

A finite element method (FEM) is used to calculate the stress. The finite element method retains second order accuracy when the grid is skewed which can arise, for example, when using Corner-Point geometry. Most of the simulator technology is available when running the coupled stress calculation such as the ability to run in black oil, compositional or thermal modes and the full use of well model features.

A plasticity model is available with the finite element calculation to describe the shear failure. Either Mohr-Coulomb or Drucker-Praeger failure criteria can be chosen, with or without hardening.

There is a cap model available for plastic deformation including both a shear failure and a yield in compression. The cap model has a failure surface and an elliptical yield cap that closes the open space between the failure surface and the hydrostatic axis. The yield cap expands in the stress space according to a specified hardening rule.

Initial in-situ stress field induced from overburden plus gravity, or tectonic boundary stresses, or thermal effects can be simulated separated or as a whole.

For this release, there are a number of important restrictions in the finite element option. Radial gridding is not allowed. There is no ability to run parallel calculations. Dual porosity is not available. These are discussed in the section "Restrictions" on page 443 of this manual.

Elastic Stress Equations

A preliminary publication that outlines some of the basic ideas discussed below can be found in [Ref. 87] of the bibliography. A more recent publication can be found in [Ref. 97].

Steady state rock momentum balance equations in the x, y and z directions can be written as:

$$\frac{\partial \sigma_x}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} = 0$$

$$\frac{\partial \sigma_y}{\partial y} + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{zy}}{\partial z} = 0 \qquad \text{[EQ 30.1]}$$

$$\frac{\partial \sigma_z}{\partial z} + \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \rho g = 0$$

Here ρ is the rock density or a combination of rock and fluid reservoir density, and g is the gravitational constant. The elastic normal stresses σ and shear stresses τ can be expressed in terms of strains, ε and γ, as:

$$\sigma_x = 2G\varepsilon_x + \lambda(\varepsilon_x + \varepsilon_y + \varepsilon_z) - \alpha P - (2G + 3\lambda)\alpha_T(T - T_r)$$

$$\sigma_y = 2G\varepsilon_y + \lambda(\varepsilon_x + \varepsilon_y + \varepsilon_z) - \alpha P - (2G + 3\lambda)\alpha_T(T - T_r)$$

$$\sigma_z = 2G\varepsilon_z + \lambda(\varepsilon_x + \varepsilon_y + \varepsilon_z) - \alpha P - (2G + 3\lambda)\alpha_T(T - T_r)$$

[EQ 30.2]

$$\tau_{xy} = G\gamma_{xy}$$

$$\tau_{yz} = G\gamma_{yz}$$

$$\tau_{zx} = G\gamma_{zx}$$

[EQ 30.3]

Note that stresses as defined in equation include the pore pressure and Biot constant and are, therefore, total stresses. Constants G, also known as the shear modulus, and λ are Lamé's constants. They are functions of Young's modulus, E, and Poisson's ratio, ν:

$$G = \frac{E}{2(1+\nu)}, \quad \lambda = \frac{E\nu}{(1-2\nu)(1+\nu)} \ .$$

[EQ 30.4]

Strains $\varepsilon_{x,y,z}$ are defined in terms of displacements in the x, y, z directions, namely u, v and w. Thus

$$\varepsilon_x = \frac{\partial u}{\partial x} \quad \varepsilon_y = \frac{\partial v}{\partial y} \quad \varepsilon_z = \frac{\partial w}{\partial z}$$

$$\gamma_{xy} = \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \quad \gamma_{yz} = \frac{\partial v}{\partial z} + \frac{\partial w}{\partial y} \quad \gamma_{zx} = \frac{\partial w}{\partial x} + \frac{\partial u}{\partial z}$$

[EQ 30.5]

These strains do not form a tensor since the shear strains $\gamma_{xy}, \gamma_{yz}, \gamma_{zx}$ are not multiplied by a factor of 1/2. However, stresses as defined above in do form a tensor. Within the simulator, the rock force is calculated in units of Newtons, lbf, or dynes depending on either `METRIC`, `FIELD` or `LAB` units are used for the simulation.

Gridding

The finite element stress calculation use the same standard corner point or cartesian simulation grid as the flow equations.

For the finite element stress calculation, a variational or Galerkin form of the above equations is set up where each equation in [EQ 30.1] is multiplied by a component of a test function. This is then summed over all components and integrated over the domain of three dimensional grids. Basis functions are piece-wise tri-linear. In all unknown rock displacements are defined at the nodes or corner-points of the grid. This is discussed further in [Ref. 106].

Fluid component conservation, rock volume conservation and volume balance equations

The default porosity-stress relation is conservation of rock volume in the grid block over the time or report step:

$$V^{n+1}(1-\phi^{n+1}) = V^{n}(1-\phi^{n}) \qquad \text{[EQ 30.6]}$$

where V is the bulk volume of the grid block, ϕ is the porosity. This is equivalent to the relation $\delta\phi = -(1-\phi)\delta\varepsilon_b$. A second porosity-stress relation is available and is discussed in the GEOMECH keyword.

Fluid component mass conservation assumes the usual form and is discussed elsewhere (see Aziz and Settari [Ref. 18], and also Quandalle and Sabathier [Ref. 8]).

In order that the fluid volumes fit into the available pore volume, a final fluid phase volume balance equation is included.

Interaction between the fluid and the rock

We follow the convention of ABAQUS [Ref. 85], and calculate a total stress as follows:

$$\sigma_T = \sigma_{eff} - \alpha P \qquad \text{[EQ 30.7]}$$

where σ_T is the total stress, σ_{eff} is the rock effective stress, α is Biot's constant and P is the pore (fluid) pressure. Tensile stresses are positive, compressive are negative. Note that this sign convention holds also when defining tabular functions of permeability and Biot constant versus principal stress.You may also choose to define compressive stresses as positive. This can be set with the 6th item of the keyword GEOMECH.

Variables and solution of the equations

Variables used in the fluid flow simulator are moles of all fluid components per unit pore volume, energy per unit bulk volume if thermal simulation is specified, pressure of one of the fluid phases. Some of these variables are eliminated before entering the solver. In particular, one of the fluid molar densities is eliminated using the volume balance equation. Variables used in the stress calculation are rock displacements in the x, y, z directions, and porosity. The linear solver used to solve the system of stress equations is an algebraic multi-grid solver, [Ref. 98].

Boundary conditions and initialization

Either stress or displacement boundary conditions can be chosen. Each cell of each side of the simulation grid, that is X-/X+ (or equally I-/I+), Y-/Y+ (or J-/J+), and Z-/Z+ (or K-/K+), can have one of the above boundary conditions prescribed. Internal boundaries (cell faces adjacent to internal inactive cells) can also have a load or displacement boundary condition applied.

Note Note however that all boundary conditions cannot be stress based, as this is an ill-posed problem for the displacements solution. If some stress boundaries are set, then part of the reservoir must have some displacements imposed.

Here, + means the outer normal direction of this boundary face is the same as the axis, while - refers to the outer normal direction that is different from the axis. For example, if X axis is from left to right, then X- means the outer normal direction of this boundary face is to the left, X+ refers to the one has right outer normal direction. To set boundaries conditions the user can specify:

- Traction boundary conditions set with the keywords GMTRABC.
- Displacement boundary conditions set with the keywords GMDISBC.
- Principal stress boundary conditions set with the keyword GMPSTBC.

Boundary conditions in local grid refinements can be set with keywords GMTRABCL, GMDISBCL or GMPSTBCL.

Plasticity

The keyword GEOYLDF allows classical yield functions to be set in specified regions of the grid. Either Drucker-Praeger (default) or Mohr-Coulomb failure criteria can be chosen. All of these are defined with a cohesion and angle of internal friction. Hardening laws and parameters can also be set.

Another plastic deformation cap model that can simulate both shear failure and compaction cap is available through keyword GEOYCAP. The cap is defined with a cohesion, angle of internal friction, and material uniaxial tensile strength. The main features of the cap model include a failure surface F_s:

$$F_s = \Gamma + \sqrt{J_2} + C\exp(BI_1) - A = 0 \qquad \text{[EQ 30.8]}$$

and an elliptical yield cap F_c:

$$F_c = R^2\Gamma^2 J_2 + (I_1 - L)^2 - (X - L)^2 = 0 \qquad \text{[EQ 30.9]}$$

that closes the open space between the failure surface and the hydrostatic axis. Here

$$A = \frac{1}{\sqrt{3}}\Gamma\sigma_s \qquad B = \frac{\alpha}{\left(\frac{1}{\sqrt{3}}\sigma_s - k\right)\Gamma} \qquad C = \left(\frac{1}{\sqrt{3}}\sigma_s - k\right)\Gamma \qquad \text{[EQ 30.10]}$$

Several material properties are required to input to define these cap surfaces.

Arbitrary geomechanical regions in the grid can be set using the keyword YLDNUM. Parameters and yield functions are then defined in each region. For example, a region can be set around a particular well.

User Interface

A Young's modulus, Poisson ratio and rock density must be entered for each grid block in the GRID section. A Biot constant may also be entered but, if not, is defaulted to unity. Stress, traction and displacement boundary conditions are entered in both the SOLUTION and SCHEDULE sections. If a plastic calculation (finite element only) is required, a cohesion, angle of internal friction and hardening parameters if desired in different regions of the grid are input in the PROPS section. Further discussion of this option is found in the GEOYLDF keyword.

Initial In-Situ Stress

Initial in-situ stresses induced from overburden stress plus gravity, or tectonic boundary condition or thermal effects can be calculated separately or as a whole.

For gravity-induced stresses, rock density is set in ROCKDEN, and overburden traction is set by OVERBDI in the SOLUTION section. Tectonic loadings are defined from TECBDI in the SOLUTION section.

Using the coupled geomechanics model

Caution The geomechanic rock stress (force) balance equations that are implemented in ECLIPSE 300 are based on the assumption of small strains. Maximum strains should not exceed 5% during the life of the simulation. This restriction applies to all work with this option.

Starting geomechanics

A partly-coupled geomechanics calculation in ECLIPSE 300 is selected using the GEOMECH keyword in the RUNSPEC section.

1. The second argument is an option on how the geomechanics rock force balance is coupled to the other component conservation equations.
   a. By default, this stress calculation is only carried out at every report step. This is the most computationally efficient choice since the reservoir simulation proceeds without calculating the rock stress until a report step is reached at which time stress is updated using the latest fluid (pore) pressures.

   The other choice is:

   b. to update the rock stress at every time step.
2. The third item is choice of the calculation method (obsolete: the supported method is the finite element FE one).

Defining the rock mechanical parameters

In the GRID section, several keywords are used to set the rock mechanical properties.

The Young's modulus, Poisson ratio and rock density are set with the keywords YOUNGMOD, POISSONR and ROCKDEN, respectively. A value of YOUNGMOD and POISSONR is needed for each grid block. Rock density is used to calculate the rock weight contribution to the gravity force as accounted for in the vertical rock force balance equation.

Interaction between the rock and fluid is determined by the Biot constant. A keyword BIOTC sets this for each grid. If not set, then this will default to 1.0 and a message is printed in the output (.PRT) file informing the user of this default. It is also possible to define a multiplier of this constant as a function of principal stress. See the next section.

When running in thermal mode, a coefficient of linear thermal expansion for each grid can be set with the GLTHEX keyword.

Modifying permeability as a function of stress

A permeability multiplier as a tabular function of maximum and minimum principal stress can be input for each geomechanics region. (See YLDNUM and the first argument of GEODIMS). The total number of geomechanics regions is set with the first argument of GEODIMS, and a region number for each grid cell is then defined with the keyword YLDNUM.

Similarly, a multiplier for the Biot constant can be set as a function of maximum and minimum principal stress. A table can be defined with the keyword BIOTCTAB in each geomechanics regions of the simulation.

For both of these multipliers, the multipliers are calculated after every rock stress update if partly coupled, or explicitly at the end of each time step if fully coupled.

Stress boundary conditions

Boundary conditions for the geomechanics rock force (stress) balance are determined with keywords GMDISBC, GMTRABC, or GMPSTBC in the SOLUTION and SCHEDULE section. A boundary condition can be set on the I+ (X+), I- (X-), J+ (Y+), J- (Y-) and K+ (Z+) or K- (Z-) boundaries (faces). Note that K- is considered to be the top of the reservoir, K+ the bottom.

If no boundary condition is entered for a particular face, then that face is assigned a zero-normal-displacement boundary condition. A user-defined boundary condition is either a constant rock displacement or a load (applied stress) on this surface.A load can be supplied as a traction using the GMTRABC keywords or with known principal stress vectors with the keyword GMPSTBC.

The rest of the input file

The remaining part of the ECLIPSE 300 input file with geomechanics remains exactly the same as before.

Performance

If partly coupled with periodic stress updates, then the simulation proceeds until either a report or an end of time step, when an update to the rock stress is requested. At this time the rock force balance equations and porosity-stress relationship are solved. These use the fluid (pore) pressures from the standard simulation and the input Biot constant to determine the total rock stress. Once this calculation is complete, any updates to geometric terms, or stress dependent permeability/Biot constant multipliers are carried out. The reservoir simulation then proceeds with updated permeabilities and porosities until the next geomechanical stress update.

Restrictions

The geomechanics functionality which is available internally within ECLIPSE 300 has the following restrictions.

1. Dual porosity calculations are not available.
2. Radial local grid refinement is not available.
3. Unstructured grids (either global or local) are not available.
4. Parallel is not available.
5. Radial gridding is not available.
6. Coordinates-defined local grid refinement is not available.

Summary of keywords

RUNSPEC section

The keywords are:

Table 30.3 RUNSPEC section

Keyword	Description
GEOMECH	Selects the geomechanics option, allows choices on partly coupled solution of the rock stress equations, stress calculation type, porosity-stress relation, initialization
GEODIMS	Sets the number of geomechanics regions in the simulation, sets table dimensions, in terms of the number of rows and columns, for the PERMSTAB and BIOTCTAB keywords.
SCDPDIMS	Sets dimensions of scale damage tables defined with keyword SCDATABG.

GRID section

The keywords are:

Table 30.4 GRID section

Keyword	Description
BIOTC	Sets the Biot constant for each grid block.
GLTHEX	Sets a coefficient of linear thermal expansion for each grid block
POISSONR	Sets the Poisson ratio for each grid block.
PROPFLOW	Defines parameters to be used with the proppant flowback feature
ROCKDEN	Sets the rock density for each grid block.
YOUNGMOD	Sets the Young's modulus for each grid block.

PROPS section

The keywords are:

Table 30.5 PROPS section

Keyword	Description
BIOTCTAB	Defines a multiplier for the Biot constant, this being a two-dimensional tabular function of maximum and minimum principal stress. A table is entered for each geomechanics region.
GEOYLDF	Defines parameters for the plastic yield function in each geomechanics region.

Table 30.5 PROPS section

Keyword	Description
GEOYCAP	Defines parameters for a shear failure surface, a yield cap in compression, and hardening laws in each geomechanics region.
PERMSTAB	Defines a permeability multiplier, this being a two-dimensional tabular function of maximum and minimum principal stress. A table is entered for each geomechanics region.
SCDATABG	Defines a scale damage table to be used together with the proponent flowback feature, keyword PROPFLOW

REGIONS section

The keywords are:

Table 30.6 REGIONS section

Keyword	Description
YLDNUM	Defines a geomechanics region number for each grid block.

SOLUTION section

The keywords are:

Table 30.7 SOLUTION section

Keyword	Description
DISPBC	Initial displacement velocity (mixed) boundary conditions
DISPBCL	Initial displacement velocity (mixed) boundary conditions in an LGR
GMDISBC	Initial displacement (Dirichlet) boundary condition
GMDISBCL	Initial displacement (Dirichlet) boundary condition in an LGR
GMTRABC	Initial traction (Neumann) boundary condition
GMTRABCL	Initial traction (Neumann) boundary condition in an LGR
GMPSTBC	Initial principal Stress boundary conditions
GMPSTBCL	Initial principal Stress boundary conditions in an LGR
OVERBDI	Overburden stresses for initial gravity induced stress field
OVERBDIL	Overburden stresses for initial gravity induced stress field in an LGR
PSTRBC	Initial principal Stress boundary conditions
PSTRBCL	Initial principal Stress boundary conditions in an LGR
STRESBC	Permits the application of a load or a velocity (rigid) boundary condition on all external and internal surfaces of the simulation grid.
STRESBCL	Initial displacement and traction boundary conditions in a local grid refinement
TECBDI	Initial tectonic boundary condition for initial tectonic induced stresses
TECBDIL	Initial tectonic boundary condition for initial tectonic induced stresses in an LGR

Table 30.7 SOLUTION section

Keyword	Description
TRACTBC	Initial traction (Neumann) boundary conditions
TRACTBCL	Initial traction (Neumann) boundary conditions in a local grid refinement
OUTSOL	Same as RPTRST below.
RPTRST	Controls output for GRAF or RESTART files.

Table 30.8 SOLUTION Mnemonics

Input Keyword Or Mnemonic	Output	E100	E300
COHESION	Cohesion from plastic calculation		
DROCKDIS	Delta rock displacements or change from initial conditions; this produces output with the names DROCKDIX, DROCKDIY, DROCKDIZ		x
EFFSTRES	Rock stresses: this produces arrays with the names EFFSTRXX, EFFSTRYY, EFFSTRZZ, EFFSTRXY, EFFSTRXZ, EFFSTRYZ		x
GENPLSTN	Generalized plastic strain from plastic calculation		
GENPSTRS	Generalized plastic strain multiplied by stiffness matrix		
GOSTRES	Effective rock stresses induced by gravity and overburden		x
PGOSTRES	Total stresses induced by gravity and overburden		x
TCSTRES	Effective stresses induced by initial tectonic stress boundary conditions		x
PTCSTRES	Principal stresses induced by initial tectonic stress boundary conditions		
THSTRES	Effective stresses induced by initial thermal effects		
PTHSTRES	Principal stresses induced by initial thermal effects		
PLASDISP	Plastic displacement		
PRSTRESS	Principal stresses; this produces magnitude output with the names PRSTRES1, PRSTRES2, PRSTRES3, and directions(l1,m1,n1), (l2,m2,n2), (l3,m3,n3)		x
ROCKDISP	Rock displacements; this produces output with the names ROCKDISX, ROCKDISY, ROCKDISZ		x
STRAIN	Strain tensor: this produces arrays with the names STRAINXX, STRAINYY, STRAINZZ, STRAINXY, STRAINXZ, STRAINYZ		x
TOTSTRES	Total stress tensor: this produces arrays with the names TOTSTRXX, TOTSTRYY, TOTSTRZZ, TOTSTRXY, TOTSTRXZ, TOTSTRYZ		x
DEFFSTRS	Delta effective stresses = change from initial conditions; this produces output with the names DEFFSTXX, DEFFSTYY, DEFFSTZZ, DEFFSTYZ, DEFFSTZX, DEFFSTXY.		x

Table 30.8 SOLUTION Mnemonics

Input Keyword Or Mnemonic	Output	E100	E300
DTOTSTRS	Delta total stresses = change from initial conditions; this produces output with the names DTOTSTXX, DTOTSTYY, DTOTSTZZ, DTOTSTYZ, DTOTSTZX, DTOTSTXY		x
DPRSTRES	Delta principal stresses = change from initial conditions; this produces output with the names DPRSTRS1, DPRSTRS2, DPRSTRS3 followed by their respective directions		x
DSTRAIN	Delta strain components = change from initial conditions; this produces output with the names DSTRANXX, DSTRANYY, DSTRANZZ, DSTRANYZ, DSTRANZX, DSTRANXY		x

SCHEDULE section

The keywords are:

Table 30.9 SCHEDULE section

Keyword	Description
DISPBC	Displacement (Dirichlet) boundary conditions
DISPBCL	Displacement (Dirichlet) boundary conditions in an LGR
GMDISBC	Displacement (Dirichlet) boundary condition
GMDISBCL	Displacement (Dirichlet) boundary condition in an LGR
GMTRABC	Traction (Neumann) boundary condition
GMTRABCL	Traction (Neumann) boundary condition in an LGR
GMPSTBC	Principal Stress boundary conditions
GMPSTBCL	Principal Stress boundary conditions in an LGR
PSTRBC	Principal Stress boundary conditions
PSTRBCL	Principal Stress boundary conditions in an LGR
RPTSCHED	Controls output to the print file
STRESBC	Permits the application of a load or a velocity (rigid) boundary condition on all external and internal surfaces of the simulation grid.
STRESBCL	Displacement and traction boundary conditions in a local grid refinement
TRACTBC	Traction (Neumann) boundary conditions
TRACTBCL	Traction (Neumann) boundary conditions in a local grid refinement
WSCTAB	Links a well to a particular scale damage table as defined with SCDATABG. This feature is part of the PROPFLOW option.

Table 30.10 SCHEDULE Mnemonics

Input Keyword or Mnemonic	Output	E100	E300
DROCKDIS	Geomechanics, rock displacements; this produces output with the names DROCKDIX, DROCKDIY, DROCKDIZ		x
EFFSTRES	Geomechanics: output of rock stresses: this produces arrays with the names EFFSTRXX, EFFSTRYY, EFFSTRZZ, EFFSTRXY, EFFSTRXZ, EFFSTRYZ		x
PRSTRESS	Geomechanics, principal stresses; this produces output with the names PRSTRES1, PRSTRES2, PRSTRES3and directions(l1,m1,n1), (l2,m2,n2), (l3,m3,n3)		x
ROCKDISP	Geomechanics, rock displacements; this produces output with the names ROCKDISX, ROCKDISY, ROCKDISZ		x
STRAIN	Geomechanics: output of strain tensor: this produces arrays with the names STRAINXX, STRAINYY, STRAINZZ, STRAINXY, STRAINXZ, STRAINYZ		x
TOTSTRES	Geomechanics: output of total stress tensor: this produces arrays with the names TOTSTRXX, TOTSTRYY, TOTSTRZZ, TOTSTRXY, TOTSTRXZ, TOTSTRYZ		x

Gi Pseudo-compositional Model

Chapter 31

Introduction

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL

This chapter is concerned with the Gi Pseudo-Compositional model for Gas Condensate/Volatile Oil Systems in ECLIPSE 100.

The scope and accuracy of gas condensate modeling using black oil reservoir simulators such as ECLIPSE is now established. Both depletion and gas cycling above the dew point can be modeled and yield an adequate match with the results from multi-component compositional simulators. The main inadequacy with the existing ECLIPSE approach lies with the treatment of gas injection below the dew point where the primary compositional effect - the stripping of liquid components in inverse proportion to their molecular weights - is completely ignored.

The standard black oil model assumes that the saturated hydrocarbon fluid properties are functions of pressure only and disregards any compositional dependence in the saturated fluid PVT properties. As a consequence, when dry gas is injected into a condensate below its dew point the gas continues to re-vaporize liquid at a rate governed only by the ambient pressure. The vapor saturates over a zone whose thickness is of the order of one grid block - in particular all the liquid in the vicinity of the injectors evaporates rapidly. Results obtained with fully compositional simulation models suggest that liquid saturation profiles would vary more slowly with increasing distance from gas injectors.

The simulation of low pressure gas injection into gas condensate reservoirs can be improved in one of two ways. One is to use a multi-component compositional model to track the movement of the individual hydrocarbon components through the grid mesh. At each time step the thermodynamic equilibrium between all the components present in each grid block is computed using an equation-of-state flash calculation or K-value correlations which define the vapor-liquid mole fraction ratios for each component over a wide range of conditions in the simulation.

An alternative approach to modeling low pressure gas injection is to extend the black oil, two pseudo-component model to take account of fluid property changes occurring during gas injection. A number of extensions of the black oil model to treat compositional effects arising during gas injection have been reported in the literature, and all methods hinge on extending the fluid property treatment so that the saturated fluid properties depend not only on pressure but also on an additional parameter which characterizes compositional changes in the reservoir liquid and vapor phases at constant pressure.

The model described here is based on the method proposed by Cook, Jacoby and Ramesh. The extra parameter introduced into the model to represent the light-end stripping effects is a cumulative injection gas flow parameter (Gi) and is analogous to tracking the flow of the dry injection gas around the reservoir grid.

Description of the model

The model requires an extra parameter to be introduced in order to characterize the compositional changes taking place in the reservoir fluids due to gas injection - this is cumulative volume of injection gas which has passed over the reservoir liquid in each grid cell.

The calculation amounts to a simple active gas tracking calculation to determine the cumulative amount of dry injection gas (Gi), which has contacted the oil in each grid cell, and this is the correlating parameter used to determine the fluid property changes at constant pressure. An extra variable has therefore to be stored in each grid cell corresponding to the Gi variable.

The tracking calculation is performed in a novel way. Suppose f_{dr} denotes the fraction of invading gas into a grid cell that is the injected gas (with $R_v = R_{v,\,\mathrm{inj}}$). Then the effective R_v of the invading gas is

$$R_{v,\,\mathrm{inv}} = f_{dr} \cdot R_{v,\,\mathrm{inj}} + (1 - f_{dr}) R_{v,\,\mathrm{cell}} \qquad \text{[EQ 31.1]}$$

which yields the following expression for the fractional injection gas flux into each grid cell:

$$f_{dr} = \frac{R_{v,\,\mathrm{inv}} - R_{v,\,\mathrm{cell}}}{R_{v,\,\mathrm{inj}} - R_{v,\,\mathrm{cell}}} \qquad \text{[EQ 31.2]}$$

If the well injects dry gas ($R_{v,\,\mathrm{inj}} = 0$), this expression reduces to $f_{dr} = 1 - R_{v,\,\mathrm{inv}} / R_{v,\,\mathrm{cell}}$.

The cumulative injection gas flowing into each grid cell (Gi) can then be defined as:

$$\mathrm{Gi} = \int \frac{f_{dr} F_g}{\mathrm{HCPV}} dt \qquad \text{[EQ 31.3]}$$

where

F_g is the surface flow rate of gas

HCPV is the hydro-carbon pore volume of the cell.

The saturated hydrocarbon PVT properties (principally R_s, R_v, B_o and B_g) should now contain a dependence on both P_o and Gi. In practice, it is usually the variation of $R_{v\,\mathrm{sat}}$ and $R_{s\,\mathrm{sat}}$ with Gi that is the most significant and should be least ignored. For example, it is likely that $R_{v\,\mathrm{sat}}$ could be reduced by an order of magnitude before complete vaporization of the liquid in a cell finally occurred. More significantly, the difference in the fluid property variations with Gi from cell to cell in a 1D compositional case is usually found to be small - this tends to suggest that Gi is a good correlating parameter for the purposes of modeling gas injection effects.

The differential equation to be solved for Gi can be easily derived. The incoming gas into each grid cell can be regarded as being composed of two components. Firstly, a component which has the same oil-gas ratio as the gas already present in the cell, and this component is regarded as being in equilibrium with the gas already present in the cell. The other component in the gas is dry ($R_v = 0$) and can cause liquid vaporization.

The inflow of the free gas which is dry is then:

$$f_{dr} \cdot F_g = \left(1 - \frac{R_{v,\,\mathrm{inv}}}{R_{v,\,\mathrm{cell}}}\right) F_g \qquad \text{[EQ 31.4]}$$

If the grid cell i contains a well completion injecting dry gas, there is an additional term to represent the dry gas inflow from the well:

$$f_{drw} \cdot Q_g = \left(1 - \frac{R_{v,\,\mathrm{inj}}}{R_{v,\,\mathrm{cell}}}\right) Q_g \qquad \text{[EQ 31.5]}$$

where

f_{drw} is the dry gas fraction of the well flow

Q_g is the well gas flow rate.

Now the change in Gi from the definition [EQ 31.3] is:

$$\delta\mathrm{Gi} = \frac{f_{dr}F_g + f_{drw}Q_g}{\mathrm{HCPV}}\delta t \qquad \text{[EQ 31.6]}$$

$$\delta\mathrm{Gi} = \frac{(R_v - R_{v,\,\mathrm{up}})F_g\delta t}{R_v \cdot \phi \cdot V \cdot (1 - S_w)} + \frac{(R_v - R_{v,\,\mathrm{inj}})Q_g\delta t}{R_v \cdot \phi \cdot V \cdot (1 - S_w)} \qquad \text{[EQ 31.7]}$$

where

R_v is the vaporized oil-gas ratio

$R_{v,\,\mathrm{up}}$ is the R_v in the upstream cell

$R_{v,\,\mathrm{inj}}$ is the R_v of the injection fluid

V is the cell volume

ϕ is the cell porosity

S_w is the water saturation

In the continuum limit ($\mathrm{VOL} \to 0$, $\delta t \to 0$) the differential equation for Gi in one dimension becomes:

$$\frac{d}{dt}\mathrm{Gi} = \frac{F_g}{\phi \cdot R_v A(1 - S_w)}\frac{dR_v}{dx} \qquad \text{[EQ 31.8]}$$

where

A is the flow area

Data generation procedure

The model requires two-dimensional tables (P, Gi) to be entered for the saturated R_s, R_v, B_o and B_g data tables spanning the range of pressures and Gi parameter values encountered in the simulation. Dependence of the phase viscosities on Gi is assumed to be negligible. Variations of the surface oil and gas gravities with Gi are also ignored.

The new PVT tables are generated by an extension to the standard constant volume depletion procedure usually used to produce black oil PVT data. Instead of simply generating black oil tables by depleting the PVT cell, we now need to deplete the PVT cell to a pressure below the dew point and then inject dry gas into the cell in a series of steps. At each step the mixture is flashed to thermodynamic equilibrium and the excess gas withdrawn until the mixture occupies the same volume before the gas injection step. The new values for R_v, R_s, B_o and B_g are calculated to correspond with the updated value of Gi. The procedure is repeated injecting gas at a lower pressure - in each case the liquid should completely evaporate to yield the full saturated PVT curves.

It is assumed in the model that the data generated by the above method is sufficiently path independent to adequately represent the compositional effects occurring during gas injection (that is, no matter how a given point is reached in (P_o, Gi) space, the R_v, R_s, B_o and B_g values will be the same as that used in the generating procedure).

Note GI model with Gi=0 will not always behave as a black oil model without GI option in it. According to [EQ 31.3] in a GI model, there will be a finite amount of Gi generation whenever there exists the combination of a finite fractional gas flux into a grid cell and a finite gas flow rate in the grid cell. Furthermore, Gi accumulates over time. In a black oil model without GI option, this does not happen.

Using the Gi pseudo-compositional model

The model should only be used at present in oil-water-gas systems. The keyword GIMODEL in the RUNSPEC section is used to activate the option. Gas condensate and live oil must also be declared active phases using keywords VAPOIL and DISGAS in the RUNSPEC section. The IMPES solution method should not be used when the Gi model is active.

The oil and gas PVT properties are specified using the PVTO and PVTG keywords in the usual way. When the Gi model is active, the data specified using the PVTG and PVTO keywords refer to fluid conditions at GI=0. A two-dimensional variation of saturated fluid properties is required for the option to specify the full (P, Gi) dependence of $R_{s\ \mathrm{sat}}$, $R_{v\ \mathrm{sat}}$, $B_{g\ \mathrm{sat}}$ and $B_{o\ \mathrm{sat}}$.

Either the keywords RVGI, RSGI, BGGI, BOGI or the single keyword GIALL are used to specify these variations, and should be defined in the PROPS section. Note that the function values entered under these keywords refer to **multiplier** values on the saturated data contained in the PVTO and PVTG keywords (that is, at Gi=0). The corresponding Gi-nodal values are specified using the GINODE keyword in the PROPS section.

The PVT properties specific to the Gi model can be echoed using the 'GINODE' mnemonic in the RPTPROPS keyword. Output of the values of Gi in each cell at each report time can be produced using the 'GI' mnemonic in the RPTSCHED keyword. Debug from the option can be produced using integer 30 of the DEBUG keyword.

Summary of keywords

RUNSPEC keywords

The keywords are:

Table 31.1 RUNSPEC section

Keyword	Description
GIMODEL	Activates the Gi Pseudo-compositional model.

PROPS section

The keywords are:

Table 31.2 PROPS section

Keyword	Description
BGGI	Variation of saturated gas FVF with pressure and Gi.
BOGI	Variation of saturated oil FVF with pressure and Gi.
GIALL	Variation of saturated properties with pressure and Gi.
GINODE	Gi nodal values.
RPTPROPS	Controls output from the PROPS section. Mnemonics GINODE etc. outputs saturated fluid properties.
RSGI	Variation of saturated GOR with pressure and Gi.
RVGI	Variation of saturated OGR with pressure and Gi.

SOLUTION section

The keywords are:

Table 31.3 SOLUTION section

Keyword	Description
GI	Initial Gi values for enumeration.

SUMMARY section

The following keywords control output of data specific to the Gi model:

Table 31.4 SUMMARY section

Block	Information
BGI	Block Gi value

SCHEDULE section

The keywords are:

Table 31.5 SCHEDULE section

Keyword	Description
RPTSCHED	Controls output from the SCHEDULE section. Mnemonic GI outputs the current Gi values.

Example problem

This problem illustrates the use of the option. The grid is 1-dimensional and the initial fluids consist of connate water and a volatile undersaturated oil. Injection of dry gas begins at t=0 in grid cell (1,1). The production well completed in cell (15,1) operates at constant BHP.

RUNSPEC

RUNSPEC section

```
RUNSPEC

TITLE
 1D Test Case - Volatile Oil Model - Gi Model Activated
DIMENS
 15    1    1  /
NONNC
OIL
WATER
GAS
DISGAS
VAPOIL
GIMODEL
METRIC
EQLDIMS
 1  100   20    1    1 /
ENDSCALE
 'NODIR'    'REVERS'       1    1  /
TABDIMS
 1    1   20   30    1   40 /
WELLDIMS
2    4    1    2 /
START
 1 'JAN' 1987  /
```

GRID

GRID section

```
GRID  ==============================================================
NOGGF
```

```
EQUALS
 'DX' 100.0  /

 'DY' 100.0  /

 'DZ' 10.0   /

 'TOPS' 3995.0 /
 'PORO' 0.25   /
 'PERMX' 200.0  /
/
RPTGRID
 'DX'  'DY'  'DZ'  'PERMX'  'PERMY'  'PERMZ'  'MULTX'
 'MULTY'  'MULTZ'  'PORO'  'NTG'  'TOPS'  'PORV'
 'DEPTH'  'TRANX'  'TRANY'  'TRANZ'  'COORDSYS'  /
```

PROPS

PROPS section

```
PROPS   ============================================================
PVTW
 408.70000   1.05200   5.00E-5   0.26000   0.00E+00 /
```

The following PVTG table describes the wet gas properties corresponding to Gi=0. The saturated gas properties depend only on pressure in the usual way.

```
PVTG
     11.40   0.00057098   0.134826   0.01316 /
     49.20   0.00018704   0.028395   0.01534 /
    107.50   0.00015314   0.012619   0.01765 /
    154.00   0.00018098   0.008819   0.01994 /
    207.30   0.00024415   0.006699   0.02343 /
    248.00   0.00031469   0.005778   0.02685 /
    275.00   0.00037429   0.005357   0.02957 /
    304.30   0.00045362   0.005019   0.03306 /
    330.00   0.00053982   0.004806   0.03675 /
    344.09   0.00058662   0.004713   0.03890 /
    358.09   0.00062914   0.004632   0.04096 /
    371.27   0.00067040   0.004566   0.04301 /
    383.69   0.00071060   0.004514   0.04507 /
    395.41   0.00075159   0.004472   0.04717 /
    406.48   0.00079051   0.004439   0.04925 /
    416.96   0.00082863   0.004413   0.05134 /
    426.88   0.00086703   0.004393   0.05348 /
    436.29   0.00090351   0.004378   0.05560 /
    445.21   0.00093977   0.004366   0.05775 /
    453.68   0.00097372   0.004358   0.05986
             0.00064914   0.004119   0.04974
             0.00032457   0.003904   0.04034
             0.00000000   0.003718   0.03230 /
/
```

The following PVTO table describes the live oil properties corresponding to Gi=0. The saturated oil properties depend only on pressure in the usual way.

```
PVTO
    3.02    11.40   1.095   0.71545  /
   24.84    49.20   1.179   0.57933  /
   56.57   107.50   1.284   0.44463  /
   83.57   154.00   1.367   0.36695  /
  120.00   207.30   1.476   0.29689  /
  153.30   248.00   1.575   0.25293  /
  179.05   275.00   1.652   0.22707  /
  211.49   304.30   1.749   0.20134  /
  245.23   330.00   1.850   0.18027  /
  266.64   344.09   1.915   0.16908
           445.41   1.840   0.18829
           546.74   1.785   0.2062   /
  284.74   358.09   1.966   0.16218  /
  302.84   371.27   2.017   0.15597  /
  320.95   383.69   2.068   0.15036  /
  339.05   395.41   2.119   0.14527  /
  357.15   406.48   2.170   0.14063  /
  375.25   416.96   2.220   0.13638  /
  393.35   426.88   2.271   0.13248  /
  411.45   436.29   2.322   0.12888  /
  429.55   445.21   2.373   0.12555  /
  447.65   453.68   2.423   0.12246
           555.01   2.322   0.13498
           656.33   2.246   0.14656  /
/
```

The Gi node values to be used in the specifying the data points in the RSGI, RVGI, BOGI and BGGI keywords are defined using the GINODE keyword

```
GINODE
    0.0
   10.0
   20.0
   40.0
  120.0
  200.0
  300.0
  500.0
  800.0
/
```

RVGI defines the variation of saturated oil-gas values with pressure and Gi. Note that the entries in this table represent multiplier values for the pressure dependent saturated OGR data entered in the PVTG table.

```
RVGI
 248.0  1.0
        0.95
        0.90
        0.85
        0.80
        0.72
        0.60
        0.50
        0.40
        0.30  /
 416.96 1.0
        0.95
        0.90
        0.85
        0.80
        0.72
        0.60
        0.50
        0.40
        0.30  /
/
```

RSGI defines the variation of saturated gas-oil values with pressure and Gi. Note that the entries in this table represent multiplier values for the pressure dependent saturated GOR data entered in the PVTO table.

```
RSGI
 248.0  1.0
        0.98
        0.95
        0.91
        0.86
        0.82
        0.78
        0.75
        0.72
        0.70  /
 416.96 1.0
        0.98
        0.95
        0.91
        0.86
        0.82
        0.78
        0.75
        0.72
        0.70  /
/
```

BGGI defines the variation of saturated gas FVF values with pressure and Gi. Note that the entries in this table represent multiplier values for the pressure dependent saturated FVF data entered in the PVTG table.

```
BGGI
 248.0  1.0
        0.998
        0.996
        0.994
        0.992
        0.990
        0.988
        0.986
        0.984
        0.982 /
 416.96 1.0
        1.0
        1.0
        1.0
        1.0
        1.0
        1.0
        1.0
        1.0
        1.0   /
/
```

BOGI defines the variation of saturated oil FVF values with pressure and Gi. Note that the entries in this table represent multiplier values for the pressure dependent saturated FVF data entered in the PVTO table.

```
BOGI
 248.0  1.0
        0.995
        0.990
        0.985
        0.980
        0.975
        0.970
        0.965
        0.960
        0.955 /
 416.96 1.0
        0.99
        0.98
        0.96
        0.94
        0.92
        0.90
        0.88
        0.86
        0.84  /
/
```

```
DENSITY
 732.000  1000.05  1.10900 /
SGFN
  0.00    0.0000   0.0
  0.03    0.0000   0.0
  0.110   0.0246   0.0
  0.307   0.1212   0.0
  0.435   0.2684   0.0
  0.491   0.3701   0.0
/
SWFN
  0.221   0.00    0.0
  0.25    0.010   0.0
  0.400   0.070   0.0
  0.547   0.163   0.0
  0.582   0.179   0.0
  0.670   0.260   0.0
  0.731   0.360   0.0
/
SOF3
  0.269   0.000000   0.000000
  0.288   0.000038   0.000000
  0.294   0.000050   0.000380
  0.300   0.000065   0.000740
  0.320   0.000170   0.001400
  0.500   0.035000   0.034000
  0.750   0.470000   0.760000
  0.779   1.000000   1.000000
/
ROCK
 407.800   7.00E-5 /
RPTPROPS
 'GINODE'  /
```

SOLUTION

SOLUTION section

```
SOLUTION ============================================================
PRESSURE
 15*370.0 /
SWAT
 15*0.2 /
SGAS
 15*0.0 /
RS
 15*266.64 /
RV
 15*0.0006 /
```

```
RPTSOL
 'PRES'  'SOIL'  'SWAT'  'SGAS'  'RS'  'RV'  'FIP'
 'EQUIL'  'RSVD'  'GI'  /
```

SCHEDULE

SCHEDULE section

```
SCHEDULE  ==========================================================
WELSPECS
 'GINJ' 'G' 1   1 4000.00 'GAS' 1*  'NO' 'SHUT' 'YES' 1* 'SEG' /
 'PROD' 'G' 15  1 4000.00 'OIL' 0.0 'NO' 'SHUT' 'YES' 1* 'SEG' /
/
COMPDAT
 'GINJ' 1  1 1 1 'OPEN' 0 1* 0.4 /
 'PROD' 15 1 1 1 'OPEN' 0 1* 0.4 /
/
WCONPROD
 'PROD' 'OPEN' 'BHP' 1*  1*  1*  1*  1*  370.0  /
/
WCONINJ
'GINJ' 'GAS' 'OPEN' 'RATE' 150000.0  /
/
RPTSCHED
 'PRES'  'SOIL'  'SWAT'  'SGAS'  'RS'  'RV'  'RESTART=2'
 'FIP'   'WELLS=2'  'SUMMARY=2'  'CPU=2'
 'NEWTON=2'  /
TSTEP
 10*20.0  /
END
```

The Gradient Option

Chapter 32

Introduction

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL

The Gradient option computes gradients of the solution (for example, pressure in a well) with respect to various property parameters defined by the user, such as horizontal permeability in a layer.

The SimOpt program uses the gradients computed by the simulator to achieve an efficient regression during a history match or sensitivity analysis, varying the property parameters via modifiers.

These property parameter modifiers include:

- X-direction transmissibility multipliers input through keyword `HMMULTX` applied to region `HMTRANX`
- Y-direction transmissibility multipliers input through keyword `HMMULTY` applied to region `HMTRANY`
- Z-direction transmissibility multipliers input through keyword `HMMULTZ` applied to region `HMTRANZ`
- X and Y-direction transmissibility multipliers input through `HMMLTXY` applied to region `HMTRNXY`
- Dual porosity sigma factor multipliers input through keyword `HMMULTSG` applied to region `HMSIGMA`.
- Pore volume multipliers input through keyword `HMMULTPV` applied to region `HMPORVM`
- Analytic aquifer strengths through keywords `HMMLCTAQ` applied to `HMAQUCT` (Carter Tracy), and `HMMLFTAQ` applied to `HMAQUFET` (Fetkovich)
- Numerical aquifer pore volume, permeability and transmissibilities input through keyword `HMMLAQUN` applied to region `HMAQUNUM`.
- Transmissibility multipliers across nominated faults through `HMMULTFT` applied to `HMFAULTS`.
- X-direction permeability multipliers input through keyword `HMMLTPX`, applied to region as in `HMPERMX`.

- Y-direction permeability multipliers input through keyword HMMLTPY, applied to region as in HMPERMY.
- Z-direction permeability multipliers input through keyword HMMLTPZ, applied to region as in HMPERMZ.
- X and Y-direction permeability multipliers input through HMMLTPXY, applied to region as in HMPRMXY.
- Saturation table end-points: HMSWCR, HMSGCR, HMSOWCR, HMSOGCR, HMSWL.
- Multipliers on vertical scaling of relative permeability: HMKRO, HMKRW, HMKRG, HMKRORW, HMKRORG.
- Multipliers on vertical scaling of capillary pressure: HMPCW, HMPCG.
- Rock compressibility multipliers input through HMMROCK applied to HMROCK, and modifiers on rock compaction parameters through HMMROCKT applied to HMROCKT.
- Inter-region transmissibility multipliers between nominated regions using keyword HMMMREGT applied to HMMULRGT.
- Multipliers on the well productivity index using keyword HMWPIMLT.
- Multipliers on the well connection transmissibility factor and skin using the keyword HMMLTWCN applied to the connections specified in HMWELCON.

All the above parameters may be used with ECLIPSE 100. In this release of ECLIPSE 300 only pore volume, transmissibility and permeability parameters are available.

The derivatives are written to a file (extension .HMD) which can be interpreted by SimOpt.

The data contained within the .HMD file is generally encoded and is meaningless unless the SimOpt program is used. A facility for unencoded gradients is also available and is controlled by the licensing system. Please contact your ECLIPSE account manager for details.

Output from the Gradient Option

Gradients of observable well quantities (for example, WBP, WOPR) are written to the HMD file at each time step and these can be used by SimOpt to match against observed data using a gradient based optimization approach.

The GRADWELL keyword specifies a list of well summary vectors for which derivatives are required.

Wells requiring gradient output are nominated in the RPTHMW keyword

ECLIPSE 100 only

It is also possible to output gradients of simulated RFT data to the GRFT file to match against observed RFT tests by using the GRADRFT keyword.

ECLIPSE 100 only

If required, it is also possible to output gradients of reservoir grid block solutions (SWAT, SGAS, SOIL, RS, RV, PRESSURE, DENO, DENW, DENG, PORO), as well as seismic impedances (ACIP, ACIS), to the RESTART files, using the GRADRESV keyword. Currently, this is only possible using the licensing system. Please contact your ECLIPSE account manager for details. The mnemonics available in the GRADRESV keyword are:

SWAT Water saturation

SGAS Gas saturation

SOIL Oil saturation

RS Dissolved gas-oil ratio

RV Vaporized oil-gas ratio

PRESSURE Oil phase pressure

DENO Oil phase gravity density at reservoir conditions

DENG Gas phase gravity density at reservoir conditions

DENW Water phase gravity density at reservoir conditions

PORO Porosity at reservoir conditions

ACIP Pressure-wave impedance

ACIS Shear-wave impedance

Using the facility

The facility is initiated by setting the appropriate number of property parameters in RUNSPEC keyword HMDIMS on which gradients are calculated.

You should supply region tables in the REGIONS section using keywords HMPORVM, HMTRANX etc. to delimit the property parameter regions on the simulation grid.

ECLIPSE 100 only Alternatively, for faults which have been specified using the FAULTS keyword, property parameters may be requested using the HMFAULTS keywords in the GRID section.

ECLIPSE 100 only Gradients for analytic aquifer properties may be requested by using the keywords HMAQUFET or HMAQUCT in the SOLUTION section, whereas gradients for numerical aquifer properties may be requested by using the keyword HMAQUNUM in the GRID section.

ECLIPSE 100 only Gradients for rock compaction and compressibility parameters can be obtained by specifying HMROCK and HMROCKT in the PROPS section.

ECLIPSE 100 only To obtain gradients of the solution with respect to inter-region transmissibility multipliers declared using keyword MULTREGT in the GRID section, the HMMULRGT keyword can be used in the GRID section.

ECLIPSE 100 only To obtain gradients of the solution with respect to well connection transmissibility factor and skin multipliers declared using keyword HMMLTWCN in the SOLUTION section, the HMWELCON keyword can be used in the SOLUTION section.

For each gradient property parameter, ECLIPSE calculates derivatives of well-flow quantities (as specified with the GRADWELL keyword) for all active wells. This information is written to the file ROOT.HMD, where ROOT is the base name of the run.

ECLIPSE 100 only Additional information can be written out to the ROOT.HMD file if requested; for example, refer to the GRADRESV keyword.

ECLIPSE 100 only If a satisfactory match has been obtained using modifiers based on the gradient analysis, then the model may be used for prediction without the overhead in calculating the derivatives, simply by specifying the NOHMD (or NOHMO) keyword in the SCHEDULE section (which switches off all gradient calculations). This is the mechanism for a ‘evaluation’ run in SimOpt.

Current restrictions

ECLIPSE 100 only

1 The Gradient option may not at present be used with the Parallel option or with Reservoir Coupling.

ECLIPSE 100 only

2 The effects of active tracers (API, Brine and the Temperature option) and Environmental Tracers or the Foam model are not currently accounted for in a rigorous fashion while calculating gradients. Changes in solution properties caused by active tracers first influence the gradient calculation one time step after they influence the fluid flow properties, that is the active tracers are treated explicitly in the gradient calculation. In most cases, this effect should be very small and can often be neglected so that it is still possible to perform a sensitivity run to screen redundant parameters or enlist a regression run. ECLIPSE no longer disallows a run with active tracers, but the gradients will not be fully accurate.

ECLIPSE 100 only

3 LGRs employing local time-stepping are not currently available in Gradient runs. However, the Gradient option is fully compatible with the LGR-in-place option, and also with the grid coarsening option. In many applications, it is possible to get an initial sensitivity analysis on a coarsened grid to screen for redundant parameters.

ECLIPSE 100 only

4 Gradients cannot at present be obtained with the Vertical Equilibrium option (`VE` or `COMPVE`), with the Gravity Drainage option (`GRAVDR` or `GRAVDRM`) in dual porosity runs, or with the water-induced rock compaction options. This is because the gradient calculation relies on the fully implicit nature of ECLIPSE; the VE option treats the pseudo-capillary pressure explicitly and the extra derivative terms are not accounted for. ECLIPSE no longer disallows a run with VE or gravity drainage, but the gradients will not be fully accurate as approximations have been made in the gradient calculation.

5 Gradients cannot at present be obtained for multi-segmented injector wells.

ECLIPSE 300 only

6 Gradients cannot be used with the Grid Coarsening Option.

Current Uses (ECLIPSE 300 only)

1 It is possible to use the Gradient option in both Black Oil and Compositional runs.

2 In addition, this option can be used in Fully Implicit and AIM (Adaptive IMplicit) modes.

3 The Gradient option may be used with the Parallel option in ECLIPSE 300.

Example

To set three transmissibility multiplier parameters (with NX=3 NY=3 NZ=3), set the number of parameters, gradient regions and sub-regions using `RUNSPEC` keyword `HMDIMS`

```
HMDIMS
  1 1 3 /
```

and set appropriate keywords in the `REGIONS` section

```
HMTRANZ
 3*0 3*1 3*0 /
COPY
 'HMTRANZ' 'HMTRANX' /
/
EQUALS
 'HMTRANY' 9*1 /
/
```

Example

To set one pore volume multiplier parameter (with NX=3 NY=3 NZ=3), set the number of parameters, gradient regions and sub-regions using RUNSPEC keyword HMDIMS

```
HMDIMS
  1 1 1 /
```

and set appropriate keyword in the REGIONS section

```
HMPORVM
 3*0 3*1 3*0 /
```

Data file overview

RUNSPEC section overview

The following keywords are specific to the Gradient option

FMTHMD Declares that the HMD output file containing the derivatives is to be formatted. The default is unformatted.

HMDIMS Specify the dimensions for the Gradient option.

RPTHMD Controls on output to the HMD file.

UNCODHMD Requests that the HMD file contains unencoded gradients.

Example 1 (ECLIPSE 100 only)

In the following example there is a maximum of 3 gradient regions, each with up to 4 subregions with only 5 gradients required and 5 faults (with gradients required for two faults). There are two analytic aquifers with up to 100 grid block connections each and a default influence table.

```
FAULTDIM
--MFSEGS
    5    /
AQUDIMS
--MXNAQN  MXNAQC  NIFTBL  NRIFTB  NANAQU NCAMAX
    0       0       1       36       2        100  /
HMDIMS
--MHISTM  NHMSTR  NHRPAR  MFHIST  MHAQUF
    3       4       5       2       2   /
RPTHMD
 0 1 1 1 /
UNCODHMD
```

Example 2

In the following example there is a maximum of 3 gradient regions, each with up to 4 subregions with only 5 gradients required.

```
HMDIMS
--MHISTM  NHMSTR  NHRPAR
    3       4       5 /
UNCODHMD
```

SUMMARY section overview (ECLIPSE 100 only)

Gradient calculation performance keywords

The following keywords measure performance of the gradient calculation and take no data:

TCPUTSHT CPU time per time step for all the gradient calculations.

TCPUHT Total CPU time for all the gradient calculations.

The following keywords measure performance for individual gradient parameters:

TCPUTSH CPU time per time step for each gradient calculations.

TCPUH Total CPU time for each gradient calculations.

HLINEARS Number of linear iterations for each gradient calculation.

HSUMLINS Cumulative number of linear iterations for each gradient parameter.

The keywords TCPUTSH, TCPUH, HLINEARS and HMSUMLINS should each be followed by a list of gradient parameter names, terminated by a slash (/). If the entire list is defaulted, the relevant quantities are output for all parameters.

Allowable parameter names are:

- Any gradient region keywords HMxxxxxx entered in the REGIONS section
- HMFAULTS
- HMAQUCT, HMAQUFET and HMAQUNUM
- HMROCK etc.
- HMMULRGT

Example

Total CPU time spent on the gradient calculations

```
TCPUH
/
```

Number of linear iterations to solve for each parameter associated with HMPORVM and each parameter for Carter-Tracy aquifers declared using HMAQUCT.

```
HLINEARS
  'HMPORVM'  'HMAQUCT'  /
```

```
HMSUMLINS
  'HMPORVM'  'HMAQUCT'  /
```

Special extensions

When unencoded derivatives are enabled, some extra summary keywords are available: these mnemonics have first character ‘Q’ and are only activated if the HMD file is unencrypted. The Q* mnemonics should be specified at the end of the summary section, after all the normal ECLIPSE SUMMARY keywords. These may be loaded into GRAF as separate RSM file if required. Note that GRAF does **not** distinguish between QWBP for different wells etc. and treats all the Q* mnemonics as user mnemonics.

Analytic and Numerical Aquifer derivatives

These are analogous to AAQR, AAQT and AAQP for analytic aquifers (and ANQR, ANQT, ANQP for numerical aquifers):

Derivative of influx rate for analytic aquifers:

```
QAAQR
 iaq ihm /
```

Derivative of total influx for analytic aquifers:

```
QAAQT
 iaq ihm /
```

Derivative of aquifer pressure (Fetkovich only):

```
QAAQP
 iaq ihm /
```

Derivative of influx rate for numerical aquifers:

```
QANQR
 iaq ihm /
```

Derivative of total influx for numerical aquifers:

```
QANQT
 iaq ihm /
```

Derivative of average numerical aquifer pressure:

```
QANQP
 iaq ihm /
```

where

iaq = aquifer number as specified in `AQUCT`, `AQUFETP` or `AQUNUM` (this may be defaulted to refer to all aquifers,

and

ihm = number of derivative for aquifer (this item may be defaulted to refer to all parameters for a given aquifer).

For a Carter-Tracy aquifer, *ihm* is used as follows:

1 - derivative with respect to permeability multiplier

2 - derivative with respect to opening angle multiplier

3 - derivative with respect to aquifer depth.

For a Fetkovich aquifer:

1 - derivative with respect to multiplier on initial water volume

2 - derivative with respect to Productivity Index multiplier

3 - derivative with respect to aquifer depth.

For a Numerical aquifer:

1 - derivative with respect to multiplier on total aquifer pore volume

2 - derivative with respect to aquifer permeability multiplier

3 - derivative with respect to aquifer to grid transmissibility multiplier.

The derivatives are only calculated when the aquifer parameters `HMAQUCT`, `HMAQUFET` or `HMAQUNUM` are selected.

Example

```
QAAQR
 1 2 /
 2 2 /
 3 1 /
/
```

Derivatives of well quantities

For derivatives of WBP with respect to all gradient parameters

```
QWBP
 WELL1 /
 WELL2 /
/
```

The available mnemonics are:

QWBP derivative of WBP

QWBP4 derivative of WBP5

QWBP9 derivative of WBP9

QWOFR derivative of WOPR or WOIR

QWWFR derivative of WWPR or WWIR

QWGFR derivative of WGPR or WGIR

QWWCT derivative of WWCT

QWGOR derivative of WGOR

QWBHP derivative of WBHP

The corresponding quantities in local grids are preceded by the letter L, for example:

```
LQWBP
 LGR1  WELL1 /
 LGR2  WELL2 /
/
```

To get derivatives of WBP for all parameters at each well:

```
QWBP
/
WBP
/
RUNSUM
SEPARATE
RPTSMRY
 1 /
```

Derivatives of grid block quantities

The available mnemonics are:

QBPR derivative of BPR

QBRS derivative of BRS

QBRV derivative of BRV

QBOSAT derivative of BOSAT

QBWSAT derivative of BWSAT

QBGSAT derivative of BGSAT

QBNSAT derivative of BNSAT.

For example:

```
QBPR
  7 1 1 idm /
 17 1 1 idm /
/
```

where *idm* is the index of the derivative. If *idm* is defaulted, then block derivative for all parameters are obtained. The *idm* parameter is the internal ECLIPSE numbering of each parameter. Usually, *idm* is defaulted and gradients for every parameter are output.

The corresponding quantities in local grids are preceded by the letter L, for example:

```
LQBPR
 'LGR1'    7 1 1 idm /
 'LGR2'   17 1 1 idm /
/
```

Plotting gradients in GRAF

GRAF does not recognize the Q* and LQ* mnemonics and does not prompt for them. Therefore it cannot prompt for well name, parameter etc. Hence if there are several mnemonics, QWBHP for example, each referring to a separate parameter, only the first can be plotted. However, by renaming the mnemonics in the SMSPEC file to be distinct, such as WBHPQ1, WBHPQ2, etc. for parameters 1 and 2, GRAF prompts for well names.

If OPTIONS keyword switch 69 is set equal to 1, then ECLIPSE modifies the names on the SMSPEC file by moving QWBHP for parameter 1 to the mnemonic WBHPQ1 so that is recognized by GRAF as a well mnemonic. A listing of the gradient mnemonics used in a run can be obtained by using the RPTSMRY keyword in the SUMMARY section.

Output from the Gradient option

The derivatives of wellbore quantities generated by the Gradient option are output to the file `ROOTNAME.HMD` (see "File Handling in ECLIPSE" on page 303). The output is generated by the standard ECLIPSE output routines and typically looks like:

```
 'CODESTAT'           1 'CHAR'
 'NOTCODED'
 'E100VER '           1 'INTE'
        901
 'E100INFO'           4 'CHAR'
 '27/05/96' '12:37:00' 'test ' 'test '
 'E100ORIG'           3 'CHAR'
 'May 26  ' '1996    ' '03:06:53'
 'GRADWELL'           9 'CHAR'
 'WOPR    ' 'WWPR    ' 'WGPR    ' 'WWCT    ' 'WGOR    ' 'WBP     ' '
WBP4    '
 'WBP5    ' 'WBP9    '
 'TIMESTEP'           2 'DOUB'
   0.18260000000000D+04   0.31000000000000D+02
 'NAMES   '           5 'CHAR'
 'PRO     ' 'HMPORVM ' '        ' 'PRODUCER' 'OPEN    '
 'HMHEADI '           2 'INTE'
          1           6
 'DERIVS  '           9 'DOUB
   0.00000000000000D+00   0.00000000000000D+00   0.00000000000000D+0
0
   0.00000000000000D+00   0.00000000000000D+00   0.26393554627632D+0
0
   0.26588984672034D+00   0.26491269649833D+00   0.26491269649833D+0
0
 'VALUES  '           9 'DOUB'
   0.00000000000000D+00   0.00000000000000D+00   0.00000000000000D+0
0
   0.00000000000000D+00   0.00000000000000D+00   0.20023871765242D+0
3
   0.20038969805460D+03   0.20031420785351D+03   0.20031420785351D+0
3
 'NAMES   '           5 'CHAR'
 'PRO     ' 'HMTRANX ' '        ' 'PRODUCER' 'OPEN    '
 'HMHEADI '           2 'INTE'
           1           1
 'DERIVS  '           9 'DOUB'
   0.00000000000000D+00   0.00000000000000D+00   0.00000000000000D+0
0
   0.00000000000000D+00   0.00000000000000D+00  -
```

```
0.15196542044025D+00
  -0.18932798370102D+00  -0.17064670207064D+00  -
0.17064670207064D+00
 'NAMES   '           5 'CHAR'
 'PRO     ' 'HMTRANZ ' '        ' 'PRODUCER' 'OPEN    '
 'HMHEADI '           2 'INTE'
           1           4
 'DERIVS  '           9 'DOUB'
   0.00000000000000D+00   0.00000000000000D+00   0.00000000000000D+0
0
   0.00000000000000D+00   0.00000000000000D+00   0.35438853150093D-
02
   0.19550833005848D-02   0.27494843077971D-02   0.27494843077971D-
02
```

Record 'TIMESTEP'

This record consists of 2 items (`DOUBLE PRECISION`)

Item 1 Current time

Item 2 Last time step used

Subsequent records are written in the order shown in Figure 32.1

Figure 32.1 Order for writing records

Record 'NAMES'

This record consists of 5 items (`CHARACTER*8`)

Item 1 Well name

Item 2 Name of Gradient parameter. This is the keyword name associated with the parameter. For example:

HMxxxxxx in the REGIONS section (for example, HMPORVM, HMPERMX etc.

HMFAULTS in the GRID section

HMMULRGT in the GRID section

HMAQUFET in the SOLUTION section

HMAQUCT in the SOLUTION section

HMKRW etc. in the PROPS section

HMSWCR etc. in the PROPS section

HMROCK etc. in the PROPS section

Item 3 Local grid name or blank for a well in the global grid

Item 4 Status of the well:

'PRODUCER' or 'INJECTOR'

Item 5 Status of the well:

'OPEN'

or 'STOPPED'

or 'SHUT'

Note Item 2: Only REGIONS section keyword HMPORVM is available at present.

Record 'HMHEADI'

This record consists of three items (INTEGER)

Item 1

For HMxxxxxx (REGIONS parameters) - the index to the current subregion

For HMFAULTS - the number of the fault (in the data order)

For HMAQUFET - the number of the Fetkovich aquifer

For HMAQUCT - the number of the Carter-Tracy aquifer

For HMAQUNUM - the number of the numerical aquifer

For HMROCK - the number of the PVTNUM or SATNUM region

For HMROCKT - the number of the ROCKNUM region

For HMMULRGT - the number of the first MULTNUM region

Item 2 The type of Gradient parameter:

1 HMTRANX

2 HMTRNXY

32 HMTRANY

4 HMTRANZ

5 HMSIGMA

6 HMPORVM

7 HMTRNX-

8 HMTRNY-

9 HMTRNZ-

12 HMPERMX

13 HMPRMXY

14 HMPERMY

15 HMPERMZ

101 HMFAULTS

102 HMAQUFET initial water volume multiplier

103 HMAQUFET productivity index multiplier

104 HMAQUFET aquifer depth

105 HMAQUCT permeability multiplier

106 HMAQUCT opening angle multiplier

107 HMAQUCT aquifer depth

108 HMWPIMLT

110 HMMULRGT

131 HMAQUNUM pore volume multiplier

132 HMAQUNUM permeability multiplier

133 HMAQUNUM transmissibility multiplier

297 HMSWCR

298 HMSGCR

299 HMSOWCR

300 HMSOGCR

301 HMSWL

305 HMKRW

306 HMKRG

307 HMKRO

309 HMKRWR

310 HMKRGR

311 HMKRORG

312 HMKRORW

313 HMSWLPC

314 HMSGLPC

315 HMPCW

316 HMPCG

1041 HMROCK

1042 HMROCKT

Item 3 For HMMULRGT - the number of the second MULTNUM region

Not used for all other parameters.

ECLIPSE 300

Note Item 1: Only REGIONS section keyword HMPORVM is available at present

Record 'DERIVS'

This record consists of a variable number of items (DOUBLE PRECISION). The record contains the derivatives with respect to each Gradient parameter of nominated well quantities. By default these are:

ECLIPSE 300

Item 1 Oil rate at surface conditions

- UNITS: sm^3/day (METRIC), stb/day (FIELD), scm^3/hr (LAB)

ECLIPSE 300

Item 2 Water rate at surface conditions

- UNITS: sm^3/day (METRIC), stb/day (FIELD), scm^3/hr (LAB)

ECLIPSE 300

Item 3 Gas rate at surface conditions

- UNITS: sm^3/day (METRIC), Mscf/day (FIELD), scm^3/hr (LAB)

ECLIPSE 300

Item 4 Water cut

- UNITS: sm^3/sm^3 (METRIC), stb/stb (FIELD), scm^3/scm^3 (LAB)

ECLIPSE 300

Item 5 Gas Oil ratio

- UNITS: sm^3/sm^3 (METRIC), Mscf/stb (FIELD), scm^3/scm^3 (LAB)

Item 6 1 point pressure average (see WPAVE and WWPAVE)

- UNITS: barsa (METRIC), psia (FIELD), atma (LAB)

Item 7 4 point pressure average (see WPAVE and WWPAVE)

- UNITS: barsa (METRIC), psia (FIELD), atma (LAB)

Item 8 5 point pressure average (see WPAVE and WWPAVE)

- UNITS: barsa (METRIC), psia (FIELD), atma (LAB)

Item 9 9 point pressure average (see WPAVE and WWPAVE)

- UNITS: barsa (METRIC), psia (FIELD), atma (LAB)

This can be changed: see the comments below on GRADWELL.

Record 'VALUES'

This record consists of a variable number of items (DOUBLE PRECISION).

This record contains the values of the well quantities referred to in the 'DERIVS' record. This is only output once for each well at each step, so if there are several Gradient parameters, the values are only output for the first parameter. The reported rates are production rates, so a negative sign indicates injection of the pertinent phase. When a well is shut, the 'DERIVS' and 'VALUES' records are not calculated and may be reported as zeroes.

ECLIPSE 100 only

As writing all this information can be quite voluminous, the amount of output may be configured using the RPTHMD keyword (to supply output only for working producers, etc.) or by the RPTHMW keyword (to switch off output for nominated wells), or only at report times, etc.

This data is used by SimOpt. Generally, the derivatives are unformatted and encoded (though the 'VALUES' record is not encoded). The 'CODESTAT' record outputs whether the file is encoded or not. SimOpt must be used to interpret the derivatives. However, the output of unencoded derivative information is controlled by the password mechanism for ECLIPSE software - please contact your ECLIPSE account manager for more information.

The following records provide version control:

ECLIPSE 100 only

- The records E100VER, E100INFO and E100ORIG provide version control. E100VER is the version of ECLIPSE; E100INFO lists the date and time at which the HMD file was created together with the dataset root name; and E100ORIG reports the date and time at which the current copy of ECLIPSE was built.

ECLIPSE 300 only

- The records E300VER, E300INFO and E300ORIG provide version control. E300VER is the version of ECLIPSE; E300INFO lists the date and time at which the HMD file was created together with the dataset root name; and E300ORIG reports the date and time at which the current copy of ECLIPSE was built.
- The convert utility may be used to convert from unformatted to formatted if required; the formatted HMD file has the extension FHMD (UNIX) or FHM (PC). The RUNSPEC keyword FMTHMD requests a formatted Derivatives file.

Alternative output of well quantities

The above default quantities loaded into the 'DERIVS' and 'VALUES' records may be inappropriate. The GRADWELL keyword allows you to specify a list of mnemonics (such as WBP, WWCT, WBHP etc.) to be output for each well. If the GRADWELL keyword has been used, then extra items are written at the top of the file to specify the GRADWELL data.

For example,

```
'GRADWELL'           3 'CHAR'
'WBP     ' 'WCT     ' 'WBHP    '
```

ECLIPSE 100 only

By default, if GRADWELL is not specified, then nine items are written as in the above example.

When a well connects across poorly-communicating layers which are not in vertical equilibrium, the WBP mnemonics are appropriate. It will be necessary to output completion properties to match measured RFT data.

Receiving HMD files from ECLIPSE through PVM (ECLIPSE 100 only)

The HMD file data are sent back from ECLIPSE to the master process if IHEAD(14) = 1 or 2 in the run header data, when using the Open-ECLIPSE protocol.

ECLIPSE sends the HMD file in a set of message packets:

1. A 'MESS' header with keyword 'GRAD'
2. Derivatives for each parameter
3. A 'MESS' header with keyword 'END'

The above packets are within the loop over all parameters at each output time. An optional 'MESS' header 'GRADEND ' signifies the HMD output has finished before run termination.

The RFT file and RFT gradients are sent back from ECLIPSE to the master process if IHEAD(16) = 1 or 2 in the run header data, when using the Open-ECLIPSE protocol.

ECLIPSE sends the RFT data in a set of message packets:

1 A 'MESS' header with keyword 'RFT'

2 RFT data for a particular well

3 A 'MESS' header with keyword 'END'

The RFT gradients are similar:

1 A 'MESS' header with keyword 'GRADRFT'

2 RFT gradient data for a particular well and parameter

3 A 'MESS' header with keyword 'END'.

The above packets are within the loop over all parameters at each output time. An optional 'MESS' header 'GFTEND' signifies the RFT gradients output has finished before run termination.

Reservoir gradients (ECLIPSE 100 only)

It is also possible to request output of gradients of reservoir pressures, saturations, GOR/OGR, densities etc. These may be used to form an objective function in conjunction with a petro-elastic model in order to compare predicted seismic response. Such a case might be the calculation of seismic impedance.

Furthermore, it is now possible to request output of gradients of pressure-wave impedance and shear-wave impedance, if the appropriate keywords have been specified in the dataset to utilize the petro-elastic model inside the simulator. The information output is controlled by the password mechanism for ECLIPSE software - please contact your ECLIPSE account manager for more information.

Output of derivatives of grid block pressures, saturations, etc. is controlled by the GRADRESV keyword.

A record 'GRADRESV' is written to the HMD and RESTART files to list the mnemonics. For example, referring to the above example

```
'GRADRESV'           2 'CHAR'
'GPRES   ' 'GSWAT   ' /
/
```

or

```
'GRADRESV'           4 'CHAR'
'GPRES   ' 'GSOIL   ' 'GSWAT   ' 'GSGAS   '/
/
```

The solution gradient data is written to the RESTART file in the following manner (see "ECLIPSE File Formats Reference Manual" for details of how restart data is written from ECLIPSE).

Header GRADSOL indicates a solution gradient mnemonic:

For example

```
'GRADSOL '            6 'CHAR'
'HMTRANZ ' 'GRADZONE' '        ''1      ' '1       ' '3       ' /
/
```

The record GRADSOL has up to 10 items:

Item 1 Parameter name

Item 2 Keyword name associated with parameter

'GRADZONE' for regional parameters defining gradzones

'FAULTS' for faults

'AQUCTS '/'AQUFET' for aquifers

'SATNUM' relative permeability shape parameters

'PVTNUM'/'SATNUM'/'ROCKNUM' for rock parameters

'MULTNUM' for inter-region transmissibility multipliers

Item 3 Feature name

Blank for regional parameters

Name of fault for fault parameters

Blank for all other parameters

Item 4 Parameter type

This is described under HMHEADI.

Item 5 Region ID or Feature number

This is the subregion number for regional parameters

Fault number for fault parameters

Aquifer number for aquifer parameters

Saturation table number for relative permeability shape parameters

Rock table number for rock parameters

First region number for inter-region connection parameters

Item 6 Associated phase

This is a pointer internal to ECLIPSE.

Item 7 Alternate region ID

This is the second region number for inter-region connection parameters

Blank for all other parameters

Items 8 to 10 Not used at present.

Following each ‘GRADSOL’ record, the solution gradients are written in the following form:

```
 'GRADSOL '           6 'CHAR'
'HMTRNXY ' 'GRADZONE' '        ' '3       ' '3       ' '1       '
'GPRES_13'         300 'REAL'
   0.45278375E+03   0.40524387E+03   0.35893173E+03  -0.87420359E+01
  -0.12985765E+02  -0.22687635E+02  -0.40819241E+02  -0.70829422E+02
  -0.11564088E+03  -0.16381274E+03   0.19503807E+03   0.19354639E+03
   0.18682153E+03  -0.16461343E+01  -0.25963242E+01  -0.47793212E+01
```

‘The ‘GPRES_nn’ mnemonic indicates oil pressures. The names in GRADRESV have been preceded by ‘G’ so they are distinct from names ‘SGAS’, ‘SWAT’ that ECLIPSE and GRAF use for the solution values.

The header is of the form ‘G’//name(1:4)//number: Characters 5 to 8 in the header is just a unique number such that each GPRES*** mnemonic is unique to any post-processing application that does not require the ‘GRADRESV’ data.

Gradient option technical description

A brief description of the method for deriving gradients in the simulator is outlined in this section. For this overview, the focus is on the fully implicit method.

Obtaining gradients in ECLIPSE

At each simulator time step, ECLIPSE forces the fully implicit residual in each simulator grid block to be zero (see "Formulation of the Equations" on page 341).

This can be written as

$$R = \frac{M_{t+dt} - M_t}{dt} + F_{t+dt} + Q_{t+dt} = 0 \qquad \text{[EQ 32.1]}$$

where the subscripts t and $t+dt$ denote quantities at the current and advance time, with a simulator time step of dt,

M is the fluid-in-place in each simulator grid block

F is the net flow rate into neighboring grid blocks

Q is the net flow rate into wells during the time step.

For any gradient parameter λ, we have, from [EQ 32.1],

$$R(X_{t+dt}, \lambda) = \frac{M_{t+dt}(X_{t+dt}, \lambda) - M_t(\lambda)}{dt} + F_{t+dt}(X_{t+dt}, \lambda) + Q_{t+dt} \qquad \text{[EQ 32.2]}$$

where X is the set of solution variables in each grid block.

For ECLIPSE 100, X represents pressure and saturations, and for ECLIPSE 300, X represents pressure and molar densities.

Differentiating [EQ 32.2] throughout with respect to λ yields

$$\frac{\partial R}{\partial X_{t+dt}} \frac{\partial X_{t+dt}}{\partial \lambda} = -\frac{\partial R}{\partial \lambda} \qquad \text{[EQ 32.3]}$$

which can be inverted as

$$\frac{\partial X_{t+dt}}{\partial \lambda} = -\left(\frac{\partial R}{\partial X_{t+dt}}\right)^{-1} \frac{\partial R}{\partial \lambda} \qquad \text{[EQ 32.4]}$$

For a fully implicit discretization,

$$\frac{\partial R}{\partial \lambda} = \frac{1}{dt}\left(\frac{\partial M_{t+dt}}{\partial \lambda} - \frac{dM_t}{d\lambda}\right) + \frac{\partial F_{t+dt}}{\partial \lambda} + \frac{\partial Q_{t+dt}}{\partial \lambda} \qquad \text{[EQ 32.5]}$$

[EQ 32.4] defines the gradient method used in ECLIPSE. Note that the Jacobian

$J = \partial R / \partial X_{t+dt}$ is the same as used to solve the reservoir equations, so that no additional matrix setup is required.

Hence each gradient parameter λ requires the solution of a set of linear equations [EQ 32.4] at each simulator time step. When there are several gradient parameters, each requires the solution of the same set of linear equations [EQ 32.4] but with different right hand sides $\partial R/\partial\lambda$ in [EQ 32.5].

ECLIPSE 100 only

An efficient linear solver accounting for the 'multiple right hand sides' is used in ECLIPSE.

As an example of the technique, consider obtaining gradients with respect to a pore volume multiplier (parameter `HMPORVM`). Then the residual [EQ 32.2] can be written

$$R(X_{t+dt},\lambda)=\frac{\lambda M_{t+dt}(X_{t+dt})-M_t(\lambda)}{dt}+F_{t+dt}(X_{t+dt})+Q_{t+dt}(\lambda) \qquad \text{[EQ 32.6]}$$

and so

$$\frac{\partial R}{\partial\lambda}=\frac{1}{dt}\left(M_{t+dt}-\frac{dM_t}{d\lambda}\right) \qquad \text{[EQ 32.7]}$$

and the Jacobian is given by

$$A=\frac{1}{dt}\frac{\partial M_{t+dt}}{\partial X}+\frac{\partial F_{t+dt}}{\partial X}+\frac{\partial Q_{t+dt}}{\partial X}$$

Regression in SimOpt using ECLIPSE gradients

The solution $\partial X/\partial\lambda$ of [EQ 32.5] consists of derivatives of solution variables in each grid block. These are converted to derivatives of meaningful observable well quantities $\partial W_{calc}/\partial\lambda$ such as well block pressure, well flow rates, etc. SimOpt uses these quantities to calculate derivatives and the Hessian of the objective function with respect to the gradient parameters for use in the regression algorithm.

It is important to note that ECLIPSE returns gradients $\partial X/\partial\lambda$ around $\lambda = 1$ for multiplier-based parameters (such as `HMPORVM`, `HMPERMX`, `HMKRW` etc.). During a gradient-based minimization loop, a sequence of multiplicative modifiers λ_i, $i = i, \ldots, \upsilon$ may be applied to the base dataset whose effect is cumulative, so that after υ iterations, the cumulative modifier is

$$\Lambda_\upsilon=\prod_{i=1}^{\upsilon}\lambda_i$$

If the regression requires gradients with respect to the total modifier Λ_ν, then

$$\frac{\partial X}{\partial\Lambda_\upsilon}=\frac{\partial X}{\partial(\Lambda_{\upsilon-1}\lambda_\upsilon)}=\frac{1}{\Lambda_{\nu-1}}\cdot\frac{\partial X}{\partial\lambda_\nu}$$

In SimOpt, the regression operates on the parameter Λ_υ and the sensitivities provided by ECLIPSE are divided by Λ_υ at each regression iteration. For regression loops such as SimOpt relying on the cumulative parameters Λ_υ in this way, it is suggested that the log transform $\tilde{\lambda} = \ln(\lambda)$ is applied.

ECLIPSE 100 only

For parameters whose cumulative modifiers are additive (such as HMSWCR, HMSOWCR etc.), the cumulative modifier would be

$$\Lambda_\upsilon = \sum_{i=1}^{\nu} \lambda_i$$

in which case no modification is required because

$$\frac{\partial X}{\partial \Lambda_\upsilon} = \frac{\partial X}{\partial(\Lambda_{\nu-1} + \lambda_\upsilon)} = \frac{\partial X}{\partial \lambda_\upsilon}$$

For any objective function $J = J(X, \lambda)$, used during regression, then

$$\frac{dJ}{d\lambda} = \sum_t \frac{\partial J(X_t, \lambda)}{\partial X_t} \cdot \frac{\partial X_t}{\partial \lambda} + \frac{\partial J}{\partial \lambda}$$

so once $\partial X_t / \partial \lambda$ is known then regression can be performed for any objective function J.

Explicit gradients (ECLIPSE 100 only)

Where the discretization is not fully implicit, ECLIPSE accounts for the derivatives of these explicit terms in computing the gradients. At present, this is the case for transmissibility multipliers induced by rock compaction (ROCKTAB, ROCKTABH) and also for the VE pseudo capillary pressure terms when the vertical equilibrium option is in use.

In these cases,

$$M_{t+dt} = M_{t+dt}(X_{t+dt}, \lambda)$$

and

$$F_{t+dt} = F_{t+dt}(X_{t+dt}, \lambda)$$

and for the well flow terms

$$Q_{t+dt} = Q_{t+dt}(X_{t+dt}, \lambda)$$

where the λ dependence assumes the dependence on the explicit terms evaluated from X_t.

Then an extra term is added to the residual $\partial R / \partial \lambda$ to account for the explicit dependence on λ,

$$\frac{1}{dt} \cdot \frac{\partial M_{t+dt}}{\partial \lambda} + \frac{\partial F_{t+dt}}{\partial \lambda} + \frac{\partial Q_{t+dt}}{\partial \lambda}$$

which just involves calculating the implicit terms for the gradient computation corresponding to the explicit treatments in ECLIPSE.

Instantaneous gradients

The instantaneous gradient facility provides an alternative set of derivatives to those generated for history matching purposes. Recall the history matching philosophy in which derivatives of various solution quantities at the current time step are computed with respect to variations in a set of parameters at time zero. In contrast, the instantaneous gradient option computes derivatives of solution quantities at the current time step with respect to variations in the parameters at the current time step. The instantaneous gradient option harness the existing ECLIPSE gradient code with the exception that the gradient data need not be propagated forward in time. As a result the instantaneous gradient option can be switched on and off during the course of a simulation via the GUPFREQ keyword.

The aim of the instantaneous gradient feature is to provide information on the sensitivity of a well to its, and other wells', control modes. For example, if the well P1 is under bottom hole pressure control the gradient of the oil flow rate with respect to the bottom hole pressure can be requested. The calculation of such a gradient combines two factors, the response of the well to a variation in its own bottom hole pressure, and the response of the reservoir to the variation in the well's bottom hole pressure. Since the response of the reservoir is part of the gradient calculation the interaction between wells can also be studied. For example, the relationship between the water flow rate of well P1 and the water injection rate of well I1. However, it should be noted that a number of approximations are made within the gradient calculation that can reduce the accuracy of the calculation when considering the influence of one well upon another.

The instantaneous gradient options is activated via the GDIMS keyword in the RUNSPEC section. Since the gradient output is available as summary data an unencoded gradient license is required. In the summary section the gradient of the oil, water, gas flow rates and the bottom hole pressure with respect to a gradient parameter can be requested with the mnemonics WOFWCn, WWFWCn, WGFWCn and WBHWCn, where n is the parameter number required. The gradient parameters are defined by specifying wells using the GWRTWCV keyword in the SCHEDULE section. Wells selected as gradient parameters must be under either oil, water, gas rate or bottom hole pressure control since the response of the group hierarchy is not included in the gradient calculation. In addition the feature is restricted to the standard well model in unrefined grids.

Example usage

Specify two instantaneous gradient parameters in the RUNSPEC section.

```
GDIMS
 2 /
```

Request gradient output in the SUMMARY section. The first item requests that the gradient of the oil flow rate for wells P1 and P2 with respect to the instantaneous gradient parameter 1. The second item that the water flow rate for wells P1 and I1 with respect to the instantaneous gradient parameter 2.

```
WOFWC1
 P1 P2 /

WWFWC2
 P1 I1 /
```

Specify the instantaneous gradient parameters in the SCHEDULE section. Define the first parameter to be the control mode of well P1, the second to be the control mode of well I1. Compute the gradients at the next time step only. The values of the requested instantaneous gradient summary mnemonics will be frozen at the last computed value.

```
GWRTWCV
 P1 / -- gradient parameter 1
 I1 / -- gradient parameter 2
/

GUPFREQ
 NEXT /
```

At a later point in the simulation re-specify the instantaneous gradient parameters and compute the gradients for all subsequent time steps.

```
GWRTWCV
 P2 / -- gradient parameter 1
 I2 / -- gradient parameter 2
/

GUPFREQ
 ALL /
```

Summary of keywords

RUNSPEC section

The keywords are:

Table 32.1 RUNSPEC section

Simulator	Keyword	Description
ECLIPSE 300	FMTHMD	Declares that the HMD file is to be formatted.
	GDIMS	Specifies dimensions for the instantaneous gradient option.
ECLIPSE 300	HMDIMS	Specifies dimensions for the Gradient option.
	RPTHMD	Specifies controls on the output to the HMD file.
ECLIPSE 300	UNCODHMD	Requests that the HMD file contains unencoded derivatives.

Parameter specification keywords

GRID section

The keywords are:

Table 32.2 GRID section

Keyword	Description
HMFAULTS	Specifies gradients for faults.
HMMULRGT	Specifies inter-region transmissibility parameters.
HMAQUNUM	Specifies gradients for numerical aquifer parameters.

PROPS section

The keywords are:

Table 32.3 PROPS section

Keyword	Description
HMROCK	Specifies rock compressibility parameters.
HMROCKT	Specifies rock compaction parameters.

REGIONS section

The keywords are:

Table 32.4 REGIONS section

Keyword	Description
HMxxxxxx	Specifies gradient parameters.

Note The following options apply to the HMxxxxxx keyword.

The options are:

Table 32.5 HMxxxxxx keyword

Simulator	Option	Description
	HMTRANX	Specifies X-direction transmissibility parameters.
	HMTRANY	Specifies Y-direction transmissibility parameters.
	HMTRANZ	Specifies Z-direction transmissibility parameters.
	HMTRNXY	Specifies X- and Y-direction transmissibility parameters.
	HMTRANR	Specifies R-direction transmissibility parameters.
	HMTRANT	Specifies theta-direction transmissibility parameters.
ECLIPSE 300	HMPORVM	Specifies pore volume parameters.
	HMSIGMA	Specifies dual porosity sigma factor parameters.
	HMPERMX	Specifies X-direction permeability parameters.
	HMPERMY	Specifies Y-direction permeability parameters.
	HMPRMXY	Specifies XY-direction permeability parameters.
	HMPERMZ	Specifies Z-direction permeability parameters.
	HMTRNX-	Specifies negative X direction transmissibility parameters.
	HMTRNY-	Specifies negative Y-direction transmissibility parameters.
	HMTRNZ-	Specifies negative Z-direction transmissibility parameters.
	HMSWCR	Specifies critical water saturation parameters.
	HMSOWCR	Specifies critical oil-in-water saturation parameters.
	HMSOGCR	Specifies critical oil-in-gas saturation parameters.
	HMSWL	Specifies connate water saturation parameters.
	HMKRW	Specifies maximum water relative permeability parameters.
	HMKRG	Specifies maximum gas relative permeability parameters.
	HMKRO	Specifies maximum oil relative permeability parameters.
	HMKRWR	Specifies maximum displaced water relative permeability parameters.
	HMKRGR	Specifies maximum displaced gas relative permeability parameters.
	HMKRORG	Specifies maximum displaced oil-in-gas relative permeability parameters.

Table 32.5 HMxxxxxx keyword

Simulator	Option	Description
	HMKRORW	Specifies maximum oil-in-water displaced relative permeability parameters.
	HMSWLPC	Specifies lower water saturation for capillary pressure parameter.
	HMSGLPC	Specifies lower gas saturation for capillary pressure parameter.
	HMPCW	Specifies maximum water capillary pressure parameter.
	HMPCG	Specifies maximum gas capillary pressure parameter.

SOLUTION section

The keywords are:

Table 32.6 SOLUTION section

Keyword	Description
HMAQUCT	Specifies gradients for Carter-Tracy aquifers.
HMAQUFET	Specifies gradients for Fetkovich aquifers.
HMWELCON	Specifies gradients for well connections

SCHEDULE section

The keywords are:

Table 32.7 SCHEDULE section

Keyword	Description
GUPFREQ	Specifies update frequency for the instantaneous gradient option.
GWRTWCV	Specifies parameters for the instantaneous gradient option.
HMWPIMLT	Specifies gradients with respect to well productivity index.

Parameter modifier keywords

GRID section

The keywords are:

Table 32.8 GRID section

Simulator	Keyword	Description
	HMMULTFT	Specifies modifiers for fault transmissibilities.
	HMMULTSG	Specifies multipliers on dual porosity sigma factor.
ECLIPSE 300	HMMULTPV	Specifies pore volume modifiers.
	HMMULTX	Specifies X-direction transmissibility modifiers.
	HMMULTY	Specifies Y-direction transmissibility modifiers.
	HMMLTXY	Specifies XY-direction transmissibility modifiers.
	HMMULTZ	Specifies Z-direction transmissibility modifiers.
	HMMULTX-	Specifies negative X-direction transmissibility modifiers.
	HMMULTY-	Specifies negative Z-direction transmissibility modifiers.
	HMMMREGT	Specifies inter-region transmissibility modifiers.
	HMMULTR	Specifies R-direction transmissibility modifiers.
	HMMULTTH	Specifies theta-direction transmissibility modifiers.
	HMMLTPX	Specifies X-direction permeability modifiers.
	HMMLTPY	Specifies Y-direction permeability modifiers.
	HMMLTPZ	Specifies Z-direction permeability modifiers.
	HMMLTPXY	Specifies XY-direction permeability modifiers.
	HMMLAQUN	Specifies modifiers for numerical aquifers.

EDIT section

The keywords are:

Table 32.9 EDIT section

Simulator	Keyword	Description
	HMMULTFT	Specifies modifiers for fault transmissibilities.
ECLIPSE 300	HMMULTPV	Specifies pore volume modifiers.
	HMMULTX	Specifies X-direction transmissibility modifiers.
	HMMULTY	Specifies Y-direction transmissibility modifiers.
	HMMULTZ	Specifies Z-direction transmissibility modifiers.
	HMMULTX-	Specifies negative X-direction transmissibility modifiers.
	HMMULTY-	Specifies negative Y-direction transmissibility modifiers.

Table 32.9 EDIT section

Simulator	Keyword	Description
	HMMULTZ-	Specifies negative Z-direction transmissibility modifiers.
	HMMULTR	Specifies R-direction transmissibility modifiers.
	HMMULTTH	Specifies theta-direction transmissibility modifiers.

PROPS section

The keywords are:

Table 32.10 PROPS section

Keyword	Description
HMPROPS	Section heading for property modifiers.

Note The following options apply to the HMPROPS keyword.

The keywords are:

Table 32.11 HMPROPS options

Option	Description
HASWCR	Specifies critical water saturation modifiers.
HASOWCR	Specifies critical oil-in-water saturation modifiers.
HASOGCR	Specifies critical oil-in-gas saturation modifiers.
HASWL	Specifies connate water saturation modifiers.
HMKRW	Specifies maximum water relative permeability modifiers.
HMKRG	Specifies maximum gas relative permeability modifier.
HMKRO	Specifies maximum oil relative permeability modifier.
HMKRWR	Specifies maximum displacing water relative permeability modifier.
HMKRGR	Specifies maximum displacing gas relative permeability modifier.
HMKRORG	Specifies maximum displacing oil-in-gas relative permeability modifier.
HMKRORW	Specifies maximum displacing oil-in-water relative permeability modifier.
HASWLPC	Specifies modifier on lower water saturation used for water capillary pressure.
HASGLPC	Specifies modifier on lower gas saturation used for gas capillary pressure.
HMPCW	Specifies modifier on water capillary pressure.
HMPCG	Specifies modifier on gas capillary pressure.

Note For a description of the above keywords, refer to the HMPROPS keyword.

SOLUTION section

The keywords are:

Table 32.12 SOLUTION section

Keyword	Description
HMMLCTAQ	Specifies modifiers for Carter-Tracy aquifers.
HMMLFTAQ	Specifies modifiers for Fetkovich aquifers.
HMMLTWCN	Specifies modifiers for well connections

SUMMARY keywords

The keywords are:

Table 32.13 SUMMARY section

Keyword	Description
TCPUTSHT	CPU time per time step for all the gradient calculations.
TCPUHT	Total CPU time for all the gradient calculations.
TCPUTSH	CPU time per time step for each gradient calculations.
TCPUH	Total CPU time for each gradient calculations.
HLINEARS	Number of linear iterations for each gradient calculation.
HSUMLINS	Cumulative number of linear iterations for each gradient parameter.
QAAQR	Derivatives of analytic aquifer influx rate.
QAAQT	Derivatives of analytic aquifer cumulative influx.
QAAQP	Derivatives of Fetkovich aquifer pressure.
QANQR	Derivatives of numerical aquifer influx rate.
QANQT	Derivatives of numerical aquifer cumulative influx.
QANQP	Derivative of numerical aquifer pressure.
W**WCn	Output from instantaneous gradient option

Miscellaneous keywords

The keywords are:

Table 32.14 Miscellaneous keywords

Simulator	Keyword	Description
	GRADRESV	Specifies derivatives of solution quantities to be output to the RESTART file.
	GRADRFT	Specifies derivatives of RFT quantities to be output to the RFT file.
ECLIPSE 300	GRADWELL	Specifies well quantities to be output to the HMD file.
	GRADGRUP	Specifies group quantities to be output to the HMD file.
	HMPROPS	Flags start of HMPROPS section.
	HMRREF	Specifies reference pressures for rock compaction modifiers.
	NOHMD	Switch off gradient calculations.
	NOHMO	Switch off gradient calculations.
	RPTHMG	Configure output of derivatives for groups.
	RPTHMW	Configure output of derivatives for wells.
	TUNINGH	Tuning parameters for gradient calculations.

Example problem (ECLIPSE 100 only)

RUNSPEC

```
TITLE
  HISTORY MATCHING TEST CASE
DIMENS
   10   10    3  /
NONNC
FAULTDIM
   20 /
OIL
WATER
GAS
DISGAS
FIELD
EQLDIMS
    1  100   10    1    1 /
TABDIMS
    2    2   16   12    3   12    1    1 /
REGDIMS
    3    3    0   /
AQUDIMS
    0    0    2    0    2  120  /
HMDIMS
 4  4  10  2  6 /
WELLDIMS
    4    1    1    4 /
START
  19 'OCT' 1982  /
NSTACK
   24 /
FMTOUT
FMTIN
FMTHMD
```

GRID

In this section, the geometry of the Simulation Grid and the Rock Permeabilities and Porosities are defined.

The X and Y direction cell sizes (DX, DY) and the porosities are constant throughout the grid. These are set in the first 3 lines after the EQUALS keyword. The cell thicknesses (DZ) and permeabilities are then set for each layer. The cell top depths (TOPS) are needed only in the top layer (although they could be set throughout the grid). The specified MULTZ values act as multipliers on the transmissibilities between the current layer and the layer below.

```
GRID ------------------------------------------------------------
--------
-- ARRAY  VALUE     ------- BOX ------
EQUALS
      'DX'    1000      /
      'DY'    1000      /
      'PORO'  0.3       /
      'DZ'    20        1  10  1  10  1  1  /
      'PERMX' 500       /
      'TOPS'  8325      /
      'DZ'    30        1  10  1  10  2  2  /
      'PERMX' 50        /
      'TOPS'  8345      /
      'DZ'    50        1  10  1  10  3  3  /
      'PERMX' 200       /
      'TOPS'  8395      /
/     EQUALS IS TERMINATED BY A NULL RECORD
-- THE Y AND Z DIRECTION PERMEABILITIES ARE COPIED FROM PERMX
--    SOURCE  DESTINATION  ------- BOX ------
COPY
      'PERMX'    'PERMY'   1  10  1  10  1  3  /
      'PERMX'    'PERMZ'   /
/
FAULTS
 'MAJOR'  3 3   1 4    1 3  'X' /
 'MAJOR'  3 5   4 4    1 3  'Y' /
 'MAJOR'  5 5   5 10   1 3  'X' /
 'MINOR'  6 6   1 3    1 3  'X' /
 'MINOR'  6 10  3 3    1 3  'Y' /
/
MULTFLT
 'MAJOR'  0.001 /
 'MINOR'  01 /
/
```

```
HMFAULTS
 'MAJOR' /
 'MINOR' /
/
OLDTRAN
EQUALS
 'HMMULTX' 1.05 2  5 1 10 1 2 /
 'HMMULTY' 1.05 1 10 2  5 1 2 /
/
--
 OUTPUT OF DX, DY, DZ, PERMX, PERMY, PERMZ, MULTZ, PORO AND TOPS DAT
A
-- IS REQUESTED, AND OF THE CALCULATED PORE VOLUMES AND X, Y AND Z
-- TRANSMISSIBILITIES
RPTGRID
  'DX'  'DY'  'DZ'  'PERMX'  'PERMY'  'PERMZ'  'MULTZ'
  'PORO'  'TOPS'  'PORV'  'TRANX'  'TRANY'  'TRANZ'  /
```

PROPS

The PROPS section defines the relative permeabilities, capillary pressures, and the PVT properties of the reservoir fluids.

Water relative permeability and capillary pressure are tabulated as a function of water saturation.

```
PROPS-------------------------------------------------------------
--------------
     SWAT   KRW   PCOW
SWFN
    0.12  0       0
    1.0   1.0     0  /
/
```

Similarly for gas:

```
--  SGAS   KRG   PCOG
SGFN
    0     0       0
    0.02  0       0
    0.05  0.005   0
    0.12  0.025   0
    0.2   0.075   0
    0.25  0.125   0
    0.3   0.19    0
    0.4   0.41    0
    0.45  0.6     0
    0.5   0.72    0
    0.6   0.87    0
    0.7   0.94    0
    0.88  0.98    0
/
/
```

Oil relative permeability is tabulated against oil saturation for Oil-Water and Oil-Gas-Connate Water cases:

```
--  SOIL     KROW     KROG
SOF3
    0        0        0
    0.18     0        0
    0.28     0.0001   0.0001
    0.38     0.001    0.001
    0.43     0.01     0.01
    0.48     0.021    0.021
    0.58     0.09     0.09
    0.63     0.2      0.2
    0.68     0.35     0.35
    0.76     0.7      0.7
    0.83     0.98     0.98
    0.86     0.997    0.997
    0.879    1        1
    0.88     1        1    /
/
```

PVT properties of water

```
--
  REF. PRES. REF. FVF  COMPRESSIBILITY  REF VISCOSITY  VISCOSIBILITY
PVTW
       4014.7     1.029        3.13D-6           0.31            0 /
/
```

Rock compressibility

```
--    REF. PRES   COMPRESSIBILITY
ROCK
        14.7          3.0D-6          /
/
```

Surface densities of reservoir fluids

```
--        OIL   WATER   GAS
DENSITY
         49.1   64.79  0.06054  /
/
```

PVT properties of Dry Gas (no vaporized oil) we would use PVTG to specify the properties of Wet Gas

```
--   PGAS   BGAS   VISGAS
PVDG
     14.7 166.666   0.008
    264.7  12.093   0.0096
    514.7   6.274   0.0112
   1014.7   3.197   0.014
   2014.7   1.614   0.0189
   2514.7   1.294   0.0208
   3014.7   1.080   0.0228
   4014.7   0.811   0.0268
   5014.7   0.649   0.0309
   9014.7   0.386   0.047   /
/
```

PVT properties of Live Oil (with Dissolved Gas) we would use PVDO to specify the properties of Dead Oil.

For each value of R_s the saturation pressure, FVF and viscosity are specified. For R_s=1.27 and 1.618, the FVF and viscosity of Undersaturated Oil are defined as a R_s, but must be supplied for the highest R_s (1.618).

```
--   RS      POIL  FVFO  VISO
PVTO
    0.001    14.7 1.062  1.04    /
    0.0905  264.7 1.15   0.975   /
    0.18    514.7 1.207  0.91    /
    0.371  1014.7 1.295  0.83    /
    0.636  2014.7 1.435  0.695   /
    0.775  2514.7 1.5    0.641   /
    0.93   3014.7 1.565  0.594   /
    1.270  4014.7 1.695  0.51
           5014.7 1.671  0.549
           9014.7 1.579  0.74    /
    1.618  5014.7 1.827  0.449
           9014.7 1.726  0.605   /
    1.630  8014.7 1.828  0.449
           9014.7 1.8    0.605   /
/
/
```

Carter Tracy aquifer influence table

```
AQUTAB
  0.01  0.112
  0.05  0.229
  0.1   0.315
  0.15  0.376
  0.2   0.424
  0.25  0.469
  0.3   0.503
  0.4   0.564
  0.5   0.616
  0.6   0.659
/
```

Output controls for PROPS data activated for SOF3, SWFN, SGFN, PVTW, PVDG, DENSITY and ROCK keywords

```
RPTPROPS
  'SOF2' '=' 1  'SWFN' '=' 1  'SGFN' '=' 1  'PVTW' '=' 1
  'PVTG' '=' 1  'DENSITY'  'ROCK'  /
```

REGIONS

The REGIONS section defines how the reservoir is split into regions by saturation function, PVT function, fluid in place region, gradient region, etc.

```
REGIONS ------------------------------------------------------------
--------
EQUALS
 'PVTNUM'  2   1 10 1 10 1 3 /
 'SATNUM'  2   1 10 1 10 1 3 /
 'HMTRANZ' 1   1 10 1 10 1 3 /
 'HMTRANX' 1   1 10 1 10 1 3 /
 'HMTRNXY' 1   1 10 1 10 1 3 /
 'HMTRANY' 1   1 10 1 10 1 1 /
 'HMTRANY' 2   1 10 1 10 1 2 /
 'HMTRANY' 3   1 10 1 10 1 3 /
/
BOX
 1 10   1 10   2 2 /
```

```
SATNUM
 1 2 1 2 1   2 1 2 1 2
 2 1 2 1 2   1 2 1 2 1
 1 2 1 2 1   2 1 2 1 2
 2 1 2 1 2   1 2 1 2 1
 1 2 1 2 1   2 1 2 1 2
 2 1 2 1 2   1 2 1 2 1
 1 2 1 2 1   2 1 2 1 2
 2 1 2 1 2   1 2 1 2 1
 1 2 1 2 1   2 1 2 1 2
 2 1 2 1 2   1 2 1 2 1
/
ENDBOX
COPY
 'SATNUM' 'FIPXXX' /
/
EQUALS
 'FIPNUM'  1   4*          1 1  /
 'FIPNUM'  2   4*          2 2  /
 'FIPNUM'  3   4*          3 3  /
 'FIPYYY'  1   1 3  1 10   1 3  /
 'FIPYYY'  2   4 7   4*         /
 'FIPYYY'  3   7 10  4*         /
/
RPTREGS
  'PVTNUM'  'SATNUM'  'EQLNUM'  'FIPNUM'  'HM'  /
```

SOLUTION

The SOLUTION section defines the initial state of the solution variables (phase pressures, saturations and Gas-Oil ratios)

```
SOLUTION ------------------------------------------------------------
-----------
-- DATA FOR INITIALISING FLUIDS TO POTENTIAL EQUILIBRIUM
--
--    DATUM  DATUM   OWC    OWC    GOC    GOC    RSVD   RVVD   SOLN
--    DEPTH  PRESS  DEPTH   PCOW  DEPTH   PCOG  TABLE  TABLE   METH
EQUIL
      8400   4800   8500    0     8200    0      1      0       0  /
-- VARIATION OF INITIAL RS WITH DEPTH
--
--    DEPTH    RS
RSVD
       8200  1.270
       8500  1.270  /
```

```
--CARTER TRACY AQUIFER CONNECTING TO BOTTOM FACE OF RESERVOIR
AQUCT
   1  5100.0  1*  20.0   0.10  3.0E-5  2400.0  140.0  360  1  1  /
/
AQUANCON
 1   1  10   1  10   3   3  'K+'  /
/
HMAQUCT
--     PERM   THETA
  1   'YES'   'YES'  /
/
-- OUTPUT CONTROLS
-- (SWITCH ON OUTPUT OF INITIAL GRID BLOCK PRESSURES
--                      FLUID VISCOSITIES
--                      PHASE RESERVOIR DENSITIES  )
RPTSOL
  'PRES'  'FIP=3'  'VOIL'  'VWAT'  'VGAS'  'DENO'
  'DENW'  'DENG'  /
```

SUMMARY

This section specifies data to be written to the SUMMARY files and, which may later be used with the ECLIPSE graphics package

```
-- FIELD OIL PRODUCTION
FOPR
-- WELL GAS-OIL RATIO FOR PRODUCER
WGOR
'PRODUCER'
/
-- WELL BOTTOM-HOLE PRESSURE
WBHP
'PRODUCER'
/
--AQUIFER INFLUX
FAQT
AAQR
/
AAQR
/
-- CPU TIMES
TCPU
TCPUH
/
```

SCHEDULE

The SCHEDULE section defines the operations to be simulated

```
SCHEDULE ------------------------------------------------------------
-----------
RPTSCHED
  'PRES'  'SOIL'  'SWAT'  'SGAS'  'RS'  'FIP=3'
  'WELLS' 'CPU=2'  'WELSPECS'  'NEWTON=2'
  'VOIL'  'VWAT'  'VGAS'  'DENO'  'DENW'  'DENG'  /
RPTRST
  'BASIC=3'  'FIP'  'FREQ=2'  'PRES'  'VISC'
  'DEN'  /
DRSDT
  100.0
/
GRADWELL
 'WBHP' /
 'WBP'  /
 'WWCT' /
 'WOPR' /
 'WWPR' /
/
RPTHMW
 'PROD*' 'ON'  /
 'INJE*' 'OFF' /
/
```

Set initial time step to 1 day and maximum to 6 months

```
TUNING
/
/
 8*  200.0 /
```

Well specification data

```
--   WELL   GROUP LOCATION BHP   PI
--   NAME      NAME   I  J    DEPTH DEFN
WELSPECS
    'PRODUCER' 'G'   10 10    8400 'OIL'  /
    'PRODUCE2' 'G'   10 1     8400 'OIL'  /
    'INJECTOR' 'G'   1  1     8335 'GAS'  /
    'INJECT2' 'G'    1  10    8335 'GAS'  /
/
```

Completion specification data

```
--     WELL     -LOCATION- OPEN/ SAT CONN  WELL
--     NAME     I  J K1 K2 SHUT  TAB FACT  DIAM
```

```
COMPDAT
    'PRODUCER'  10 10 3  3 'OPEN' 0   -1   0.5  /
    'PRODUCE2'  10 1  3  3 'OPEN' 0   -1   0.5  /
    'INJECTOR'   1  1 1  1 'OPEN' 1   -1   0.5  /
    'INJECT2'   1  10 1  1 'OPEN' 1   -1   0.5  /
  /
WCONHIST
     'PRODUCER' 'OPEN' 'ORAT' 20000 /
     'PRODUCE2' 'OPEN' 'ORAT' 20000 /
  /
WELTARG
     'PRODUCER' 'BHP' 1000 /
     'PRODUCE2' 'BHP' 1000 /
  /
```

Injection well controls

```
--      WELL     INJ   OPEN/  CNTL    FLOW
--      NAME    TYPE   SHUT   MODE    RATE      BHP
WCONINJ
'INJECTOR' 'GAS' 'OPEN' 'RATE'  100000 3* 10000  /
     'INJECT2' 'GAS' 'OPEN' 'RATE'  100000 3* 10000  /
  /
TSTEP
 6*40
 /
END
```

ECLIPSE 300

Note The above example shows the majority of parameters available for use within ECLIPSE 100. At present the only parameters available for use within ECLIPSE 300 are pore volume, transmissibility and permeability multipliers.

Group Production Rules

Chapter 33

Introduction

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL

This chapter gives details of the implementation of the PRORDER keyword actions, which are performed by ECLIPSE 100 whenever a group fails to meet its target production rate set in keyword GCONPROD. This feature is currently not available in ECLIPSE 300.

The available actions are:

1 Drill new wells from the drilling queue

2 Open new connections on AUTO in existing wells

3 Reduce each well's THP limit to a specified lower value

4 Change VFP table numbers (for example to model a retubing operation)

5 Change well ALQ values (that is apply a fixed amount of artificial lift)

6 Switch on, or increase power of, compressors linked to the group.
(This option is only available if the Extended Network model is in use - see "The extended network model" on page 693).

The order in which the actions are performed is dictated by the order in which their respective mnemonics are entered in record 1 of the PRORDER keyword. Whenever a group fails to meet its target production rate set in keyword GCONPROD, ECLIPSE first looks for a well on which to perform the first action. If no suitable well can be found, or if rig availability constraints prevent this action from being performed at that time, ECLIPSE looks for a well on which to perform the second action in the list. If it cannot perform the second action on a well, it looks for a well on which to perform the third action, and so on. The implementation of each of the five actions is described in detail below.

Drilling new wells

A drilling queue must be set up either with keyword QDRILL (for a sequential drilling queue) or keyword WDRILPRI (for a prioritized drilling queue), containing production wells declared as SHUT in keyword WCONPROD. In a sequential drilling queue, wells are considered for opening in the sequence in which they were placed in the queue. In a prioritized drilling queue, wells are considered for opening in decreasing order of their drilling priority. Well drilling priorities may be set manually in keyword WDRILPRI or calculated from their current potentials according to a function defined with keyword DRILPRI. Restrictions on the rate at which wells can be drilled may be imposed by declaring the time taken to drill each well (keyword WDRILTIM item 2) and defining drilling rigs (keyword GRUPRIG). When a group cannot achieve its target rate of the required phase, ECLIPSE scans the drilling queue for a suitable well to open. A well is rejected if:

1. it is not subordinate to the group that is unable to achieve its production target
2. it is subordinate to an intermediate level group that is already operating at its target or limiting rate for the required phase, and so cannot increase its flow further
3. it is subordinate to a prioritization group (keyword GCONPRI). The drilling queue logic is not activated under prioritization group control
4. the well takes a non-zero time to drill (keyword WDRILTIM item 2) and all drilling rigs available to its group (keyword GRUPRIG) are already occupied for the whole of the current time step
5. its group, or any superior group in the hierarchy, already has its maximum allowed number of open wells (keyword GECON item 9)
6. the oil saturation in all its completed grid blocks is below a specified minimum value (keyword WELSOMIN)
7. the well has a connection within a grid block that also contains a connection belonging to another open well, and keyword WDRILRES has been entered to prevent more than one well being completed in the same block.

The first producer in the drilling queue that survives all the above tests is drilled.

Opening new connections (re-perforating)

Well connections may be placed on “automatic opening”, by specifying AUTO in item 6 of keyword COMPDAT (or in keyword WELOPEN or COMPDATL as appropriate). Wells will be worked over in turn to open these connections. Restrictions on the rate at which wells can be worked over may be imposed by declaring the time taken to perform a workover (keyword WORKLIM) and defining workover rigs (keyword GRUPRIG). When a group cannot achieve its target rate of the required phase, ECLIPSE examines each open production well in turn, rejecting the well if:

1. it has no connections in its AUTO opening queue
2. it is already operating at its own target or limiting rate for the required phase, and so cannot increase its flow further
3. it is not subordinate to the group that is unable to achieve its production target
4. it is subordinate to an intermediate level group that is already operating at its target or limiting rate for the required phase, and so cannot increase its flow further
5. workovers take a non-zero time (keyword WORKLIM) and all workover rigs available to its group (keyword GRUPRIG) are already occupied for the whole of the current time step.

For each well that survives all these tests, ECLIPSE determines $\sum T_{wj} M_{pj}$, the sum of the connection factors multiplied by the mobility of the required phase over all the connections in its AUTO queue. ECLIPSE then selects the well having the largest value of this sum, and opens its next connection in its AUTO queue. Within each well, connections on AUTO are opened in the order in which they are first declared with the keyword COMPDAT.

This scheme allows you to dictate the order in which the new connections are opened in any given well (for example from the bottom up). However, ECLIPSE decides for itself which well to re-perforate, on the basis of the combined productivity of the connections remaining on AUTO. Note that the procedure does not necessarily work over the ‘best’ connection each time. Consider a well whose first connection on AUTO is in a layer with poor mobility but which has additional connections on AUTO in highly productive layers. The procedure may choose this well on the strength of its productive AUTO connections, but opens the unproductive connection first. Thus the unproductive connection does not ‘block’ the opening of the productive ones further down the queue. In fact, opening the unproductive connection may not be sufficient to achieve the group’s rate target, and another connection may have to be opened also. The time taken to perform a workover is assumed to be independent of the number of connections opened. Thus, if a rig has been allocated to the well to open a connection, opening a second connection in the well during the same time step will not take up any additional rig time.

There are two additional options that relate to the opening of connections on AUTO. Firstly, connections can be lumped together into ‘completions’ (see keyword COMPLUMP). During automatic workovers, all connections belonging to the same completion are opened or closed together. When a well has been selected for re-perforating, its completions are examined in increasing order of their completion number, and the first one found having connections on AUTO will have all its connections opened together. Secondly, a minimum oil saturation can be specified with keyword WELSOMIN. Any connection or lumped completion on AUTO with a grid block oil saturation below this minimum value is not opened. In lumped completions, the connection factor weighted average of the grid block saturations is used.

Note that a connection on AUTO is also opened automatically when another connection in the same well is closed in a workover at the end of the time step, which could be triggered for example by the violation of an economic limit (keywords WECON, GECON). Also, if the production rate of a well falls below its minimum economic limit (keyword WECON items 2 and 3), its next connection on AUTO is opened. If it has no connections remaining on AUTO, the well is shut or stopped. These actions result in AUTO connections being opened according to the behavior of the wells individually rather than as a group, and takes place irrespective of whether or not group targets are being met.

Reducing the THP limit

Each production well can be given a secondary THP limit in item 8 of keyword WLIFT (which should be lower than the limit currently specified in WCONPROD or WELTARG). A decrement value may also be set in item 11 of WLIFT, in which case ECLIPSE reduces the THP to its secondary limit in stages, a decrement at a time. When a group cannot achieve its target rate of the required phase, ECLIPSE examines each open production well in turn, rejecting the well if:

1. it has no secondary THP limit, or the THP has already been reduced to its secondary limit
2. it is currently operating under a control mode other than THP control
3. it is not subordinate to the group that is unable to achieve its production target
4. it is subordinate to an intermediate level group that is already operating at its target or limiting rate for the required phase, and so cannot increase its flow further.

For each well that survives all these tests, ECLIPSE obtains an estimate of its possible increase in production rate resulting from a reduction in THP by its next decrement (or to its secondary limit if no decrements are specified), using the current values of the phase mobilities at its connections. The well with the highest estimated increase multiplied by its efficiency factor will have its THP reduced by the next decrement. If no decrements are specified for the well, or if its next decrement would cross the secondary THP limit, the THP is reduced to its secondary limit.

Thus the wells have their THPs reduced in decreasing order of the resulting increase in production rate. The THP switching process is treated as instantaneous, and there is no limit on the rate at which wells can be switched.

Secondary THP limits should not be set for wells whose group is part of a network (see "The Network Option" on page 681), as the wells' THP limits are set by their group's nodal pressure in the network. THP switching will not be performed on any such wells.

Note that wells also switch to their secondary THP limits if their production rate falls below a specified value (keyword WLIFT item 2), or if their water cut or gas-liquid ratio exceeds a specified limit (keyword WLIFT items 7 and 9), or if the well dies under THP control. This triggers the switch according to the behavior of the individual wells, irrespective of whether or not group targets are being met. If you wish the secondary THP values to be applied only using the group production rules to help meet a group target, or if the well dies under THP control, then you must give the rate, water cut and GLR limits in items 2, 7 and 9 of keyword WLIFT a default, zero or negative value.

Changing VFP table numbers (retubing)

Each production well can be given a secondary VFP table number in item 4 of keyword WLIFT (which should be a table with lower tubing pressure loss). When a group cannot achieve its target rate of the required phase, ECLIPSE examines each open production well in turn, rejecting the well if:

1. it has no secondary VFP table number, or it has already switched to its secondary table
2. it is currently operating under a control mode other than THP control
3. it is not subordinate to the group that is unable to achieve its production target
4. it is subordinate to an intermediate level group that is already operating at its target or limiting rate for the required phase, and so cannot increase its flow further.

For each well that survives all these tests, ECLIPSE obtains an estimate of the possible increase in production rate resulting from the switch to its new VFP table, using the current values of the phase mobilities at its connections. The well with the highest estimated increase multiplied by its efficiency factor is switched to its secondary VFP table.

Thus the wells are switched to their secondary VFP tables in decreasing order of the resulting increase in production rate. The switching process is treated as instantaneous, and there is no limit on the rate at which wells can be switched.

Note that wells can also be made to switch to their secondary VFP tables if their production rate falls below a specified value (keyword WLIFT item 2), or if their water cut or gas-liquid ratio exceeds a specified limit (keyword WLIFT items 7 and 9), or if the well dies under THP control. This triggers the switch according to the behavior of the individual wells, irrespective of whether or not group targets are being met. If you wish the secondary VFP tables to be applied only through the group production rules to help meet a group target, or if the well dies under THP control, then you must give the rate, water cut and GLR limits in items 2, 7 and 9 of keyword WLIFT a default, zero or negative value.

Changing well ALQ values (applying lift)

Each production well can be given a secondary ALQ value in item 5 of keyword WLIFT, which represents the application of a fixed quantity of artificial lift (for example, a fixed rate of lift gas injection). An increment value may also be set in item 10 of WLIFT, in which case ECLIPSE increases the ALQ to its secondary value in stages, an increment at a time. A new well efficiency factor can also be defined in item 6 of keyword WLIFT if required, which comes into effect at the same time as the secondary ALQ value (or the first ALQ increment) is applied. There is an option to restrict the total amount of lift applied within a particular group, and the number of wells on lift within a group (keyword GLIFTLIM). When a group cannot achieve its target rate of the required phase, ECLIPSE examines each open production well in turn, rejecting the well if:

1. it has no secondary ALQ value set in keyword WLIFT, or it has already switched fully to this ALQ value
2. it is currently operating under a control mode other than THP control
3. it is not subordinate to the group that is unable to achieve its production target
4. it is subordinate to an intermediate level group that is already operating at its target or limiting rate for the required phase, and so cannot increase its flow further
5. its group, or any superior group, would exceed its limit on total lift or the number of wells on lift, set in keyword GLIFTLIM.

For each well that survives all these tests, ECLIPSE obtains an estimate of the possible increase in production rate resulting from an increase in ALQ by its next increment (or to its secondary value if no increments are specified), using the current values of the phase mobilities at its connections. The effects of a new efficiency factor, if specified, are also taken into account. The well with the highest estimated increase will have its ALQ value increased by the next increment. If no increments are specified for the well, or if its next increment would cross the secondary ALQ value, the ALQ will be increased to its secondary value.

Thus the wells are given lift in decreasing order of the resulting increase in production rate. The lift switching process is treated as instantaneous, and there is no limit on the rate at which wells can be switched.

If the Network option is in use, the estimate of production increase will not take account of the increased pressure losses in the network; if these are important, you should use the Gas Lift Optimization facility instead of WLIFT and GLIFTLIM (see "Gas Lift Optimization" on page 385). This offers a much more sophisticated means of optimising the distribution of lift gas to the individual wells. The facility automatically allocates lift gas to the wells until group production targets are met or other operating limits are reached. Gas lift allocations are calculated at specified intervals, or during the first NUPCOL Newton iterations of each time step, depending on the instructions in keyword LIFTOPT (regardless of the presence or absence of the LIFT mnemonic in PRORDER). By default, all the requested production rule actions will be attempted **before** gas lift is allocated. However, if LIFT is entered in the PRORDER keyword, gas lift allocation takes precedence over any production rule actions whose mnemonics are entered after LIFT. Thus, for example, it is possible to specify that drilling new wells should only be attempted if group targets cannot be met by applying gas lift. Essentially, all production rule actions entered after LIFT are only performed immediately after a gas lift optimization calculation, and not at any other time or Newton iteration, if a group production target still cannot be met. In this case it is best to configure LIFTOPT so that gas lift is optimized during each of the first NUPCOL iterations of every time step.

Note that wells can also be made to switch on lift if their production rate falls below a specified value (keyword WLIFT item 2), or if their water cut or gas-liquid ratio exceeds a specified limit (keyword WLIFT items 7 and 9), or if the well dies under THP control. This triggers the switch according to the behavior of the individual wells, irrespective of whether or not group targets are being met. If you wish lift to be applied only through the group production rules to help meet a group target, or if the well dies under THP control, then you must give the rate, water cut and GLR limits in items 2, 7 and 9 of keyword WLIFT a default, zero or negative value.

Activating compressors

This option is only available if the Extended Network model is in use (see "The extended network model" on page 693). The Extended Network model includes a special type of compressor that is turned on automatically when a nominated group fails to meet its production rate target. These compressors are defined with the keyword `NETCOMPA`. Each compressor is located in a specified branch of the network, and its operation is modeled by changing the ALQ value used to look up the VFP table associated with the branch. Compressors may be single-level or multi-level. Single-level compressors switch directly to the new ALQ value when they are turned on. Multi-level compressors change their ALQ value in a user-defined number of increments until they are fully on at their maximum ALQ value.

It is possible for a group to have more than one compressor responding to it. The compressors can be located anywhere in the network, but of course they should be positioned to boost the production of their nominated group. If two or more compressors respond to the same group, the order in which they are activated is controlled by their 'switching sequence number' (item 12 in keyword `NETCOMPA`). Compressors responding to the same group are activated in increasing order of their sequence number. If more compression is still required when the first compressor is fully on, the next compressor that responds to the group will be turned on (incrementally, if it is a multi-stage compressor).

When a group cannot achieve its target rate, ECLIPSE scans the network for a compressor that responds to the group. Compressors that are already fully on are ignored. If more than one compressor responds to the group, the compressor with the lowest switching sequence number is selected first. If it is a single-level compressor, it is turned fully on. If it is a multi-level compressor, its ALQ value is increased one increment at a time until either the group's rate target is met or the compressor is fully on.

The effect of closing wells during drilling and workovers

The rate at which new wells can be drilled and existing wells worked over can be controlled by specifying non-zero durations for these processes (see keywords WDRILTIM and WORKLIM) and defining drilling and workover rigs (see keyword GRUPRIG). A drilling or workover process will occupy the appropriate rig for the specified time. A well cannot be drilled (or worked over) in a given time step if all available drilling (or workover) rigs are occupied for the whole time step.

By default, a well starts flowing at the beginning of the time step in which its drilling operation starts, and it continues flowing while it is being worked over. However, there is an option to treat wells as shut for the duration of their drilling and workover processes (see keyword WDRILTIM item 3). This is achieved by giving the wells temporary reduced efficiency factors, which reflect the proportion of the time step in which they are unable to flow. But while this option gives more realistic behavior, it can cause a problem in the operation of the group production rules.

When a well is treated as shut for the duration of its drilling or workover process, the full benefit of this action is not realized until the time step after the process has been completed. In fact the immediate effect of a 'REPERF' action may be a temporary reduction in production rate, if the workover takes a non-zero time and the well is closed for its duration. Consequently, the immediate effect of a 'DRILL' or 'REPERF' action may not be sufficient to enable the group to reach its target rate, and ECLIPSE attempts more actions from the production rule list, even if these would not have been necessary once the drilling or workover process is complete.

If you wish to treat wells as shut while being drilled or worked over, it is advisable to set the item in record 2 of the PRORDER keyword to 'NO'. This prevents the performance of any further production rule actions for the same group following a 'DRILL' or 'REPERF' action until the benefits of this action are fully realized.

Using production rules with the network option

When the Network option is in use (see "The Network Option" on page 681), by default all actions except for compressor switching are performed in any network balancing iteration when a flow target is not met. Thus it is possible, for example, that a well may be opened from the drilling queue prematurely because an intermediate network balancing iteration has high nodal pressures and consequentially low well productivity. The exception is the compressor switching action, which is only performed if more flow is required once the network has converged.

It is possible to instruct ECLIPSE to prevent any of the Production Rule actions from being performed before the network balancing iterations have converged, by using the `WAITBAL` keyword. When this option is selected, all production rule actions are suppressed while the network is being balanced. Once the network balancing iterations have converged, if more flow is still required to meet a group flow target an appropriate Production Rule action is performed and the network has to be balanced again. Using this option therefore requires more network balancing iterations in total.

Hysteresis

Chapter 34

Introduction

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

This chapter describes the Hysteresis option.

This option enables you to specify different saturation functions for drainage (decreasing wetting phase saturation) and imbibition (increasing wetting phase saturation) processes.

You supply two saturation function table numbers for each cell, using the `SATNUM` and `IMBNUM` keywords in the `REGIONS` section. These provide respectively the primary drainage and pendular imbibition curves.

The primary drainage curve is for a process which starts at the maximum possible wetting phase saturation, S_{wmaxd}. (This value will depend upon the endpoints of the saturation tables specified using the `SATNUM` keyword.) If the wetting phase saturation decreases to S_{wmin}, this primary drainage curve is used.

In a similar way, if the initial saturation is S_{wmin}, and the wetting phase saturation increases to S_{wmaxi}, the imbibition table data will be used. (The maximum wetting phase saturation which can be reached, S_{wmaxi}, is determined from the endpoints of the tables specified using the `IMBNUM` keyword, and will generally be less than S_{wmaxd}.)

If the drainage or imbibition process is reversed at some point, the data used does not simply run back over its previous values.

The way in which this hysteresis process is treated in ECLIPSE follows the techniques described by Aziz and Settari [Ref. 18], Killough [Ref. 19], Carlson [Ref. 20], Land [Ref. 21] and J. Jargon.

Using the hysteresis option

The Hysteresis option in ECLIPSE is enabled by specifying the item 'HYSTER' in the keyword SATOPTS in the RUNSPEC section. The normal saturation table numbers in the REGIONS section are taken as defining the drainage curves. A second set of keywords, IMBNUM, IMBNUMX, IMBNUMX-,..., IMBNUMZ-, is used to specify the tables to be used for each cell to supply imbibition curves. If the imbibition and drainage table numbers for a cell are the same, no hysteresis will occur for that cell.

The mnemonics IMBNUM, IMBNUMX, IMBNUMX-, and so on., in the RPTREGS keyword will control the output of the entered imbibition table region number data.

In ECLIPSE it is possible to apply hysteresis to either the relative permeabilities, or capillary pressures or both.

The keyword EHYSTR in the PROPS section is specific to hysteresis. This sets the values of two parameters that determine the form of the scanning curves for capillary pressure hysteresis and wetting phase relative permeability hysteresis, and selects one of a choice of models for relative permeability hysteresis. It also allows you to control the application of hysteresis to either the relative permeability, capillary pressure, or both. The default is for hysteresis in both relative permeability and capillary pressure.

There is also a choice of options in the EHYSTR keyword to apply the Killough hysteresis model to the wetting phase, and also to model an oil-wet system.

The reservoir is always equilibrated using the drainage curves.

ECLIPSE 100 The Hysteresis option can not be used in combination with the Vertical Equilibrium option in the same run.

ECLIPSE 100 The Hysteresis option can not be used in combination with the Surfactant model.

If the Hysteresis option is used in a non-VE run with a COMPRP or COMPVE keyword present, the well connections must use the same saturation tables for both imbibition and drainage.

For wells, the drainage and imbibition table numbers may be specified using the connection keywords COMPDAT and COMPIMB, and also with the WELLCOMP keyword.

Capillary pressure hysteresis is always applied using the Killough method [Ref. 19].

ECLIPSE 100 Relative permeability hysteresis may be modeled by the Carlson [Ref. 20] or Killough [Ref. 19] methods.

ECLIPSE 300 Relative permeability hysteresis may be modeled by the Carlson [Ref. 20] or Killough [Ref. 19] or Jargon methods.

Relative permeability hysteresis in the non-wetting phase

A typical pair of relative permeability curves for a non-wetting phase is shown in Figure 34.1. The curve 1 to 2 represents the user-supplied drainage relative permeability table, and the curve 2 to 3 represents the user-supplied imbibition relative permeability table. (Note that non-wetting phase saturation increases from right to left in this diagram). The critical saturation of the imbibition curve (S_{ncri}) is greater than that of the drainage curve (S_{ncrd}). The two curves **must** meet at the maximum saturation value (S_{nmax}).

Figure 34.1 A typical pair of relative permeability curves for a non-wetting phase

The primary drainage curve is for a process which starts at the maximum possible wetting phase saturation, S_{wmaxd}. (This value will depend upon the end points of the saturation tables specified using the `SATNUM` keyword.) If the wetting phase saturation decreases to S_{wmin}, this primary drainage curve is used.

In a similar way, if the initial saturation is S_{wmin}, and the wetting phase saturation increases to S_{wmaxi}, the imbibition table data will be used. (The maximum wetting phase saturation which can be reached, S_{wmaxi}, is determined from the endpoints of the tables specified using the `IMBNUM` keyword, and will generally be less than S_{wmaxd}.)

If the drainage or imbibition process is reversed at some point, the data used does not simply run back over its previous values but runs along a **scanning curve**.

Consider a drainage process starting at point 1. If a full drainage process is carried out, the bounding drainage curve is followed to point 2. If an imbibition process then occurs, the water saturation increasing, the bounding imbibition curve is followed to point 3, the imbibition critical saturation.

But suppose that the drainage process is reversed at some intermediate point 4. A scanning curve results (curve 4 to 5 in the diagram). The critical saturation remaining at point 5 is the trapped critical saturation (S_{ncrt}), which is a function of the maximum non-wetting phase saturation reached in the run (S_{hy}).

If a further drainage process begins from any point on the scanning curve 5 to 4, the same scanning curve is retraced until S_{hy} is reached, at which point the drainage curve is rejoined. S_{hy} is updated during the run, so that further imbibition processes would occur along the appropriate scanning curves.

ECLIPSE 100

There is a choice of two methods for the generation of scanning curves from a given value of S_{hy} using Carlson's method [Ref. 20] or Killough's method [Ref. 19]. The choice of method is governed by Item 2 in keyword `EHYSTR`.

ECLIPSE 300

There is a choice of three methods for the generation of scanning curves from a given value of S_{hy} using Carlson's method [Ref. 20] or Killough's method [Ref. 19] or J. Jargon's method. The choice of method is governed by Item 2 in keyword `EHYSTR`.

Carlson's method for relative permeability hysteresis

Carlson's method produces a scanning curve that is parallel to the imbibition curve. It can be visualized by shifting the imbibition curve horizontally until it cuts the drainage curve at the saturation S_{hy}. When this method is chosen, it is important to ensure that the imbibition curve is always steeper than the drainage curve at the same K_r value. If this is not the case, the scanning curve could cross to the right of the drainage curve, which may produce a negative value of S_{ncrt}.

Killough's method for relative permeability hysteresis

Killough's method does not have such a simple geometric interpretation. For a given value of S_{hy} the trapped critical saturation is calculated as

$$S_{ncrt} = S_{ncrd} + \frac{S_{hy} - S_{ncrd}}{1 + C(S_{hy} - S_{ncrd})} \qquad \text{[EQ 34.1]}$$

where

$$C = \frac{1}{S_{ncri} - S_{ncrd}} - \frac{1}{S_{n\,\max} - S_{ncrd}} \qquad \text{[EQ 34.2]}$$

(Killough's formulae have been adapted to allow for non-zero values of S_{ncrd}.)

The relative permeability for a particular saturation S_n on the scanning curve is

$$K_{rn}(S_n) = \frac{K_{rni}(S_{norm})K_{rnd}(S_{hy})}{K_{rnd}(S_{n\,\max})} \qquad \text{[EQ 34.3]}$$

where K_{rni} and K_{rnd} represent the relative permeability values on the bounding imbibition and drainage curves respectively, and

$$S_{norm} = S_{ncri} + \frac{(S_n - S_{ncrt})(S_{n\,\max} - S_{ncri})}{S_{hy} - S_{ncrt}} \qquad \text{[EQ 34.4]}$$

With Killough's method S_{ncrt} will always lie between S_{ncrd} and S_{ncri}. But if the drainage and imbibition curves are made to coincide, the scanning curve will not necessarily follow this combined curve, except at its end points.

ECLIPSE 300

If the DRAINAGE keyword is specified, the scanning curve will be forced always to lie below the drainage curve.

The Killough method described above has been modified slightly to overcome an un-physical behavior when S_{hy} is small. As S_{hy} tends to S_{ncrd} in [EQ 34.1], S_{ncrt} tends to S_{ncrd}, which leads to an infinite imbibition relative permeability gradient. This in turn can cause convergence problems, while having little material effect on the simulation results. [EQ 34.1] has been modified as follows:

$$S_{ncrt} = S_{ncrd} + \frac{S_{hy} - S_{ncrd}}{A + C(S_{hy} - S_{ncrd})} \qquad \text{[EQ 34.5]}$$

where

$$A = 1 + a \cdot (S_{n\,\max} - S_{hy}) \qquad \text{[EQ 34.6]}$$

The parameter a is input through the fourth data item in the EHYSTR keyword and defaults to 0.1.

Jargon's method for relative permeability hysteresis

ECLIPSE 300

In this model the trapped saturation S_{ncrt} is constructed by moving the drainage critical saturation towards the imbibition critical saturation by the same fraction that the hysteresis saturation has moved towards the maximum nonwetting saturation:

$$S_{ncrt} = S_{ncrd} + \frac{(S_{ncri} - S_{ncrd})(S_{nhy} - S_{ncrd})}{(S_{nmax} - S_{ncrd})} \qquad \text{[EQ 34.7]}$$

We now construct a function which represents the ratio of imbibition and drainage curves as a function of the saturation value scaled between the drainage curve end point and the maximum saturation:

$$R(X) = \frac{K_{ri}(S_n)}{K_{rd}(S_n)} \qquad \text{[EQ 34.8]}$$

where X is defined from the non-wetting saturation S_n by

$$X = \frac{S_n - S_{ncrd}}{S_{nmax} - S_{ncrd}} \qquad \text{[EQ 34.9]}$$

Clearly X lies between 0 and 1 and R goes from 0 at $X = 0$ to 1 at $X = 1$ where the two curves meet. If the drainage curve is always above the imbibition curve, R also must lie between 0 and 1.

The scanning relative permeability value is then constructed by multiplying the drainage curve value by the function R, where R is evaluated at an X-value which reflects the fractional distance of the current saturation between the drainage curve end point and the hysteresis saturation:

$$K_{rn}(S_n) = R(X_s)K_{rnd}(S_n) \quad \text{[EQ 34.10]}$$

$$X_s = \frac{S_n - S_{ncrd}}{S_{hy} - S_{ncrd}} \quad \text{[EQ 34.11]}$$

This can also be written as

$$K_{rn}(S_n) = \frac{K_{ri}(\hat{S})}{K_{rd}(\hat{S})} K_{rnd}(S_n)\text{, where}$$

$$\hat{S} = S_{ncrd} + (S_n - S_{ncrd})\frac{(S_{nmax} - S_{ncrd})}{(S_{hy} - S_{ncrd})}$$

Differences in hysteresis treatment in three-phase cases

ECLIPSE 300

The treatment of relative permeability hysteresis differs between the Stone's method and default options.

In the STONE2 case, K_{row} is obtained at $S_n = 1 - S_w$ equal to the hydrocarbon saturation in the block. It is thus natural to take the maximum historical hydrocarbon saturation as S_{nhy} and to do hysteresis in the K_{row} part of the Stone II expression only. Gas relative permeability occurs in K_{rg} only.

In the default case, both K_{row} and K_{rog} are obtained at the grid block oil saturation, and hysteresis is done using this variable. The K_{rog} end points should normally be the same in the drainage and imbibition curves.

Consistency requirements

Certain consistency relationships between the end-points on the imbibition and drainage curves should be met in order to use the above models. As discussed above, the imbibition and drainage curves should meet at the maximum saturation $S_{n\,\max}$. Clearly also $s_{ncri} > s_{ncrd}$ is required.

ECLIPSE 100

ECLIPSE 100 checks these consistency relationships if the HYSTCHCK keyword is included in the PROPS section.

Relative permeability hysteresis in the wetting phase

There is an option to use the Killough model for wetting phase hysteresis. Otherwise the same curve will be used to obtain the wetting phase relative permeability in both drainage and imbibition processes (you can select either the drainage curve or the imbibition curve).

The option is selected in Item 2 of the `EHYSTR` keyword.

A typical pair of wetting phase relative permeability curves suitable for the Killough model is shown in Figure 34.2. The curve 1 to 2 represents the user-supplied drainage relative permeability table, and the curve 2 to 3 represents the user-supplied imbibition relative permeability table. The two curves **must** meet at the connate saturation ($S_{wco} = 1 - S_{nmax}$). The maximum saturation on the imbibition curve is $1 - S_{ncri}$.

Figure 34.2 A typical pair of relative permeability curves for a wetting phase

An initial drainage process would follow the drainage curve (point 1 to point 2). An imbibition process starting at point 2 ($S_w = S_{wco} = 1 - S_{nmax}$) follows the bounding imbibition curve (point 2 to point 3). Point 3 ($S_w = 1 - S_{ncri}$) is the maximum wetting phase saturation that can be reached starting from S_{wco}, since the trapped non-wetting phase saturation is S_{ncri}.

An imbibition process that starts from an intermediate saturation (point 4) will follow a scanning curve (point 4 to point 5). The saturation at point 4 is $S_w = 1 - S_{hy}$, where S_{hy} is the maximum non-wetting phase saturation reached. The maximum saturation that can be reached on the scanning curve (point 5) is $S_w = 1 - S_{ncrt}$, where S_{ncrt} is the trapped critical saturation of the non-wetting phase, as defined in the previous section.

If a further drainage process begins from any point on the scanning curve, the same scanning curve is retraced until point 4 is reached, where the drainage curve is rejoined.

Killough's method for calculating the scanning curves uses some of the quantities derived in the previous section for the non-wetting phase. The trapped critical non-wetting phase saturation S_{ncrt} is determined for the particular value of S_{hy}. The wetting phase relative permeability at the complementary saturation is calculated, thus fixing the position of point 5,

$$K_{rw}(1-S_{ncrt}) = K_{rwd}(1-S_{ncrt}) + (K_{rwi}(1-S_{ncri}) - K_{rwd}(1-S_{ncri}))\left(\frac{S_{ncrt}-S_{ncrd}}{S_{ncri}-S_{ncrd}}\right)^{A}$$

[EQ 34.12]

where the exponent A is a curvature parameter entered in Item 3 of the keyword EHYSTR. K_{rwd} and K_{rwi} represent the wetting phase relative permeability values on the bounding drainage and imbibition curves respectively.

The relative permeability for a particular saturation S_w on the scanning curve is

$$K_{rw}(S_w) = K_{rwd}(1-S_{hy}) + \frac{(K_{rw}(1-S_{ncrt}) - K_{rwd}(1-S_{hy}))K_{rwi}(1-S_{norm})}{K_{rwi}(1-S_{ncri})}$$

[EQ 34.13]

where S_{norm} is the function of S_n (= 1-S_w) defined in the non-wetting phase hysteresis section. As with Killough's non-wetting phase hysteresis model, if the drainage and imbibition curves are made to coincide the scanning curve will in general only meet this combined curve at its end points (points 4 and 5).

Modeling oil wet systems

It is possible to apply the non-wetting phase hysteresis models (Carlson or Killough or Jargon) to the water phase. This enables the modeling of trapped water by oil in an oil wet rock.

In the standard case, the `SATNUM` curves are used when Sw is decreasing, the `IMBNUM` curves when Sw is increasing. This still applies in an oil-wet case.

The gas phase will use the appropriate non-wetting phase model. The oil relative permeabilities will be taken from the drainage curves unless the Killough wetting phase model has been specified.

In order to be coherent with the physical meaning of an oil-wet rock, the way one has to input the saturation function is slightly modified if Killough wetting phase model is applied to oil (option 7, Item 2 of `EHYSTR` keyword). In these cases, we must have

$S_{g\,\max} = 1.0 - S_{oco}$ and also $S_{w\,\max} = 1.0 - S_{oco}$, while $S_{o\,\max}$ can be equal to 1.0

(S_{oco} is the **connate** oil saturation).

Note One has to be aware that if $S_{oco} > 0$ then $S_{w\,\max} < 1$ even in the aquifer.

This option is selected by setting the second data item in the `EHYSTR` keyword to 5 for the Carlson model and 6 for the Killough model. If option 7 is selected, the Killough model will apply to both the wetting and non-wetting phases.

Capillary pressure hysteresis

Given the method described in the previous section for calculating S_{ncrt}, the trapped non-wetting phase saturation, we now consider capillary pressure hysteresis. Again, the data available are the primary drainage and imbibition curves.

In the three-phase and gas-water cases both water and gas capillary pressure hysteresis occurs.

Water capillary pressure

Consider the water capillary pressure case shown in Figure 34.3.

Figure 34.3 The water capillary pressure case

Again, suppose the drainage process starting at point 1 is reversed at an intermediate saturation S_{hy} (point 4). The capillary pressure again traverses a scanning curve from point 4 to the trapped critical non-wetting phase saturation at 5. This curve is formed as a weighted average of the drainage and imbibition curves, starting with 100% drainage curve at point 4. The form used is that suggested by Killough:

$$P_c = P_{cd} + F(P_{ci} - P_{cd}) \qquad \text{[EQ 34.14]}$$

with

$$F = \left(\frac{1}{S_w - S_{why} + E} - \frac{1}{E}\right) \Big/ \left(\frac{1}{S_{wma} - S_{why} + E} - \frac{1}{E}\right) \qquad \text{[EQ 34.15]}$$

in which

E is a curvature parameter of the order of 0.1.

S_{why} is the water saturation at the hysteresis reversal point 4, that is the minimum historical water saturation in the cell

S_{wma} is the maximum water saturation attainable allowing for the trapped non-wetting phase saturation (that is at $S_n = S_{ncrt}$).

S_{wma} is $1-(S_{ocr}-S_{gcrt})$, the critical gas saturation reflecting the trapping saturation due to relative permeability hysteresis (depending on the choice of Carlson's Killough's or Jargon's method).

when $S_w \rightarrow S_{why}$, $F \rightarrow 0$ and $P_{cw} \rightarrow P_{cwd}$

and when $S_w \rightarrow S_w^{\max}$, $F \rightarrow 1$ and $P_{cw} \rightarrow P_{cwi}$.

The curvature parameter E can be entered in Item 1 of the `EHYSTR` keyword.

The imbibition capillary pressure P_{ci} in [EQ 34.14] is obtained by looking up the capillary pressure on the imbibition curve after scaling between the end-points S_{wcri} and S_{wma}, where S_{wcri} is the critical water saturation on the imbibition curve.

Suppose that a second reversal, back to a drainage process, occurs at point 6, rather than continuing to S_{wma}. Unlike the case of relative permeability hysteresis, this does not necessarily re-traverse the scanning curve (marked by a 'b' in the above diagram), but may follow a new drainage to imbibition to drainage scanning curve that returns to the point 4, marked by a 't' in the above diagram. The form of this is:

$$P_c = P_{ci} + G(P_{cd} - P_{ci}) \qquad \text{[EQ 34.16]}$$

with

$$G = \left(\frac{1}{S_{dep} - S_w + E} - \frac{1}{E}\right) \Big/ \left(\frac{1}{S_{dep} - S_{why} + E} - \frac{1}{E}\right) \qquad \text{[EQ 34.17]}$$

in which

S_{why} the water saturation at the hysteresis reversal point 4

S_{dep} is the departure saturation, the point at which G = 0.

In ECLIPSE, the default is for the imbibition scanning curve 'b' described by [EQ 34.14] to be **re-traversed** when this reversal back to drainage occurs.

ECLIPSE 100

To obtain the **new** scanning curve 't' described by [EQ 34.16] for secondary drainage, the sixth switch in the `EHYSTR` keyword should be set to '`NEW`'. However, the occurrence of multiple reversals onto different scanning curves may inhibit convergence in some cases.

The case in which an imbibition process is reversed onto an imbibition to drainage scanning curve is entirely analogous. (When the initial water saturation is less than the critical drainage saturation, the initial process must be imbibition.) The form of the scanning curve is

$$P_c = P_{ci} + F(P_{cd} - P_{ci}) \qquad \text{[EQ 34.18]}$$

with

$$F = \left(\frac{1}{S_{why} - S_w + E} - \frac{1}{E}\right) \Big/ \left(\frac{1}{S_{why} - S_{wcrd} + E} - \frac{1}{E}\right) \qquad \text{[EQ 34.19]}$$

in which

S_{why} is the water saturation at the hysteresis reversal point. In this case it is the minimum historical water saturation in the run.

S_{wcrd} is the critical water saturation on the drainage curve, which is the same for the scanning curves

When a second reversal back to imbibition occurs at a new departure point S_{dep} on the imbibition to drainage scanning curve, a second scanning curve is then followed during further imbibition from S_{dep} to S_{why}.

Gas capillary pressure

Similar expressions are used for gas capillary pressure hysteresis. In this case the gas saturation is the non-wetting saturation, and drainage to imbibition scanning curves return to the trapped critical gas saturation.

Consider the gas capillary pressure case shown in Figure 34.4.

Figure 34.4 The gas capillary pressure case

Analogously to the case of P_{cw},

$$P_c = P_{cd} + F(P_{ci} - P_{cd}) \qquad \text{[EQ 34.20]}$$

with

$$F = \left(\frac{1}{S_{ghy} - S_g + E} - \frac{1}{E}\right) \Big/ \left(\frac{1}{S_{ghy} - S_{gcr} + E} - \frac{1}{E}\right) \qquad \text{[EQ 34.21]}$$

in which

E is a curvature parameter of the order of 0.1.

S_{ghy} is the gas saturation at the hysteresis reversal point 4

S_{wma} is the maximum water saturation attainable allowing for the trapped non-wetting phase saturation (that is, at $S_n = S_{ncrt}$).

when $S_w \rightarrow S_{why}$, $F \rightarrow 0$ and $P_{cw} \rightarrow P_{cwd}$

and when $S_w \rightarrow S_w^{\max}$, $F \rightarrow 1$ and $P_{cw} \rightarrow P_{cwi}$.

Hysteresis in WAG Floods

Chapter 35

Introduction

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

This chapter describes the WAG (Water Alternating Gas) Hysteresis option.

The hysteresis models described in "Hysteresis" on page 517 can be thought of as two-phase models where a phase relative permeability is typically a function of its saturation and of its history. In WAG systems the relative permeability of a phase is also dependent on the history of the third phase. For example it has been observed that the gas relative permeability is higher when displacing pure oil than when displacing a water oil mixture following a water flood.

The WAG hysteresis model aims to provide a simple method of modeling these 3-phase effects. The model essentially consists of three components: a non-wetting phase model for the gas phase, a wetting phase model for water and a modification to the residual oil saturation in the STONE 1 three-phase oil relative permeability model.

The non-wetting gas phase hysteresis model is based on the theory developed by Land and Carlson. It differs from the standard ECLIPSE implementation of the Carlson model in that the imbibition curves are calculated analytically from the drainage curve and an input Land's parameter. In the gas phase the imbibition tables (`IMBNUM`) are ignored.

The wetting phase model (for the water phase) is based on input two-phase and three-phase relative permeability curves. In this case the `IMBNUM` curves are interpreted as the 3-phase water relative permeability, that is the relative permeability following a gas flood.

It is possible to utilize the standard hysteresis models in conjunction with the WAG hysteresis model. For example, oil phase hysteresis may be modeled in the usual way while the WAG hysteresis model is active.

The Land and Carlson theory is described in [Ref. 20] and [Ref. 21].

Using the WAG hysteresis option

The WAG hysteresis model is activated by using the WAGHYSTR keyword. The Hysteresis option must be enabled in the RUNSPEC section by specifying the item 'HYSTER' in the SATOPTS keyword. The model is only available in 3-phase runs.

The WAGHYSTR keyword requires a record of data for each saturation region (SATNUM). For each region the first data item, Land's parameter, must be specified to describe the imbibition curves. The second data item is the reduction factor used to describe the secondary drainage curve for the non-wetting phase (gas). The keyword in addition selects which models are active, enables the Carlson model to be modified to prevent infinite gradients in the imbibition relative permeabilities, and controls the threshold for moving from the two-phase model to the three-phase model.

The EHYSTR keyword can be used alongside the WAGHYSTR keyword to control the standard two-phase hysteresis models.

Both the eventual trapped gas saturation (the saturation that would be trapped if a water/oil flood continued) and the dynamic trapped gas saturation (the saturation deemed to be trapped at a given saturation state) can be output to the PRINT, RESTART and SUMMARY files. The SGTRAP mnemonic in the RPTSCHED keyword requests output of both arrays to the Print file. The equivalent arrays are output to the RESTART file provided a full (not graphics only) RESTART file is requested. The SUMMARY keywords BGTRP and BGTPD output grid block values to the SUMMARY files.

Non-wetting phase model (Gas)

Two-phase model

The gas phase model is based on the theory developed by Land and Carlson. The 2-phase model differs from the standard ECLIPSE implementation in that we use an analytic version of the Carlson model based on the input Land's parameter, and ignore the imbibition curve input using the IMBNUM keyword.

Consider a typical drainage process followed by an imbibition process (as shown in Figure 35.1).

Figure 35.1 A typical drainage process followed by an imbibition process

Consider a drainage process reaching a maximum gas saturation Sg_m followed by a imbibition process leading to a trapped gas saturation Sg_{trap}. The trapped gas saturation Sg_{trap} is given by

$$Sg_{trap} = Sg_{cr} + \frac{(Sg_m - Sg_{cr})}{(1 + C \cdot (Sg_m - Sg_{cr}))} \qquad \text{[EQ 35.1]}$$

where

C is Land's parameter, input with the WAGHYSTR keyword, or specified by the transition parameter as discussed later.

Note As for the generic hysteresis calculation, the Killough method has been modified slightly to overcome non-physical behavior when S_{gm} is small. As S_{gm} tends to S_{gcr} in [EQ 35.1], S_{gtrap} tends to S_{gcr}, which leads to an infinite imbibition relative permeability gradient. This in turn can cause convergence problems, while having little material effect on the simulation results. [EQ 35.1] has been modified as follows:

$$S_{gtrap} = S_{gcr} + \frac{S_{gm} - S_{gcr}}{A + C(S_{gm} - S_{gcr})}$$

where $A = 1 + a \cdot (S_{g\,\max} - S_{gm})$. The parameter a is input through the fourth data item in the EHYSTR keyword and defaults to 0.1.

The gas relative permeability on the scanning curve is given by:

$$K_{rg}(Sg) = K_{rg}^{drain}(Sg_f) \qquad \text{[EQ 35.2]}$$

where:

$$Sg_f = Sg_{cr} + \frac{1}{2}\left\{(Sg - Sg_{trap}) + \sqrt{(Sg - Sg_{trap})^2 + \frac{4}{C}(Sg - Sg_{trap})}\right\} \quad \text{[EQ 35.3]}$$

This transformation typically has an infinite gradient dSg_f/dSg when Sg is close to Sg_{trap}. As the input drainage relative permeability functions are represented as piece-wise linear functions, the final relative permeability function also has an infinite gradient. As this causes convergence problems the transformation Sg to Sg_f is modified to a linear function near to the trapped gas saturation. The proportion of the transformation that departs for the pure Carlson equation can be controlled using the sixth data item in the `WAGHYSTR` keyword.

Note In order to make a section of the imbibition curve linear to prevent an infinite gradient in the final relative permeability curve, the Carlson mapping algorithm is applied and the Land's parameter C in [EQ 35.1] and [EQ 35.3] can be replaced by the transition parameter C^{trans} defined as

$$C^{trans} = \frac{1}{S_{ncri} - S_{ncrd}} - \frac{1}{S_{n\,\max} - S_{ncrd}}$$

If the gas saturation increases following an imbibition process, the gas relative permeability follows the imbibition curve provided that the model remains in 2-phase mode, that is if the displacing phase is oil. The criterion for the model to remain in 2-phase mode is that the water saturation at the start of the secondary drainage process must be less than the connate water saturation plus a threshold value. The threshold value is input in the seventh data item in the `WAGHYSTR` keyword, which defaults to 0.001.

Note that if the imbibition curve is retraced and the saturation reaches Sg_m then the primary drainage curve is followed for increasing gas saturation.

If the water saturation exceeds the connate saturation plus the threshold value, the three-phase model described below is used.

Dynamic trapped saturation

Sg_f in equation [EQ 35.3] can be interpreted as the free gas saturation for a given saturation state. Hence the trapped gas saturation is given by

$$Sg_{td} = Sg - Sg_f \quad \text{[EQ 35.4]}$$

Three-phase model

The gas relative permeability follows a secondary drainage curve when the gas saturation increases and where the water saturation exceeds the connate value at the turn-around point. A typical secondary drainage curve is illustrated in Figure 35.2.

Figure 35.2 Schematic diagram showing a typical secondary drainage curve

These secondary drainage curves are calculated using the following equation

$$K_{rg}^{drain} = [K_{rg}^{input} - K_{rg}^{input}(Sg^{start})] \cdot \left[\frac{Sw_{co}}{Sw^{start}}\right]^{\alpha} + [K_{rg}^{imb}(Sg^{start})] \qquad \text{[EQ 35.5]}$$

Where:

K_{rg}^{drain} is the calculated secondary drainage relative permeability as a function of Sg

K_{rg}^{input} is the input relative permeability at Sg

$K_{rg}^{input}(Sg^{start})$ is the input relative permeability at the gas saturation at the start of the secondary drainage curve

Sw_{co} is the connate water saturation

Sw^{start} is the water saturation at the start of the secondary drainage curve.

$K_{rg}^{imb}(Sg^{start})$ is the relative permeability at the start of the secondary drainage process (that is the K_r at the end of the imbibition curve)

α is the reduction exponent input in the `WAGHYSTR` keyword.

Subsequent imbibition curves are treated in an analogous way to an imbibition process from the primary drainage curve. The saturation is transformed using the Carlson equation [EQ 35.3] and looked up on the secondary drainage curve. The trapped gas saturation is taken as the maximum historical trapped gas saturation, not the trapped gas saturation for this cycle.

If the transformed saturation Sg_f falls below the saturation at the start of the secondary drainage process, the secondary drainage curve cannot be used. In this case the transformation is modified and the primary drainage curve is used.

Note WAG (Water Alternating Gas) Hysteresis option can be applied to calculate the gas trapped saturation during CO2 storage process. Although, WAG hysteresis option was initially developed for three-phase system, it is also applicable now in `GASWAT` and `CO2STORE` options, which describe a two-phase system. In `GASWAT` and `CO2STORE` options, three-phase constitutive equations for WAG Hysteresis are applied to a two-phase system.

Modification to the residual oil

If the STONE 1 three-phase oil relative permeability model is employed, (see "Stone's first model (Modified)" on page 880) optionally it is possible to modify the SOM parameter to account for the trapped gas saturation. If this modification is employed, the assumption is made that original SOM represents the trapped hydrocarbon rather than just the trapped oil.

$$SOM_{mod} = SOM - (a \cdot Sg_{td}) \qquad \text{[EQ 35.6]}$$

where

SOM is the minimum residual oil saturation

Sg_{td} is the dynamic trapped gas saturation given by [EQ 35.4]

a is input as item 8 of the `WAGHYSTR` keyword.

Wetting phase model (Water)

The wetting phase model is based on the observation that the water relative permeability curve measured following a gas flood shows significantly less mobility. An example of the two relative permeability curves is shown in Figure 35.3.

Figure 35.3 Two-phase and three-phase relative permeability curves

The two-phase curves are taken to be the drainage curves (SATNUM) and the three-phase curves to be the imbibition curves (IMBNUM). Note that these curves are not strictly drainage and imbibition but apply to the two-phase and three-phase cases respectively.

For an imbibition process, with Sw increasing, the relative permeability function used is interpolated between the two-phase and three-phase curves using the following equation

$$K_{rw}^{imb} = K_{rw2} \cdot \left(1 - \frac{Sg^{start}}{Sg_{max}}\right) + K_{rw3} \cdot \left(\frac{Sg^{start}}{Sg_{max}}\right) \qquad \text{[EQ 35.7]}$$

where:

K_{rw}^{imb} is the calculated imbibition relative permeability

K_{rw2} is the two-phase relative permeability at Sw

K_{rw3} is the three-phase relative permeability at Sw

Sg_{max} is the maximum attainable gas saturation, $1 - Sw_{co} - So_{gcr}$

Sg^{start} is the gas saturation at the start of the imbibition process

Sw_{co} is the connate water saturation

So_{gcr} is the critical oil-to-gas saturation.

A subsequent drainage relative permeability is calculated by interpolation between the imbibition curve and either the three-phase curve or the two-phase curve, depending on the gas saturation. A typical case is shown in Figure 35.4.

Figure 35.4 Subsequent drainage relative permeability

An initial imbibition process may reach point A. The subsequent drainage process in this case moves to point B, on a curve interpolated between the original imbibition curve and the input three-phase curve.

The drainage curve moves either side of the imbibition curve depending on the prevailing gas saturation relative to the gas saturation at the start of the drainage process Sg^{sdrain}.

If $Sg > Sg^{sdrain}$ then

$$K_{rw}^{drain} = K_{rw}^{imb} \cdot \left(1 - \frac{Sg - Sg^{sdrain}}{Sg_{max} - Sg^{sdrain}}\right) + K_{rw3} \cdot \left(\frac{Sg - Sg^{sdrain}}{Sg_{max} - Sg^{sdrain}}\right) \qquad \text{[EQ 35.8]}$$

If $Sg < Sg^{drain}$ then

$$K_{rw}^{drain} = K_{rw}^{imb} \cdot \left(1 - \frac{Sg^{sdrain} - Sg}{Sg^{sdrain}}\right) + K_{rw2} \cdot \left(\frac{Sg^{sdrain} - Sg}{Sg^{drain}}\right) \qquad \text{[EQ 35.9]}$$

Where:

Sg^{sdrain} is the gas saturation at the start of the drainage process, point A

K_{rw}^{drain} is the calculated drainage curve

K_{rw}^{imb} is the previous imbibition relative permeability at Sw

K_{rw2} is the input two-phase curve at Sw

K_{rw3} is the input three-phase curve at Sw.

Subsequent secondary drainage processes follow an interpolated curve in much the same manner as the primary imbibition, but using a modified Sg^{start} arranged such that the curve runs through point B.

Independent Reservoir Regions

Chapter 36

Introduction

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL

The Independent Reservoir Region facility allows the efficient simulation of models containing two or more reservoirs that only communicate with each other through the well controls.

If there are no flow connections between the two reservoirs (through normal transmissibility, non-neighbor connections, or multiple-connected wells), the linear equations solved by ECLIPSE for the two reservoirs are independent. The solution of the two sets of linear equations in general require different numbers of linear iterations.

The Independent reservoir option allows ECLIPSE to utilize this potential efficiency gain.

Note This option is not compatible with the parallel option.

Using the option

1. Set NRFREG (the maximum number of independent reservoir regions) in item 3 of the RUNSPEC section keyword REGDIMS.
2. Use the ISOLNUM keyword in the GRID section to establish the separate regions.

   There must be at least one active cell allocated to each reservoir region between 1 and NRFREG. Any cells given a reservoir region number of 0 or > NRFREG are treated as inactive.

   The ISOLNUM data can be written out to the debug file for an existing model by using the RPTISOL keyword in the GRID section. The ISOLNUM array can be subsequently added to the dataset.
3. The ISOLNUM array can be output to the PRINT file using the mnemonic ISOLNUM in the RPTGRID keyword.
4. If the mnemonic NEWTON in the RPTSCHED keyword is activated, a separate summary line for each reservoir is printed at each Newton iteration.

Merging two or more corner point grids

Corner point grids are usually supplied for the entire grid from a pre-processing program such as PETREL. However, if a number of separate models have been run independently and are later required to be run as a single model, there are a number of facilities in ECLIPSE 100 to enable the corner point grids to be input separately.

If the separate grids are to be placed side by side in the X or Y directions, the COORD and ZCORN keywords can be used in conjunction with a BOX keyword to input the data for a portion of the field. Note that the grids should be separated by a row of inactive cells to prevent the coordinate lines overlapping. Also any inactive parts of the merged model should have dummy COORD and ZCORN values input; the cells can then be made inactive using the ACTNUM keyword. See "Example 1" on page 540.

If the separate grids are to be 'stacked' in the Z direction, either the grids must use the same coordinate lines or the multiple reservoir option must be employed (set NUMRES > 1 in the RUNSPEC section keyword NUMRES). If NUMRES is set > 1, you must identify which layers belong to which reservoir with the keyword COORDSYS. To combine the grid data for each reservoir into a single model, the RESVNUM keyword can be used. Data is then expected for a single reservoir at a time. A layer of inactive cells should be left between the separate grids, or a zero-transmissibility multiplier should be applied, to stop communication between the reservoirs. See "Example 2" on page 541.

Examples

Example 1

To merge a `2*4*1` grid with a `3*2*1` grid side by side in the X direction.

```
RUNSPEC section:
DIMENS
 6  4  1  /
GRID section:
BOX
 1 2   1 4   1 1   /
COORD....    for the first grid.
ZCORN....
BOX
 4 6   1 2   1 1   /
COORD....    for the second grid.
ZCORN....
EQUALS
 'ACTNUM' 0 3 3 1 4 1 1 /   inactivate the row between the grids.
 'ACTNUM' 0 4 6 3 4 1 1 /   inactivate the redundant region of
/                           the grid.
```

Example 2

To merge a `2*4*1` grid with a `3*2*1` grid, ‘stacked’ in the Z direction.

```
RUNSPEC section:
DIMENS
 3  4  3 /
NUMRES
 2 /
GRID section:
COORSYS
 1 2 /
 3 3 /
RESVNUM
 1 /
BOX
 1 2   1 4   1 1   /
COORD....    for the first grid.
ZCORN....
RESVNUM
 2 /
BOX
 1 3   1 2   3 3   /
COORD....    for the second grid.
ZCORN....
EQUALS
 'ACTNUM' 0  1 3  1 4  2 2 / inactivate the layer between the
                                                   grids.
 'ACTNUM' 0  3 3  1 4  1 1 /  inactivate the redundant region of
 'ACTNUM' 0  1 3  3 4  3 3 /  the grid.
/
```

Initializing the Study

Chapter 37

Introduction

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

The initial reservoir conditions can be defined in one of three ways:

1 They can be read from a RESTART file generated by an earlier run (see keyword RESTART).

2 They can be set directly in each grid block, using the keywords PRESSURE or PRVD, SWAT, SGAS, RS or PBUB, and RV or PDEW.

3 They can be calculated by the equilibration facility.

The equilibration facility is a means of calculating the initial conditions on the basis of hydrostatic equilibrium. If necessary, the reservoir can be divided into separate 'equilibration regions' in which hydrostatic equilibrium exists independently of the other regions. The number of equilibration regions (NTEQUL) must be specified using keyword EQLDIMS in the RUNSPEC section. Within each equilibration region all the grid blocks must use the same pressure table for their PVT properties, but they can use different saturation tables.

Data requirements

The data quantities required by the equilibration facility are entered using the keyword EQUIL. For each equilibration region, you should specify the pressure at a given datum depth, the positions of the gas-oil and water-oil contacts, and the value of the capillary pressure at each contact. If there is no gas cap, the gas-oil contact should be placed above the top of the reservoir; and likewise if there is no mobile water, the water-oil contact should be placed far enough below the bottom of the reservoir to exclude the transition zone.

Black oil

In ECLIPSE 100 and black oil mode problems in ECLIPSE 300, the dissolved gas concentration (R_s) or the oil's bubble point pressure (P_b) can be entered as a function of depth using the keywords RSVD and PBVD respectively. But this procedure is optional; if a table is not referenced, the dissolved gas concentration in under-saturated oil is everywhere set equal to the saturated R_s value at the gas-oil contact. If an R_s versus depth (or a P_b versus depth) table is required for any equilibration region, however, a table must be entered for each EQLNUM region using the keyword RSVD or PBVD. The maximum number of depth values in any such table must be set using keyword EQLDIMS in the RUNSPEC section. At any position in the reservoir, the R_s value derived from an R_s or P_b versus depth table is subject to an upper limit equal to the saturated value at the local pressure, since the R_s value cannot reasonably exceed this.

In black oil mode gas condensate problems, the vaporized oil concentration (R_v) in the gas can be set in a similar manner. Either R_v or the dew point pressure (P_{dew}) may be entered as a function of depth using the keywords RVVD and PDVD respectively. If a table of R_v versus depth (or P_{dew} versus depth) is not referenced by a particular equilibration region, the vaporized oil concentration in under-saturated gas is everywhere set equal to the saturated R_v value at the gas-oil contact. At any position in the reservoir, the R_v value obtained from a table or the default setting is subject to an upper limit equal to the saturated R_v value at the local pressure.

The datum depth for the pressure can lie anywhere in the equilibration region, unless the default setting of R_s or R_v is required. If in a live oil problem an R_s or P_b v depth table is not referenced for the region, the datum depth must be set at the gas-oil contact. The same restriction is imposed in gas condensate problems whenever an R_v or P_{dew} v depth table is not referenced.

In gas-water problems, instead of a gas-oil contact and a water-oil contact there is a gas-water contact. There is no need to enter any data concerning the non-existent oil phase. But it is also possible to have a gas-water contact in a three-phase problem; namely in a gas condensate problem where all the oil is initially vaporized in the gas phase. In this case the water-oil contact and the gas-oil contact should both be placed at the gas-water contact. In problems containing a gas-water contact, the gas-water capillary pressure should be supplied in the water saturation tables (keyword SWFN). For consistency, the capillary pressure data in the gas saturation tables should be set to zero.

Initial composition with respect to depth

In ECLIPSE 300 compositional mode, an extra complication exists: as well as ensuring that the initial saturations are in equilibrium, the thermodynamic equilibrium between phases must be considered. Three methods exist of setting up the initial composition variation, which is usually with depth:

1. For single hydrocarbon phase initial states, such as condensates above the dew point, or oils above the bubble point. No gas-oil phase contact exists within the reservoir. However, this option should be used for "supercritical" reservoirs in which a smooth transition exists between oil and gas, the saturation pressure being less than the fluid pressure throughout the depth range.

   Use `ZMFVD` to specify the initial composition with respect to depth. The gas-oil contact may be set above or below the reservoir if the single phase hydrocarbon is an oil or gas. In the supercritical case the gas-oil contact depth may be set to a point within the reservoir. The simulator sets the phase identification method to produce a phase change at this depth. For a condensate field the oil-gas contact depth is set to the oil-water contact depth. This identifies the study as a condensate to the simulator.

2. For cases in which there is a gas-oil contact within the reservoir, and the vapor composition is known.

   Use `ZMFVD` to specify the initial vapor composition. A retrograde dew point pressure calculation is performed for the vapor and the pressure is set to this at the contact. The composition of the liquid in equilibrium with the saturated vapor is used below the contact. As pressure increases with depth this is undersaturated.

   Alternatively, `COMPVD` may be used to specify a compositional variation in the liquid as well as in the vapor. However, the composition specified at the gas-oil contact must be that of a gas.

3. For cases in which there is a gas-oil contact within the reservoir, and the liquid composition is known.

   Use `ZMFVD` to specify the initial oil composition. A bubble point pressure calculation is performed for the liquid and the pressure is set to this at the contact. The composition of the vapor in equilibrium with the saturated liquid is used above the contact. As pressure decreases above the contact this may yield an oil phase upon flashing. If so, only the vapor is used, so that saturated vapor only exists above the contact.

   Alternatively, `COMPVD` may be used to specify a compositional variation in the vapor as well as in the liquid. However, the composition specified at the gas-oil contact must be that of an oil.

The condition that the liquid and vapor be in equilibrium, in the compositional case, is defined by [EQ 37.9]. Note that if this is satisfied, the liquid and vapor may be present in any amounts. Alternatively, the simulator sets an approximation for mole fractions of the oil and gas based on the phase fugacity functions [EQ 37.9] and recalculates the total fractions using [EQ 37.10] and [EQ 37.11].

The selection of the option required is made using the tenth argument of the `EQUIL` keyword. The 11th argument may be set to 1 to suppress the setting of the pressure at the contact to the saturation pressure at the contact in options 2 and 3 above. This yields a system that is not in initial equilibrium, but honors the specified pressure.

Equilibration methods such as those described above are the normal method of initializing a simulation. The aim is to set up a static initial configuration: one in which phases present are in equilibrium and in which the interblock flows are zero.

In some cases, however, it may be required to start a study from a point in existing production. An example might be initialization of a compositional study from the status of a black oil model at some stage in its history. As flow patterns have developed, this cannot be done by equilibration, and you must specify the initial pressures and saturations. In such a non-equilibrium initialization, a sufficient set of data is required to define the initial state. Pressures and saturations are defined using the `PRESSURE`, `SGAS` and `SWAT` keywords. Alternatively, these may be taken from an ECLIPSE 100 `RESTART` file using the `GETSOL` keyword.

In the compositional case the liquid and vapor compositions must also be specified. Two possibilities exist:

- Specify the liquid and vapor compositions explicitly using the `XMF` and `YMF` keywords.
- Enter tables of composition with respect to pressure using the `XMFVP` and `YMFVP` keywords. These tables can be generated by the PVT*i* package from constant volume depletion or differential liberation experiments to ensure that the chosen liquid and vapor are close to interphase equilibrium. This is to avoid large transient effects if initial interphase equilibrium is not valid.

This type of initialization is discussed in "Non-equilibrium initialization (ECLIPSE 300)" on page 553.

The equilibration algorithm

The main keyword involved in this is EQUIL, in the SOLUTION section. This enables the water-oil and oil-gas contact depths to be set. If either of these contacts is not required, it may be set outside the depth range of cells in the reservoir. For example, for an all-condensate initialization, set both contact depths below the lowest cell.

Given the fluid densities, the equilibration procedure sets up saturation against depth curves such that, in the transition zone when two phases are mobile, the hydrostatic pressure variation is balanced by the capillary pressure between the phases. This can be done on a finer grid than that used for the simulation, the resulting saturations being integrated over the cell volumes at the end of the equilibration to provide a starting point for the simulation.

In general, the use of a fine grid equilibration will yield a more accurate fluid in place, but may lead to some fluid movement upon starting the simulation.

Basic hydrostatic equilibration

The reservoir can be divided into a number of ‘equilibration regions’ in which hydrostatic equilibrium exists independently of other regions. The number of equilibration regions is specified in the RUNSPEC section using the EQLDIMS keyword, and cells are associated with these tables using the EQLNUM keyword. Cells in a given equilibration region should have the same pressure table number, and different cells in different equilibration regions should be non-communicating.

For each equilibration region, the fluid contact depths, the capillary pressures at these depths and the pressure at a given datum depth are specified. In black oil models the dissolved gas concentration or the bubble point pressure may be entered as a function of depth using the keywords RSVD and PBVD respectively. In gas condensate models the vaporized oil concentration or the dew point pressure may be entered as a function of depth using the keywords RVVD and PDVD respectively.

The datum depth for the pressure can lie anywhere in the equilibration region.

In gas-water problems, the water-oil contact and the gas-oil contact should both be placed at the gas-water contact. The gas-water capillary pressure should then be supplied as a function of the water saturation.

The water-oil and gas-oil contacts should also be placed at the same depth for three phase condensate cases in which the initial state is above the dew point.

Calculating the initial conditions

Within each equilibration region, the calculation is performed in two stages. The first stage sets up an internal table of phase pressures, and R_s and R_v for black oil mode simulations, against depth. The second stage interpolates this table to obtain the fluid conditions in each grid block in the region.

The internal table stores the values of the phase pressures (P_o, P_w, P_g). If in black oil mode, the dissolved gas concentration (R_s), and in gas condensate problems the vaporized oil concentration (R_v), are also stored at a number of depths within the equilibration region. The depth points are spaced equally throughout the region. The number of depth points in this table is set using the keyword `EQLDIMS` in the `RUNSPEC` section. The value should be increased when finer definition is required, for example in problems where the oil zone occupies only a small fraction of the overall reservoir thickness.

Between each pair of depth points in the table, the pressure gradient of each phase is calculated iteratively using a density consistent with the average pressure within the depth step. If the datum depth lies in the oil zone, the oil pressure values are calculated first, stepping from the datum depth up to the top and down to the bottom of the equilibration region. The water pressure on the oil-water contact can now be obtained, and the water pressure values are calculated. Similarly the gas pressure values are calculated starting from the gas-oil contact. In black oil mode, the R_s values are calculated from an R_s or P_b versus depth table supplied in the input data, or by defaulting to the saturated R_s value on the gas-oil contact. The R_s values are subject to an upper limit equal to the saturated R_s value at the local pressure. The R_v values are obtained in a similar manner in gas condensate problems.

In the second stage of the equilibration calculation, the local fluid conditions are determined in each grid block in the equilibration region. The internal table is interpolated to obtain the values of P_o, P_w, P_g, R_s and R_v at the grid block center depth. The water saturation is determined by inverse look-up of the water capillary pressure table (entered with keyword `SWFN` or `SWOF`) for the grid block, such that

$$P_{cow}(S_w) = P_o - P_w \qquad \text{[EQ 37.1]}$$

If $P_o - P_w$ exceeds the highest capillary pressure value in the `SWFN/SWOF` table (corresponding to the lowest saturation value S_{wmin}), the water saturation is set equal to S_{wmin}.

If $P_o - P_w$ is less than the lowest capillary pressure in the table (corresponding to the highest saturation value S_{wmax}), the water saturation is set equal to S_{wmax} and the oil pressure is adjusted to follow the water pressure gradient.

The gas saturation is similarly determined by inverse look-up of the gas capillary pressure table (keyword `SGFN`, `SGOF` or `SLGOF`) for the grid block, such that

$$P_{cog}(S_g) = P_g - P_o \qquad \text{[EQ 37.2]}$$

If $P_g - P_o$ is less than the lowest capillary pressure value in the table (corresponding to the lowest gas saturation value S_{gmin}), the gas saturation is set equal to S_{gmin}.

If $P_g - P_o$ exceeds the highest capillary pressure in the table (corresponding to the highest gas saturation value S_{gmax}), the gas saturation is set equal to S_{gmax} and the oil pressure is adjusted to follow the gas pressure gradient.

In black oil problems, the R_s value is simply interpolated from the internal table at the grid block depth. However, if there is a non-zero gas saturation at that depth the R_s value is re-set to the saturated R_s value at the local pressure. The R_v value is obtained in a similar manner in gas condensate problems.

Since the water and gas saturations outside the transition zones are set to the respective end-point values in the saturation function tables, the end-points of the two tables must be consistent. Normally the lowest gas saturation in the `SGFN` or `SGOF` table should be zero (so that $S_g = 0$ below the gas transition zone), and the highest gas saturation in this table should be $1 - S_{wmin}$ (so that $S_o = 0$ above the gas transition zone). The lowest water saturation value in the `SWFN` or `SWOF` table should be the connate value (so that $S_w = S_{wco}$ above the water transition zone), and the highest water saturation in this table should be 1.0 (so that $S_w = 1.0$ below the water transition zone). The consistency requirements between saturation tables are detailed in "Saturation Functions" on page 869.

In cases where the water-oil and gas-oil transition zones closely overlap, the treatment outlined above may produce negative oil saturations. For example, where the water-oil transition zone extends above the gas-oil transition zone, the water saturation ($S_w > S_{wmin}$) and the gas saturation ($S_g = S_{gmax}$) add to more than 1.0. If this occurs, the water and gas saturations are recalculated from the gas-water capillary pressure, which is taken as the sum of the water-oil and gas-oil capillary pressures

$$P_{cow}(S_w) + P_{cog}(S_g = 1 - S_w) = P_g - P_w \qquad \text{[EQ 37.3]}$$

ECLIPSE 300 thermal only

Note If the water-oil contact lies above the reservoir and if, based on the composition provided at the datum depth, oil is heavier than water, then the pressure in `EQUIL` item 2 refers to the oil phase and the reservoir will be initialized with heavy oil. A non-zero value for `OPTIONS3` item 217 can be used to restore the pre-2010.1 behavior for thermal heavy oil equilibration.

Accurate fluids in place calculation

There is a choice of methods to calculate the initial fluid in place values (the initial oil, water and gas saturations) in each grid block. These are selected using the ninth argument of the `EQUIL` keyword, which specifies the number of sub-layers (N) used to obtain the initial saturations:

Center-point equilibration (N=0)

The simulator sets the fluid saturations in each grid block according to the conditions at its center. This is the fastest method, and yields a stable initial state since the phase pressure differences in the simulation are also taken between cell centers. But it is the least accurate method, particularly in cases where the fluid contact passes through large grid blocks.

Horizontal fine grid equilibration (N<0)

The top and bottom halves of each grid block are each divided into -N layers of equal thickness, and the saturations are determined locally in each layer. The phase saturations for the block are set equal to the average of the saturations in each layer. This option provides a more accurate calculation of the fluids in place, but may yield a solution that is not completely stable in the initial state.

Tilted block fine grid equilibration (N>0)

This option is similar to the previous option, but it takes into account the slope of each grid block. The top and bottom faces of the blocks are treated as planes that are tilted about their central points. If the NEWTRAN keyword has been entered, the tilt is determined from the block corner point depths. Otherwise the faces are tilted towards the centers of the corresponding faces of the neighboring blocks within the equilibration region. The top and bottom halves the grid block are each divided into N horizontal layers of equal thickness, but the thickness of the layers in the top half is generally different from the thickness of the layers in the bottom half. This is because the distance from the block center line to the highest corner is not the same as the distance from the block center line to the lowest corner, if the upper and lower faces have different tilts. The phase saturations in each block are calculated as a weighted average of the saturations in the 2 N layers, weighted according to the area of each layer that is enclosed within the block multiplied by the layer thickness. This option provides the most accurate calculation of fluids in place, but may yield a solution which is not completely stable in the initial state.

ECLIPSE 100

With fine grid equilibration there is a redistribution of fluids between grid blocks near the contacts when the simulation begins, which occurs independently of any external driving force (wells etc.). The reason is that a steady state solution on the fine equilibration grid (in which each block is subdivided into several layers) is not necessarily a steady state solution on the coarser simulation grid. If the redistribution of fluids produces a significant transient when the simulation is started, you can overcome this by setting the ‘quiescence switch' (the switch QUIESC in keyword EQLOPTS in the RUNSPEC section). If this switch is on, modifications are applied to the phase pressures to make the initial solution a true steady state. These pressure modifications are applied for the duration of the run.

ECLIPSE 100

If the Vertical Equilibrium option is used (keyword VE in the RUNSPEC section), an exact VE equilibration method can be chosen by selecting center-point equilibration. Instead of the center-point calculation, the option calculates the fluid saturations in each grid block directly from the contact depths. It employs the same relationship between saturations and contact depths that is subsequently used for the simulation. This is the recommended option in VE runs.

The threshold pressure facility

If the reservoir contains two or more communicating equilibration regions having different pressures or contact depths, fluid flows between these regions immediately the simulation begins. The ‘quiescence switch’ only produces local steady state conditions within each separate equilibration region. Global quiescence, however, can be achieved by using the Threshold Pressure facility. The facility prevents flow between adjacent equilibration regions from occurring until the potential difference exceeds a specified threshold value. Thereafter the potential difference for flow across equilibration region boundaries is reduced by the appropriate threshold value.

The Threshold Pressure facility is enabled by including the switch THPRES in keyword EQLOPTS in the RUNSPEC section. Each interface between adjacent equilibration regions then has to have its own threshold pressure. If the threshold pressure is also irreversible (the switch IRREVERS in keyword EQLOPTS in the RUNSPEC section), each interface has two threshold pressures, one for flow in each direction. The threshold pressure values are set using the keyword THPRES. Values can be defaulted to the maximum potential difference between communicating cells across the interface (that is, just sufficient to prevent flow occurring across the interface before the reservoir is perturbed).

The Quiescence option

ECLIPSE 100

When using the fine grid equilibration algorithm to achieve a more accurate fluids-in-place calculation, the initial conditions may no longer be in hydrostatic equilibrium for each phase. This is because the saturations are now taken as average values over a grid block rather than the values at cell centers. The values of pressure in the fine grid equilibration are still taken from block center values.

The Quiescence option achieves hydrostatic equilibrium for flows of each phase. It modifies the initial (oil phase) pressure P and introduces cell-dependent modifiers $PMODW$ and $PMODG$ to the water and gas phase pressures such that

$$P_{wat} = P - P_{cow} + PMODW \qquad \text{[EQ 37.4]}$$

and

$$P_{gas} = P + P_{cog} + PMODG \qquad \text{[EQ 37.5]}$$

The phase pressure modifications $PMODW$ and $PMODG$ are determined to achieve quiescence at initial conditions and are then applied throughout the simulation. These can be thought of as frozen pseudo-capillary pressure modifications. The quiescent pressures are constructed from the initial tables of phase pressure versus depth, p_{oil}, p_{gas} and p_{wat} over the depth of the field. Then, in the absence of capillary pressures, $p_{oil} = p_{gas}$ at the oil-water contact and $p_{oil} = p_{gas}$ at the gas-water contact.

The (oil phase) pressure in each grid block center, P is modified by

$$P = max(P, p_{wat} - (\tilde{\rho}_{wat} - \tilde{\rho}_{oil})H/2, p_{gas} - (\tilde{\rho}_o - \tilde{\rho}_g)H/2) \qquad \text{[EQ 37.6]}$$

where H is the height of a cell, and $\tilde{\rho}$ denotes the phase gravity density.

The water and gas phase pressure modifications are then determined from

$$PMODW = sign(p_{wat} - P + P_{cow}) \times min(|p_{wat} - P + P_{cow}|, (\rho_w - \rho_o)H/2) \qquad \text{[EQ 37.7]}$$

and

$$PMODG = sign(P + P_{cog} - p_{gas}) \times min(|P + P_{cog} - p_{gas}|, (\rho_o - \rho_g)H/2) \qquad \text{[EQ 37.8]}$$

These phase pressure modifications are chosen such that the water phase pressure P_{wat} approximately follows the hydrostatic water pressure curve p_{wat} in the presence of mobile water, and likewise so that the gas phase pressure P_{gas} approximately follows the hydrostatic gas pressure curve p_{gas} in the presence of mobile gas.

Note that even for a model with zero capillary pressures, the phase pressure modifications is not necessarily zero. The phase pressure modifications can be reported using the mnemonic '`QUIESC`' in the `RPTSOL` keyword.

Non-equilibrium initialization (ECLIPSE 300)

ECLIPSE 300

Let us consider a case in which we wish to start a study from an explicitly entered state. This might be in equilibrium or have pressure gradients implying flows in the initial state. Such an initial state is defined by pressures and saturations for each grid block (which can be obtained from a black oil run), and liquid and vapor phase composition data. These compositions may be defined either explicitly, or be taken from a constant volume depletion or differential liberation experiment as functions of pressure. In the latter case a good approximation to inter-phase equilibrium may be obtained, even if the pressure varies over the field.

Specifying the initial state of a run

The initial state of a cell is defined by P, S_w and S_g. In the black oil case the bubble point may also be defined, while in the compositional case the mole fractions of the oil and gas must be specified, (x_i and y_i, $i = 1, \ldots, N_c$). The condition that the liquid and vapor be in equilibrium, in the compositional case, is:

$$f_{iL}(P, x) = f_{iV}(P, y) \qquad \text{[EQ 37.9]}$$

Note that if this is satisfied, the liquid and vapor may be present in any amounts. The mole fractions of oil and gas present may be established by obtaining their molar densities, and then the total mole fractions are obtained by:

$$z_i = x_i L + y_i V \qquad \text{[EQ 37.10]}$$

with

$$L = \frac{S_o b_o^m}{S_o b_o^m + S_g b_g^m} \qquad \text{[EQ 37.11]}$$

where b_o^m and b_g^m are the oil and gas molar densities. As long as [EQ 37.9] above is satisfied, flashing a system with total composition z_i then reproduces the original values of L, V, x_i, y_i, S_o and S_g.

If only pressure and saturation data is available, this presents a problem. A single oil and a single gas composition may be specified. This, however, can only be in equilibrium at a single pressure, and [EQ 37.9] is not satisfied at any other pressure. The result is that the initial flash yields different liquid and vapor mole fractions and saturations from those specified, and there may well be serious convergence problems in the initial state. Even, if these are resolved, the system generally no longer has the required saturation values.

Apart from the oil and gas compositions, the interphase equilibrium condition only involves the pressure P. If a set of equilibrium x_i and y_i values are available parameterized by pressure, it would be possible to start cells at any pressure in equilibrium.

Such a parameterized set of compositions can be obtained from a constant volume depletion or differential liberation experiment. In these, for a condensate or an oil respectively, vapor is extracted at each pressure step that is in equilibrium with the liquid. Further, these experiments attempt to simulate the depletion process that occurs in the reservoir, so are likely to yield representative composition versus pressure tables. Such an experiment can be performed in the PVT*i* program, and the resulting XMFVP and YMFVP (x_i mole fraction and y_i mole fraction versus pressure) tables used as input to ECLIPSE 300. The pressure steps must be small to avoid interpolation errors in the table, and the range of pressures must span that occurring in the reservoir in initial state.

In some areas of the reservoir the oil may be under- saturated. In this case a bubble point exists, the pressure at which the cell would become two-phase. Only oil exists in this cell, and if it is given a composition taken from the equilibrium oil composition at P_b rather than P, it must be in equilibrium with gas at P_b, and so have the correct saturation pressure.

Using this method, it is possible to set up a set of representative compositions such that interphase equilibrium is observed for any specified cell pressures.

Keyword input for non-equilibrium initialization

Two options are available for specifying the initial pressure and saturation distributions:

1. Specify using keywords. The PRESSURE, SGAS, SWAT and PBUB keywords may be entered like grid keywords. These enable values of P, S_g, S_w and P_b to be specified in each cell. Like GRID and EDIT section keywords, these may also be set using EQUALS, ADD and MULTIPLY applied to boxes of cells.
2. Read from an ECLIPSE 100 RESTART file. The values of P, S_g, S_w and P_b may also be read from an ECLIPSE 100 RESTART file using the GETSOL keyword. The format of this is:

```
GETSOL
PRESSURE BASE1 10 /
SWAT BASE1 10 /
/
```

This would imply that the pressure and water saturation were to be obtained from the 10th report step of the RESTART file with root name BASE1. (Such files are also produced by ECLIPSE 300, controlled by the OUTSOL keyword, to act as an interface to GRAF.)

In the compositional case the liquid and vapor compositions must be specified. Again, two possibilities exist:

1. Specify the liquid and vapor compositions explicitly using the XMF and YMF keywords. These enter values for every cell and for every component.
2. Enter tables of composition with respect to pressure using the XMFVP and YMFVP keywords. These tables would usually be generated by the PVT*i* package from constant volume depletion or differential liberation experiments, and would specify a set of equilibrium liquid and vapor compositions as a function of pressure.

Phase Pressures in ECLIPSE 300

By default, oil and gas phase pressures are derived from the pressure solution variable P using the formula

$$Poil = P - Pcog \times \left(\frac{S_g}{So + Sg}\right)$$

and

$$Pgas = P + Pcog \times \left(\frac{S_o}{So + Sg}\right)$$

where $Pcog$ is the gas-oil capillary pressure, S_o and S_g are the oil and gas saturations respectively. This Pc splitting technique is performed to enable better convergence and property smoothness above the critical point. It may be disabled using keyword FORMOPTS, option NOPCSPT. When reporting initialization data using the RPTRST, RPTSOL or OUTSOL keywords, the mnemonic PRESSURE is the solution variable P which is derived from the phase pressures calculated during the EQUIL hydrostatic equilibration and reported with the SOLVD option in RPTSOL, POIL is the oil phase pressure P_{oil} and PGAS is the gas phase pressure P_{gas} as discussed in the above two equations.

Note The Pc-splitting technique applies only to the ECLIPSE 300 compositional mode.

Licenses

Chapter 38

License Overview

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

To run ECLIPSE, and also to run any of the additional ECLIPSE options, you need to obtain an appropriate FLEX license.

The FLEX license checking procedure checks for the availability of an appropriate license for an ECLIPSE option when a keyword activating the feature is encountered. In most cases, the activating keyword is in the RUNSPEC section, so is encountered at the beginning of the run.

Activating keywords for some features, however, may not be encountered until the SCHEDULE section, and perhaps even part way through a run. For example, the Standard Network option is activated by the GRUPNET keyword, and the Gas Field Operations model may not be activated until the SWINGFAC keyword is read. If an appropriate license is not available when a particular feature is activated, the run stops.

A DATACHECK license is available, which allows ECLIPSE to run a dataset in NOSIM mode to check for data errors without taking up a full license.

ECLIPSE 100

To avoid the run stopping part way through, it is possible to reserve all the required licenses at the beginning of the run, using the LICENSES keyword in the RUNSPEC section to identify the features in the run for which licenses will be required.

Note The LICENSES keyword is used by the ECLIPSE launchers to work out which licenses are used, except for parallel licenses, in order to implement the license aware job scheduling for ECLIPSE 100 and ECLIPSE 300.

This chapter provides information about the licenses used by specific simulators. A description of the keywords required to invoke these features is also given.

There are a number of special cases which will be dealt with separately:

- Open-ECLIPSE
- Parallel
- Multiple Realization (MR).

Licenses used by specific simulators

In most cases, the licenses being used by the simulators are quite straight forward to work out. A base `eclipse` license is checked out and, depending on other factors, additional licenses may also be used.

Primarily it is the keywords in the dataset that determine which licenses are required. Standard usage is described below along with some exceptions and special cases.

ECLIPSE 100 uses a base `eclipse` license. A more complex example involving reservoir coupling is as follows:

> It is possible to couple several ECLIPSE models together. See "Reservoir Coupling" on page 811. If a dataset is coupled to others, it will contain the `SLAVES` keyword. In this case, the total license requirement is the same as if all the datasets had been run concurrently, with the additional requirement for a `rescoupling` license. For example, if the following datasets are coupled:
>
> a A master dataset which only requires an `eclipse` license.
>
> b A slave which requires an `eclipse`, `lgr` and a `networks` license.
>
> c Another slave that requires an `eclipse`, `lgr` and a `multisegwells` license.
>
> The total license usage is 3x `eclipse`, 2x `lgr`, 1x `networks`, 1x `multisegwells` `and` 1x `rescoupling`.

ECLIPSE 300 typically requires a `compositional` and an `eclipse` license. Again, there are several possibilities.

a If the ECLIPSE 300 dataset contains the `THERMAL` keyword, then ECLIPSE 300 will only checkout an `eclipse` and a `thermal` license.

b If the user has an `e300` license available it will be used instead of an `eclipse` and a `compositional` license. However, the `e300` license is no longer sold.

c For shale gas, if the user launches a dataset with `COMPS=1` and the `CBM` keyword in the `RUNSPEC` section, then the user will only need an `eclipse` and a `coalbed` license to run ECLIPSE 300.

d If the dataset contains the `BLACKOIL` keyword, a user can run ECLIPSE 300 in blackoil mode with an `eclipse` license and there is no need for a `compositional` license. Similarly if the dataset contains the `TEMP` keyword, the user can run ECLIPSE 300 in blackoil mode with an `eclipse` license and there is no need for a `thermal` license.

Which keywords invoke which features?

The table below shows the feature names, the keywords that invoke the functionality and the license key name. Where two license names appear, the simulator will attempt to checkout the first one and if this fails it will attempt to checkout the second.:

Table 38.1 Feature names, keywords that invoke the functionality and the license key name

License Feature Name	Keywords	License Key	ECLIPSE 100	ECLIPSE 300
CO2 EOR & Storage (RENTAL ONLY)	CO2STORE	co2_eor_storage		x
Coal Bed Methane	COAL, CBMOPTS, DUALPORO, NMATRIX	coalbed	x	x
Compositional	COMPS	compositional		x
Datacheck	NOSIM	datacheck	x	x
ECLIPSE SynSize	PEDIMS, GRADRESV	eclipse4d	x	
Flux Boundary	REGDIMS item 4 specifies the maximum number of flux regions in the run, DUMPFLUX, USEFLUX	flux	x	x
EOR Foam	FOAM	foam	x	
Well Bore Friction (MAINTENANCE MODE)	FRICTION	friction multisegwells	x	x
Lagrangian Tracer Solver	LAGTRAC	fs_streamtrace frontsim		x
Gas Field Operations	GASFIELD, GASPERIO, GASYEAR, SWINGFAC	gasfield	x	x
Gas Lift Optimization	GLIFTOPT, LIFTOPT, WLIFTOPT	gaslift	x	x
Gas Calorific Value Based Control	CALTRAC, DCQDEFN, GCALECON, GCONCAL, GCONENG, GUIDECAL, WCALVAL	gasquality	x	
Geomechanics	GEOMECH	geomechanics		x
Pseudo-Compositional	GIMODEL	gioption	x	

Table 38.1 Feature names, keywords that invoke the functionality and the license key name

License Feature Name	Keywords	License Key	ECLIPSE 100	ECLIPSE 300
Local Grid Refinement/ Coarsening	LGR, CARFIN, RADFIN, AUTOREF, COARSEN, AUTOCOAR	lgr	x	x
Multiple Realization	MULTREAL is generated by a pre-processor	multiple_realisation	x	x
Multi-segmented wells	WSEGDIMS, WSEGSOLV	multisegwells	x	x
Networks	NETWORK, GNETINJE, GRUPNET	networks	x	x
OpenECLIPSE Runtime License	Called by controller	openeclipse	x	x
Parallel ECLIPSE	PARALLEL	parallel	x	x
EOR Polymer	POLYMER	polymers	x	
Proppant flowback in a fracture	PROPFLOW	proppantflow		x
Reservoir Coupling	DUMPCUPL, GRUPMAST, GRUPSLAV, RCMASTS, SLAVES, USECUPL	rescoupling	x	
Reservoir Optimization	RESOPT	resopt		x
EOR Solvent	SOLVENT	solvents	x	
EOR Surfactant	SURFACT, SURFACTW	surfactant	x	
Sand Management Advisor	WELLSMA	sma		x
Thermal	THERMAL	thermal		x
Environmental Tracer	TRACERS item 4 specifies the maximum number of environmental tracers, EQLDIMS, PARTTRAC	tracer	x	
Unencoded Gradients	FMTHMD, GDIMS, HMDIMS, RPTHMD, UNCODHMD	unencodedhmd	x	x
Water Services	RIVRDIMS, SAMG and summary vectors reporting hydraulic heads.	waterservices	x	

Special Cases

Open-ECLIPSE

Open-ECLIPSE enables ECLIPSE 100 and ECLIPSE 300 to be controlled by, and communicate interactively with, other applications. Although ECLIPSE is primarily used in a batch stand-alone mode controlled solely by the contents of the input data file, there are many situations in which it would be advantageous to have ECLIPSE controlled by another software application.

For instance, there may be a requirement to have ECLIPSE tightly coupled to a surface gathering system model, or a specialized production optimization application. It could also be useful to couple ECLIPSE to an interactive controller, to allow you to view the current status of the simulation and make well management decisions during the course of the run. In addition to the licenses that ECLIPSE usually checks out, the simulator will also request an `openeclipse` license.

Parallel

The parallel option allows the simulation of a single data set to be distributed across a number of processors. This allows large simulations to be carried out in a shorter time than would normally be the case.

Both ECLIPSE 100 and ECLIPSE 300 are optimized to provide the shortest time to solution, and consequently tailor the number of domains, and hence the linear solver, to the number of available processors.

This has a number of consequences. Firstly, the results obtained on different numbers of processors will agree to engineering accuracy, not to machine accuracy. Secondly, the scalability is sub-optimal, as the linear solver becomes less efficient as the number of processors increases.

Running ECLIPSE in parallel requires the same set of licenses as running in serial, plus a number of `parallel` licenses. The number of parallel licenses required is equal to the first argument of the `PARALLEL` keyword.

The number of `parallel` licenses are required irrespective of how the model is run. As an example, say an ECLIPSE model is split into 8 domains, which could be run in a number of configurations. For instance, 1 computer with 8 processors, or 2 computers running 4 processes each, or 8 computers running 1 process each. For all these cases, the model is split into 8 domains, so 8 `parallel` licenses would be required. If the model was split into 16 domains, then 16 `parallel` licenses would be required.

Multiple Realization

Multiple Realization (MR) licensing was implemented as a way to lower the cost of running multiple similar versions of the same model.

Firstly, the user must generate MR datasets as they cannot be made by hand. Examples of programs able to generate MR datasets include MEPO and Petrel. The dataset differs from a standard dataset in that it includes the `MULTREAL` keyword.

Following the `MULTREAL` keyword is a UID (Unique Identifier), which is a long string of numbers and a word ('yes' or 'no'). All MR datasets generated from the same base case at the same time will have the same UID. The word, Yes or No, tells ECLIPSE whether to continue if MR licenses are not available. This is however not yet implemented in the generating applications.

When the first of a group with a single UID starts running, the standard set of licenses that would be required for that job is checked out. In addition, an MR license is checked out as well. When the second and subsequent jobs from the same group start, they each only checkout an MR license. Even if the first job finishes, the standard set of licenses are kept checked out until all running jobs from that group finish.

Parallel and reservoir coupling are excluded from this licensing scheme, and if any dataset includes these features, they will be checked out as normal. The example in Figure 38.1 shows 6 MR jobs running through a system that can handle 3 jobs at a time.

Figure 38.1 Example shows 6 MR jobs running through a system that can handle 3 jobs at a time

Lagrangian Tracer Solver

The Lagrangian tracer solver feature uses a FrontSim streamline (`fs_streamtrace`) license in the first instance. However, if no `fs_streamtrace` license is available then a full `frontsim` licence will be taken instead. A Petrel locked frontsim license will not activate this feature.

Listing of available licenses

The list of available licenses is not output by default. This output can be enabled by setting the `LISTLICENSES` keyword to `TRUE` in the configuration file.

Local Grid Refinement and Coarsening

Chapter 39

Introduction

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL

The Local Grid Refinement option allows enhanced grid definition near wells. The local models may be 2D radial, 3D radial or 3D Cartesian. Local models may have more layers than the global model. Transmissibilities between the local models and the global model are computed automatically by ECLIPSE. The properties of the cells in the local grid can be inherited from the Global grid or specified explicitly for the refined cells.

Grid coarsening may be used to amalgamate cells in regions of the reservoir where accuracy is unimportant.

ECLIPSE 100

The autorefinement facility may be used to refine an existing global model as the reverse process to grid coarsening.

Local grid refinement

Defining radial local grid refinements

Radial local grid refinements may be defined either in a single column of cells with the RADFIN keyword, or in a box of four columns with the RADFIN4 keyword.

Radial refinement in a single column

To refine column (I=7, J=8) in layers 3 to 5 inclusive with a 3D radial grid with 6 radial segments, 4 angular segments and 18 vertical segments use:

```
RADFIN
-- NAME     I J K1 K2 NR NTHETA NZ  NWMAX
 'SOUTH'    7 8  3  5  6   4    18 /
INRAD
 0.2 /
ENDFIN
```

in the GRID section. This inserts a local grid with 6*4*18 = 432 cells. The INRAD keyword specifies the inner radius of the innermost block of the radial refinement.

This creates a refinement that looks like Figure 39.1.

Figure 39.1 Single column radial grid refinement

Alternatively it is possible to use a two-dimensional model with NTHETA=1 that looks like Figure 39.2.

Figure 39.2 Single column radial grid refinement with NTHETA=1

By default, non-neighbor connections are generated automatically within the local grid to complete the circle connecting angular segment ITHETA=1 to ITHETA=4, unless circle completion is specifically disallowed by specifying 'INCOMP' in item 3 of the COORDSYS keyword entered between the RADFIN and ENDFIN keywords. The ENDFIN keyword tells ECLIPSE to finish reading GRID data for the local grid and to continue reading GRID data for the global grid. The permitted values of NTHETA (the number of angular segments) are 1 for 2D radial refinements and 4 for 3D radial refinements. NWMAX is the maximum number of wells in the refinement. If NWMAX is not specified it is defaulted to 1, which is the normal value in a radial refinement.

To reserve memory for the refinement, set MAXLGR = 1 and MAXCLS = 432 in the keyword LGR in the RUNSPEC section, for example:

```
LGR
--MAXLGR  MAXCLS
    4      432 /
```

where MAXLGR is the maximum number of local grid refinements and MAXCLS is the maximum number of cells in any local grid.

Radial refinement in a box of four columns (ECLIPSE 100)

To refine the box of four columns (I=7,8; J=8,9) in layers 3 to 5 inclusive with a radial grid having 6 radial segments, 8 angular segments and 18 vertical segments use:

```
RADFIN4
-- NAME     I1  I2  J1  J2  K1  K2  NR  NTHETA  NZ    NWMAX
  'SOUTH'   7   8   8   9   3   5   6   8       18  /

INRAD
 0.2  /
ENDFIN
```

in the GRID data section. This inserts a local grid with 6*8*18 = 864 cells. The INRAD keyword specifies the inner radius of the innermost block of the radial refinement.

This creates a refinement which looks like Figure 39.3.

Figure 39.3 Radial refinement in a four column box

Alternatively, setting NTHETA=4 creates a refinement which looks like Figure 39.4.

Figure 39.4 Radial refinement in a four cell box with NTHETA=4

By default, non-neighbor connections are generated automatically within the local grid to complete the circle connecting angular segment `ITHETA`=1 to `ITHETA`=8, unless circle completion is specifically disallowed by specifying '`INCOMP`' in item 3 of the `COORDSYS` keyword entered between the `RADFIN4` and `ENDFIN` keywords. The `ENDFIN` keyword tells ECLIPSE to finish reading `GRID` data for the local grid and to continue reading `GRID` data for the global grid. The permitted values of `NTHETA` (the number of angular segments) are 4 or 8. `NWMAX` is the maximum number of wells in the refinement. If `NWMAX` is not specified it is defaulted to 1, which is the normal value in a radial refinement.

To reserve memory for the refinement, set `MAXLGR` = 1 and `MAXCLS` = 864 in the keyword `LGR` in the `RUNSPEC` section, for example:

```
LGR
--MAXLGR   MAXCLS
    4       864 /
```

Placing wells in local grids

To place a well in a local grid, use keywords `WELSPECL` and `COMPDATL` instead of `WELSPECS` and `COMPDAT`. `WELSPECL` is similar to `WELSPECS` except that there is an additional item - the name of the local grid - between the group name and the I-J location. `COMPDATL` has exactly the same items as `COMPDAT`. In these keywords the I, J and K locations of the wellhead and the connecting cells refer to the cell coordinates in the local grid system. The I and J locations of the connecting cells must be entered explicitly in `COMPDATL`, and should not be defaulted as they can be in `COMPDAT`.

To complete well PRODUCER in layers 9 to 11 of the refined grid named 'SOUTH', in all four theta sections of the innermost block:

```
WELSPECL
--   WELL   -- GROUP -- LOCAL GRID -- LOCATION --  BHP  --  PREF
--   NAME   -- NAME  --    NAME    --  I   J   -- DEPTH --  PHASE
 'PRODUCER'    'G'       'SOUTH'       1    1      8060     'OIL'  /
/
COMPDATL
--   WELL   --   LOCATION   --  OPEN  -- SAT -- CONN -- WELL
--   NAME   -- IR JTH K1 K2 --  SHUT  -- TAB -- FACT -- DIAM
 'PRODUCER'    1  1   9  11    'OPEN'     0      -1     0.4  /
 'PRODUCER'    1  2   9  11    'OPEN'     0      -1     0.4  /
 'PRODUCER'    1  3   9  11    'OPEN'     0      -1     0.4  /
 'PRODUCER'    1  4   9  11    'OPEN'     0      -1     0.4  /
/
```

ECLIPSE 300 allows wells to be completed in any combination of local grids. With ECLIPSE 100 a well cannot be completed in a cell outside its local grid. However, it is possible to complete a single well in two or more Cartesian local grids by amalgamating these grids into a single entity (see "Amalgamating cartesian local grids (ECLIPSE 100)" on page 571).

Wells not completed in local grid systems should have their specification and connection data set with the keywords `WELSPECS` and `COMPDAT` as normal. ECLIPSE checks that these wells are not completed in a cell that is host to a refined grid system.

Cartesian refinements

Cartesian refinements are specified by keyword `CARFIN`. For example, to replace layers 3 and 4 in column (2,3) of the global model with a 5*5*8 Cartesian refinement with new porosities and permeabilities:

```
CARFIN
-- NAME   I1-I2 J1-J2 K1-K2 NX NY NZ NWMAX--
  'LGR1'  2  2  3  3  3  4  5  5  8  5      /
PORO
 200*0.2 /
EQUALS
 'PERMX' 500 /
 'PERMY' 500 /
 'PERMZ' 50 /
/
ENDFIN
```

This creates a refinement that looks like Figure 39.5 (in the areal direction).

Figure 39.5 Cartesian grid refinement

It is not necessary to insert ENDFIN between successive RADFIN, RADFIN4 and CARFIN keywords. The primary purpose of ENDFIN is to revert ECLIPSE to reading data for the global grid system.

Note that the CARFIN keyword is designed to refine a general box, rather than just a single column of cells as in RADFIN or a box of four columns of as in RADFIN4. The maximum number of wells within the Cartesian refinement is specified by NWMAX (=5 in the example above).

To place wells in a Cartesian refined grid, follow the same procedure as described above for radial grids. For example, to place a horizontal well that fully penetrates the refined grid from the x-direction, use the following procedure to complete the well in five horizontally-adjacent cells in the refined grid:

```
WELSPECL
--   WELL  --  GROUP -- LOCAL GRID -- LOCATION --  BHP  -- PREF
--   NAME  --  NAME  --    NAME    --   I  J   -- DEPTH -- PHASE
    'HORIZ'    'G'        'LGR1'        1  3       8060    'OIL'  /
/
COMPDATL
-- WELL ---- LOCATION --- OPEN - SAT - CONN - WELL - KH - S - D - PEN
-- NAME -- IX JY K1 K2 -- SHUT - TAB - FACT - DIAM -    -   -   - DRN
 'HORIZ'    1  3  3  3   'OPEN'   0     -1     0.4   3*          'X' /
 'HORIZ'    2  3  3  3   'OPEN'   0     -1     0.4   3*          'X' /
 'HORIZ'    3  3  3  3   'OPEN'   0     -1     0.4   3*          'X' /
 'HORIZ'    4  3  3  3   'OPEN'   0     -1     0.4   3*          'X' /
 'HORIZ'    5  3  3  3   'OPEN'   0     -1     0.4   3*          'X' /
/
```

Defining local grid block sizes

Most GRID section keywords may be inserted between CARFIN, RADFIN or RADFIN4 and ENDFIN to alter refined grid block properties from those of the host cells. Alternatively, grid data for local grids may be input using the REFINE keyword if the local grid has previously been defined using CARFIN, RADFIN or RADFIN4.

If DX, DY or DZ are specified for the local grid then ECLIPSE checks for consistency with the dimensions of the global host cells. Local grid block sizes can also be defined in terms of fractions of the host grid using the following keywords, which are especially useful if the global grid has been defined in corner geometry:

NXFIN, NYFIN, NZFIN Define number of local cells in each host cell

HXFIN, HYFIN, HZFIN Define ratios for splitting the host cell into local blocks.

For example, to define a local grid of 5 layers in a host of 2 layers:

```
CARFIN
 'LGR2'  3  5  1  2  4  5  6  4  5 /
NZFIN
 3   2  /
HZFIN
 1.0  2.0  3.0
 2.0  1.0  /
```

This gives 3 local layers in the top host layer, with DZ values in the ratio 1:2:3 and 2 local layers in the bottom host layer, with DZ values in the ratio 2:1. For example, if the host layer thicknesses are 60 ft. and 30 ft., local DZ values are 10 ft., 20 ft., 30 ft., 20 ft., 10 ft.

This option could be used to set up a logarithmic refinement, for example for a Cartesian refinement:

```
HXFIN
 1.0 10.0 100.0 /
```

Note that the numbers used in the HXFIN, HYFIN, HZFIN data could correspond to actual DX, DY, DZ values, but these need not fit exactly in the host cell, as they are adjusted automatically. If no ratios are given, the default is to split the host cell in equal proportions. Data may be defaulted for some or all host cells, but data must be complete if given for a particular host cell. For example, to divide a host grid of 3 layers into 7 local cells, with splitting ratios defined for the middle layer:

```
NZFIN
 3 2 2 /
HZFIN
 3* 1.0 2.0 2* /
```

The keywords HZFIN and NZFIN can also be used in radial refinements declared by RADFIN or RADFIN4, to define the vertical splitting. INRAD should always be specified for radial refinements. If OUTRAD is specified after RADFIN or RADFIN4 then its value applies to the inner radius of the outermost cell of the local grid, because the outermost cell assumes the odd shape required to fit into the Cartesian host cell. If DR or DRV values are specified after RADFIN or RADFIN4 then the sum of INRAD and the DR values must be less than the DX and DY values

of the host cells so that the refinement fits within the host cell dimensions. An incomplete set of DR or DRV values may be provided for a local grid and any missing data for the outer rings is completed automatically as a geometric series (ECLIPSE 100), or logarithmic series (ECLIPSE 300) to fit within the host block. Rather than supplying the DR values directly, you can instead define relative sizes for the DR values using the HRFIN keyword:

```
RADFIN
--NAME  I1 I2 J1 J2 K1 K2 NR NT NZ
 'LGR3'   6  6  4  4   1  5  5  4  20 /
INRAD
 0.25  /
OUTRAD
 3000.0 /
HRFIN
 1.0 2.0 3.0 4.0 /
```

This gives $NR-1$ linearly increasing DR values scaled such that their sum is equal to OUTRAD. HRFIN could be useful when the default logarithmic distribution creates tiny cells near the well bore, which can lead to convergence problems.

If the global grid is corner-point, then the ZCORN and COORD keywords may be used between the CARFIN /ENDFIN and RADFIN/ENDFIN pairs etc. to provide finer control over the local grid geometry. If multiple reservoirs are used (see RUNSPEC section keyword NUMRES) then local grids cannot straddle two or more reservoirs; only one reservoir is allowed within each local grid. In order to model a local grid straddling two reservoirs, you should specify several adjacent constituent local grids stacked vertically which can then be amalgamated using the AMALGAM keyword (see below). If the local grid geometry is input in this way, the local corner points should be consistent with those of the host cells. ECLIPSE issues an error message if this is not so.

Defining local grid rock properties

By default, values of porosity, permeability, transmissibility multipliers, etc., are copied to local grid cells from the global host cells. In most cases this is adequate, but it is possible to define properties directly for some or all of the cells within a local grid by specifying grid arrays between the CARFIN/ENDFIN, RADFIN/ENDFIN, RADFIN4/ ENDFIN or REFINE/ENDFIN pairs. When the end of the GRID section is reached, any values that have not been explicitly defined for local cells are then copied from the global host cells. For example:

```
CARFIN
-- NAME     I1-I2 J1-J2 K1-K2 NX NY NZ NWMAX
   'NORTH'  5 10   5 10   1 4 18 18  8   5  /
EQUALS
 'PORO'  0.3  1 18 1 18 1 4 /
 'PERMZ' 50.0 1 18 1 18 1 4 /
 'NTG'   0.9  1 18 1 18 1 8 /
/
MULTIPLY
 'PORO'  0.5  6 13 6 13 1 2 /
 'NTG'   0.2  6 13 6 13 1 2 /
/
ENDFIN
```

This defines the porosities only in the top four layers on the local grid, and these are subsequently modified for the top two layers with the MULTIPLY keyword. The porosities for cells in the bottom four layers are copied from their host cells in the global grid.

Since by default ECLIPSE does not copy any properties from host cells until the end of the GRID section is reached, it is not permitted to modify any property for cells within the local grid (by using the ADD, COPY, MULTIPLY or OPERATE keywords) without previously defining the property for those cells.

This behavior can be changed by specifying the LGRCOPY keyword in the RUNSPEC, GRID, EDIT, PROPS or REGIONS section, depending on the simulator used. This will cause global values to be copied 'just in time' to the corresponding local grid cells. There are limitations on the use of this associated with the order of the keywords in the data file - for instance, global values of DX must have been set before the LGR operations are carried out in order for the correct global parent cell to be allocated to each LGR cell. If DX is redefined for the global cells after an operation requiring the LGRCOPY functionality to be used has been invoked, this will trigger an error. Similar limitations apply to all keywords which may affect which global cell a particular LGR cell is physically located in.

Amalgamating cartesian local grids (ECLIPSE 100)

ECLIPSE 300 allows local grids to be placed anywhere in the reservoir provided they do not overlap. In ECLIPSE 100 a local grid cannot be placed next to another local grid.

It is however possible to amalgamate two or more Cartesian local grids into a single entity, by using the keyword AMALGAM. This facility provides a means of completing a well in more than one local grid. Thus the grid may be refined more efficiently around a deviated well by employing a number of smaller refinements in a zig-zag pattern instead of a single large refinement.

Local grids within the same amalgamation are allowed to touch, but must not overlap. Several geometrical configurations allows amalgamation. Using the notation of the drawing below we have:

- Z-direction: If $Z_{1B} = Z_{2A} + 1$ (A above B) then an amalgamation is defined if $X_{1B} \le X_{2A}$ and $X_{1A} \le X_{2B}$ and $Y_{1B} \le Y_{2A}$ and $Y_{1A} \le Y_{2B}$. This case corresponds to the drawing below. Swap indices A and B for the case B above A.
- X-direction (B on the right side of A): $X_{1B} = X_{2A} + 1$ then an amalgamation is defined if $Y_{1B} \le Y_{2A}$ and $Y_{1A} \le Y_{2B}$ in the case where NNCs and NEWTR are set. If no NNCs or NEWTR is set, then the following condition on the Z-coordinates is used: $Z_{1B} \le Z_{2A}$ and $Z_{1A} \le Z_{2B}$. For B on the left of A swap indices A and B.
- Y-direction: same as X-direction.

Note that this test is done for all pairs of active LGRs where a Flux Boundary (USEFLUX) is not used. This includes active and inactive cells.

Any valid Cartesian grids may be amalgamated, even if they do not touch. ECLIPSE treats an amalgamation as a single entity by lumping together all its constituent local grids, with non-neighbor connections between any adjacent grids in the amalgamation. The amalgamation is handled in the same way that ECLIPSE treats a single local grid; thus the time step is synchronized across all the local grids within an amalgamation. Use of this facility to mimic nested refinements is not encouraged because this leads to a large number of non-neighbor connections between local grids. A well may be completed in any local grid within the same amalgamation, but may not be completed outside its amalgamation. Wells within local grid amalgamations should be specified in the usual way with keyword WELSPECL, naming any of the constituent local grids in item 3. The keyword COMPDATL should be used instead of COMPDAT to specify connection data for wells completed in local grids.

For a well in an amalgamation, keyword COMPLMPL should be used instead of COMPLUMP to lump connections for automatic workovers. Similarly, WELOPENL should be used instead of WELOPEN to open or shut wells or well connections; and WFRICTNL must be used instead of WFRICTN for the ECLIPSE 100 Wellbore Friction option. The '-L' versions of the keywords each contain an extra data item to specify which local grid the following connection belongs to. Either keyword from each pair may be used for wells completed in a single local grid.

For example, consider defining a deviated well in neighboring local grids 'WEST' and 'EAST'. The GRID section might look like:

```
CARFIN
-- NAME    I1-I2  J1-J2  K1-K2 NX NY NZ NWMAX--
  'WEST'    3 4    2 3    1 2   8  4  4  1  /
CARFIN
-- NAME    I1-I2  J1-J2  K1-K2 NX NY NZ NWMAX--
  'EAST'    4 5    4 5    1 2   8  4  4  1  /
AMALGAM
  'EAST' 'WEST' /
/
```

This generates a composite refinement which looks like Figure 39.6 (in the areal direction).

Figure 39.6 A composite refinement

A well may be specified in the SCHEDULE section in the following way:

```
WELSPECL
--   WELL  --  GROUP -- LOCAL GRID -- LOCATION --  BHP  -- PREF
--   NAME  --  NAME  --    NAME    --   I  J   -- DEPTH -- PHASE
    'DEV'       'G'       'EAST'        2  2      8000     'OIL'  /
/
COMPDATM
-- WELL - LOCAL -   LOCATION   - OPEN - SAT - CONN - WELL - KH  - S - D - PEN -
-- NAME - GRID  - I  J  K1  K2 - SHUT - TAB - FACT - DIAM              - DRN -
  'DEV'   'WEST'  6  3  3  3  'OPEN'   0     -1    0.3    3*            'X' /
  'DEV'   'WEST'  6  4  2  2  'OPEN'   0     -1    0.3    3*            'X' /
  'DEV'   'EAST'  2  1  2  2  'OPEN'   0     -1    0.3    3*            'X' /
  'DEV'   'EAST'  2  2  1  1  'OPEN'   0     -1    0.3    3*            'X' /
/
```

Editing local grids

It is possible to modify data for local grids in the EDIT section by inserting modifications between a REFINE/ENDFIN pair, just as for the global grid, in order to modify pore volume, depth and transmissibility data. This is not recommended for general use and should be used with care. For example:

```
REFINE
 'NORTH'  /
EQUALS
 'PORV'  1000.0  1 18 1 18 1 8 /
 'TRANZ'    0.0  1 18 1 18 4 4 /
/
ENDFIN
```

edits PORV and TRANZ within the local grid NORTH.

If the pore volume is edited for any cells in the global grid that have been refined, then ECLIPSE carries over the modification to local cells by applying an additional pore volume multiplier to the local grid, which is defined as the ratio of edited to unedited pore volumes for the global host cells. Similarly, when transmissibilities are edited on the global grid, ECLIPSE imposes an additional transmissibility multiplier to the local grids which is determined in the same manner. This additional transmissibility multiplier applies to regular as well as to non-neighbor connections within the local grids, and also to connections between the local and global grids.

Grid reports and tuning

Just as WELSPECL and COMPDATL (or COMPDATM) replace WELSPECS and COMPDAT for refined wells, so RPTGRIDL replaces RPTGRID in the GRID data section to produce grid reports for all local grids. RPTGRID may also be used in the normal way to produce global grid reports.

ECLIPSE 100

ECLIPSE 100 allows the local grid refinements to be run with a completely different time-stepping scheme from the global grid. If separate tuning parameters are required for local grids, use TUNINGL and/or TUNINGS in the SCHEDULE section. TUNINGL supplies a common set of tuning parameters for all local grids. In addition, specific sets of tuning parameters can be specified for individual local grids with the TUNINGS keyword. The TUNING keyword may be used in the usual way for the global system. NEXTSTPL can be used in place of TUNINGL to restrict the length of the next time step in all local grids.

Timing reports

ECLIPSE 100

Extra timing mnemonics have been added to the timing reports. These reports are initiated by using mnemonic CPU (12th control) in RPTSCHED.

CPSU	Calculation of the local component of the global matrix.
TLGR	Solving the local grid systems.

Vertical equilibrium (ECLIPSE 100)

Vertical Equilibrium may be used for the global grid system, with either dispersed flow or VE in the local grid systems. The Vertical Equilibrium model is activated in a local grid system by using the VEFIN keyword. The relative fraction of the VE and the rock relative permeability curves can be modified on a local cell by cell basis using the REFINE and VEFRACV keywords in the PROPS section.

The global host cells are described by tables of distinct depths versus volumes, that correspond to the distinct corner depths in the local grid. This accurate description of the global cell prevents inconsistencies between the global and local systems that could lead to jumps in the global contact depths. However, if the number of local cells in a global host cell is large, the cost of setting up a large table of distinct depths can be high. To keep the cost down, you can specify a smaller number of distinct depths for the table. The number of distinct depths in the table for each global host cell can be controlled by setting NVEPT in the VEFIN keyword. Defaulting this item causes the original description of the global host cells to be used (with up to 8 distinct depths in the table).

The compressed VE option may be used to run a global 3D model as a 2D VE areal run, with dispersed flow in the local 3D grids. To do this, use the keyword VE in the RUNSPEC section.

Within an LGR amalgamation, all the constituent local grids should employ the same physical model (dispersed flow or VE).

Dual, multiple and triple porosity

In dual, multiple and triple porosity runs, the global coordinates specified in CARFIN and RADFIN refer to the (outer, in the case of multi-porosity) matrix blocks only. For example, in dual porosity, in an 8*8*6 global grid, to refine the matrix cell at (8, 8, 2) together with its corresponding fracture cell at (8, 8, 5) use;

```
CARFIN
--          I1 I2 J1 J2 K1 K2 NX NY NZ NW
 'CARF882'  8  8  8  8  2  2  5  5  10 3 /
ENDFIN
```

The refinement has 5*5*5 matrix cells and 5*5*5 fracture cells. For multiple and triple porosity, the same principle applies, namely that the global grid z- or k-extents (K1 and K2, data items 6 and 7) are always defined for the matrix grid only, but the size of the local grid in the z- or k-direction (NZ, data item 10) covers all porosities. To find its size in one porosity only (e.g. in the matrix) it must be divided by NPOROS, which is NMTRIX+1, where NMTRIX is the number of matrix porosities (for multiporosity, it is data item 1 of keyword NMATRIX, or it is 1 for dual porosity), or the number of matrix and vug porosities (for triple porosity, it is data item 1 of keyword TRPLPORO). Conversely, NZ must be divisible exactly by NPOROS.

ECLIPSE 100

If the keyword DPNUM has been used to specify regions of the reservoir for which a conventional single porosity model is to be used in a dual porosity run, then the local grid may not straddle different DPNUM regions. A local grid in a dual porosity run must either lie entirely within a dual porosity region or entirely within a single porosity region. Within an LGR amalgamation, all the constituent local grids must be in the same DPNUM region.

Unstructured Local Grid Refinements

Unstructured (PEBI) grids can now be run as local grid refinements. The EXTFIN keyword can be used to specify an unstructured local grid refinement. The EXTHOST keyword can be used to specify the global host cells for an unstructured local grid refinement, or similarly, the EXTREPGL keyword can be used to specify the replaced global cells for an unstructured local grid refinement. Finally, the TRANGL keyword can be used to specify the global-local connections and associated transmissibilities for an unstructured local grid refinement.

Between any EXTFIN/ENDFIN bracket, either EXTHOST or EXTREPGL should be used, but not both. The four new keywords mentioned above are generally supplied by another program and are not modified by the user.

Region data and end point scaling

By default, cells in the local grid systems are assigned REGIONS section data values (SATNUM, PVTNUM etc.) and end-point scaling data (if the option is active) from their corresponding global host cells. In many cases this will prove adequate.

However it is possible to modify selected data quantities from the REGIONS and PROPS sections within local grids, using the REFINE keyword. This keyword operates in exactly the same way as the CARFIN and RADFIN keywords in the GRID section, and can be applied in either the REGIONS or PROPS sections. Any data following a REFINE keyword is applied to the named local grid refinement until either another REFINE keyword or an ENDFIN keyword is encountered. Local grid quantities that can be set in this way include end point scaling data and Vertical Equilibrium option data. An example that sets SATNUM values in a local grid containing 250 cells is:

```
REFINE
 'CARF882' /
SATNUM
 125*1
 125*2 /
ENDFIN
```

Switching refinements on and off (ECLIPSE 100)

Refinements may be switched on or off at any time during a simulation using keywords LGRON and LGROFF in the SCHEDULE data section. For example,

```
LGROFF
 'SOUTH' /
```

switches off the refinement called SOUTH, while

```
LGRON
 'NORTH' /
```

switches on the refinement called NORTH.

Refinements can be made to switch on and off automatically depending on the number of active (that is, open or stopped, but not shut) wells they contain. For example, using either

```
LGROFF
--NAME     NWELLS
  'SOUTH'  2      /
```

or

```
LGRON
--NAME     NWELLS
  'SOUTH'  2      /
```

causes the refinement called SOUTH to be on whenever it contains two or more active wells, and off whenever it contains less than two.

If LGROFF and LGRON are not used then all refinements defined in the GRID data section are activated throughout the entire simulation.

Switching refinements on part way through a run is not advised, in general, since there is no valid procedure for allocating material to refined cells except at initial conditions when equilibration is used. If a refinement is switched on during a simulation, material is distributed uniformly to the refined cells within each host cell.

It is possible to add extra local grids when restarting a run. This is analogous to switching on an LGR part way through a run, but allows an LGR to be added without re-running the base case. The new LGRs should be added to the GRID section after all the existing LGRs have been specified. The new LGRs are turned on at the next time step, unless instructed otherwise with the LGROFF keyword. This facility allows you to add new LGRs into the grid where new wells are about to be opened. It is generally not permitted to add a new LGR around a well that already exists. If this is required, however, the existing well and all its connections must be shut before the first TIME, TSTEP or DATES keyword is read in the restart run. In order to continue to model the well, a new counterpart well should be defined in the local grid (with a different name) using WELSPECL and COMPDATL or COMPDATM, which has all the attributes of the existing shut well. ECLIPSE then ignores the existing global well, and an error condition results if the well is ever to be reopened. This procedure is not recommended in general.

ECLIPSE 300

Flexible restarts can be used in ECLIPSE 300 to add LGRs on a restart run, given that no LGRs existed in the base run. The new LGRs should be added to the GRID section (as normal). No other keywords are required. The new LGRs are turned on at the next time step. It is important to note that when adding an LGR using a restart, refinement around an existing well is not possible. If this is the case, an error message is output.

ECLIPSE 300

Flexible restarts can also be used in ECLIPSE 300 to turn LGRs off in a restart run. Any LGRs found in the RESTART file of the base run is ignored by ECLIPSE 300. See also the GETGLOB and GETSOL keywords.

Summary files

In LGR runs, two sets of summary data are output. The usual SUMMARY files contain summary data produced at the global time-steps. In addition SUMMARY file quantities are output at the local time-steps; this local data is output to a separate LGR file. The RPTONLY keyword to limit the frequency of SUMMARY file output does not apply to the LGR file.

To save space, the creation of LGR files can be prevented by using the NOSUMLGR keyword in the SUMMARY section. When running lgr-in-place (see "time steps" on page 579), there is little point in dumping SUMMARY files for the local time-steps since the global and local time-steps are all synchronized.

The local time-step summary data is stored in a file with the following name:

ROOT.FLGR (or ROOT.FLG)	Formatted
ROOT.LGR	Unformatted

See the SUMMARY section overview for information on how to write local time-step data to this file.

Transmissibility calculations

For radial local grid refinements, the standard method based on pressure equivalent radii is used for flow between local radial blocks. Flow between the outer local block and the adjoining host block is treated as linear, but with the ‘transmissibility center' taken at the pressure equivalent radius of the local block, rather than the geometric center.

For Cartesian refinements, transmissibilities between the local and global grid are calculated from the usual formulae (described in "Transmissibility Calculations" on page 1103) when the NEWTRAN method is used. However, when OLDTRAN or OLDTRANR is used for block-centered grids, then the representative area chosen for the transmissibility between local and global cells is the face area of the local grid cells. For example, for the transmissibility in the X-direction between a local cell l and a global cell g, when OLDTRAN is used

$$\mathrm{TRANN}_{lg} = \frac{\mathrm{CDARCY} \cdot \mathrm{TMLTX}_l \cdot \mathrm{DIPC}}{B} \qquad \text{[EQ 39.1]}$$

where

CDARCY	is Darcy's constant in the appropriate units
TMLTX_l	is the X+ transmissibility multiplier for global cell g
DIPC	is the dip correction

and

$$B = \frac{1}{2A_l} \cdot \left(\frac{DX_g}{PERMX_g} + \frac{DX_l}{PERMX_l} \right) \quad \text{[EQ 39.2]}$$

The area is given by

$$A_l = DY_l \cdot DZ_l \cdot NTG_l \quad \text{[EQ 39.3]}$$

The dip correction is given by

$$DIPC = \frac{DHS}{DHS + DVS} \quad \text{[EQ 39.4]}$$

with

$$DHS = \left(\frac{DX_l + DX_g}{2} \right)^2 \quad \text{[EQ 39.5]}$$

and

$$DVS = (DEPTH_l - DEPTH_g)^2 \quad \text{[EQ 39.6]}$$

Note The convention used for transmissibility multipliers: For connections in the X-, Y-, Z- directions, the global value for the adjoining block is used. For connections in the X+, Y+, Z+ directions, the local grid block value is used.

Transmissibilities between local cells in adjacent amalgamated local grids are calculated from the usual formulae when the NEWTRAN method is used. When OLDTRAN or OLDTRANR is used, connections are possible between local cells on faces of adjacent refined cells in the global grid. The interface areas used for the calculation account for partial overlap. These overlap areas are obtained by projecting the refinements on the faces of adjacent refined cells in the global grid onto the unit square; then the proportion of cell areas in each overlap is applied as a multiplier to the actual local face areas in the OLDTRAN formula.

A report on transmissibilities between the local and global grids and between any amalgamated local grids can be obtained by using the mnemonic ALLNNC (switch 24) in RPTGRIDL.

Treatment of transmissibility multipliers in local grids

It is possible to enter transmissibility multipliers MULTX, MULTY, MULTZ etc. for local grids between the CARFIN/ENDFIN, RADFIN/ENDFIN, RADFIN4/ENDFIN and REFINE/ENDFIN pairs in the GRID section. If any of these multipliers are not defined for the local grid, they are inherited from the host cells of the global grid according to the following rule: The multiplier on the transmissibility between two adjacent local grid cells is set to the global multiplier in the appropriate direction at a cell boundary of the global grid, and is set to 1.0 otherwise. When the local MULTR and MULTTHT are undefined within a radial local grid, these are inherited from the appropriate global MULTX and MULTY respectively.

time steps

Local time-stepping (ECLIPSE 100)

Efficiency is achieved by solving each local grid (or amalgamation of local grids) individually using “local time-stepping”. This allows the local grids to have shorter time steps than the global grid, in order to resolve coning effects etc., without holding up the progress of the global grid simulation. Each local grid may require several time steps to reach the end of a global time-step. This is very efficient in cases where the local grids naturally require much shorter time steps than the global grid.

Local time-stepping has some disadvantages, however. The global grid and each of the local grids may individually be solved fully implicitly, but since both the global and local grid solutions provide boundary conditions for each other, a semi-explicit coupling is required. Thus the coupled solution is not fully implicit and may even exhibit instabilities in some cases. Also, local time-stepping can cause group flow rates to drift significantly away from their group control targets. The approach of local time-stepping is the default method for solving local grids and was the only method available in versions of ECLIPSE 100 prior to 97A.

When local time-stepping is used, the sequence for calculating each global time-step is as follows:

1. The global model is advanced through a new time step in the usual way. At this stage the local grids are represented by their host cells in the global grid. Wells in refined grids are solved using the grid block properties of the refined system. Properties of host cells are computed using local grid information.
2. Each local simulation is then advanced through a series of local time-steps that are often smaller than the global time-step. The local time-steps are chosen to synchronize with the end of each global time-step.
3. Flows between the global and local systems are computed dynamically as boundary conditions to each local simulation using currently updated variables in the local grid and fixed variables in the global grid (linearly interpolated in time over the global time-step). Spatial interpolation of the pressure field may also be performed for Cartesian refinements to enhance accuracy.
4. The process is completed by updating well rates and material in the host cells of the global model to be consistent with the local models. The global model is then ready to advance to the next global time-step.
5. After the completion of a global time-step the material for each phase in each host cell is compared with the material sum for the corresponding local cells. Subsequent global time-steps are often selected to keep these material calculations within 2.5%. The convergence report summary uses the mnemonic LGRC followed by the name of the constraining LGR when this criterion is used. For example:

```
STEP  16 TIME=  116.32  DAYS (  +3.5  DAYS LGRC/LGRWEST   4 ITS)
PAV=  3185.2
```

   This time-step selection criterion may be altered using keyword TUNING.

6. The first local grid specified within an LGR amalgamation is chosen to be the representative of the amalgamation. The convergence reports for each amalgamation are identified with this name.

In-place LGRs

In ECLIPSE 100, an alternative fully implicit solution method, “LGR in-place”, is available to overcome the shortcomings of local time-stepping. The global and local grids are solved simultaneously and fully implicitly as a coupled system. This avoids any problems with stability or meeting group control targets associated with local time-stepping. However, the coupled system chooses a time step that is the minimum of the natural time-step over the global and all local grids. In cases where one LGR requires a time step much smaller than the rest of the grid (say for a horizontal well in a Vertical Equilibrium model, or for a radial LGR), running the coupled system at this time step may severely limit the efficiency of the simulation. Thus it is possible to nominate local grids individually to be solved in-place (keyword `LGRLOCK`). Any local grids not nominated as in-place are advanced using local time-stepping. If all local grids are nominated as in-place, then a fully coupled implicit solution is obtained. This procedure is commonly adopted in ECLIPSE 300.

ECLIPSE 300

ECLIPSE 300 solves LGRs using the same time step as the global grid. There is no local time-stepping option.

Restrictions on the use of local grids

ECLIPSE 100 only

1. Separate local grid refinements must not be placed in adjacent cells of the global grid, unless the refinements belong to the same amalgamation.
2. Partially overlapping LGRs are not allowed.
3. Keyword `NINENUM` is not compatible with the Local Grid Refinement option.

ECLIPSE 100

4. It is possible to define a refinement within another local grid (nested refinements), provided the parent grid is a cartesian local grid. Nested refinements can only be solved “in place” and do not work with the parallel option. They are also not allowed if there are amalgamated local grids in the data set or with the vertical equilibrium option.
5. It is not currently possible to coarsen within a local grid.

ECLIPSE 100 only

6. Radial local grids may not be amalgamated. Radial LGRs need to be separated by a minimum of non-LGR containing global cell.

ECLIPSE 300 only

7. Adjoining radial LGRs are allowed.
8. Each local cell must be identified with a single global cell. It cannot be split between two global cells. The number of local cells in the X, Y or Z direction must be an integral multiplier of the global grid dimensions, unless sufficient local data are provided to allocate local cells to host cells according to size (for example: `DX`, `COORD`) or to define how the host cells are split up (for example: `NZFIN`, `HZFIN`).
9. Local grids may not straddle different reservoirs (if the number of reservoirs is set > 1 in `RUNSPEC` keyword `NUMRES`). To achieve this, separate local grids can be stacked vertically, one in each reservoir, and then amalgamated using the `AMALGAM` keyword.
10. Local grids may not straddle different flux regions (specified by keyword `FLUXNUM` for the ECLIPSE Flux Boundary facility). In dual porosity runs, local grids may not straddle the boundary between single porosity and dual porosity regions (specified using keyword `DPNUM`).
11. Numerical aquifers cannot be attached to local grids.

12 The diffusion options (Molecular Diffusion and Environmental Tracers) are not available in local grid runs.

13 The NNC or EDITNNC keywords for the global grid should not refer to non-neighbor connections connecting refined cells in the global grid.

14 It is not possible to use the grid coarsening option when restarting a run.

Treatment of wells in local grids

Wells are placed in local grids by using keywords WELSPECL and COMPDATL instead of WELSPECS and COMPDAT. The I, J and K locations of the wellhead and connections refer to the cell coordinates within the well's refined grid system. In ECLIPSE 100 a well cannot be completed in any other cells outside its local refined grid or outside its amalgamation.

ECLIPSE 100

In other keywords relating to well connections, apart from COMPLUMP, COMPLMPL, WELOPEN, WELOPENL, WPIMULT, WPIMULTL, COMPRPL, COMPVEL, WFRICTN and WFRICTNL, the coordinates of the host cells in the global grid must be used. The data specified for each connection in the global grid will be applied to all the well's connections in the refined grid cells contained within the global grid connection cell.

ECLIPSE 300

In contrast to the above, ECLIPSE 300 only permits specialized LGR keywords (ending in 'L') to be used when well connections need to be specified. The exception to this is when all the connection values are defaulted and the action of the keyword applies to all the well connections.

For wells in local grids, keywords COMPLMPL, WELOPENL, WPIMULTL and WFRICTNL should be used respectively to lump connections, open and close connections, apply PI multipliers to connections, and define friction wells. (In fact, the keywords COMPLUMP, WELOPEN, WPIMULT and WFRICTN may also be used for wells in single local grids, but they cannot be used for wells in amalgamated local grids.) In runs employing the Vertical Equilibrium option, well connection depth data should be set, if required, using keyword COMPVEL rather than COMPVE. In each of these keywords, I, J and K coordinates within the respective local grid systems should be used to identify the connections.

Group control is applied to wells in the usual way when the global grid is advanced through each global time-step. When using local time-stepping, the target rates allocated to the wells under group control are then held fixed while each local grid system is advanced to the end of the global time step. If the flow rate of any local grid well changes over the length of the global time-step, the target rates of the other wells cannot be adjusted to accommodate the change. Thus the group flow target is not met exactly. The group flow discrepancy may be significant if a local grid well changes its flow rate significantly over the length of the global time step. If this occurs, possible remedies are to limit the size of the global time-steps, use Coarsening instead of Local Grid Refinement, or run LGR in-place for those local grids containing wells that contribute to the group flow discrepancy.

ECLIPSE 100

When local time-stepping is employed, economic limits applied to local wells and their connections (WECON and CECON) are checked at the end of each local time-step in the local grid system, and workovers etc. are performed as required. Group economic limits (GECON) and rate limits involving workovers (GCONPROD) are checked at the end of each global time-step. These may result in more workovers to local wells.

In automatic workovers, connections are closed one at a time. This may be on too fine a scale for a well in a refined grid. In particular, a well in a 3-D radial refinement could have one of its connections on a given layer closed while the other connections on that layer remain open - not a realistic situation. It is therefore advisable to lump the connections together into sets (which we call "completions"), which are the smallest unit that can be closed in an automatic workover.

When a workover limit is exceeded, the program searches for the worst-offending completion, and close all its connections together. Keyword COMPLMPL may be used to lump the connections into completions. For example, consider the well 'PRODUCER' that is completed in layers 9 - 11 in the 3-D radial refined grid. To lump the four connections on each layer into one completion (making three completions in total), use:

```
COMPLMPL
-- WELL ---- LOCAL GRID ---- LOCATION ---- COMPLETION
-- NAME ----    NAME    --- I  J  K1 K2 -- NUMBER
 'PRODUCER'      1*         2*    9  9       1      /
 'PRODUCER'      1*         2*   10 10       2      /
 'PRODUCER'      1*         2*   11 11       3      /
/
```

Current restrictions with local grid wells

ECLIPSE 100

1 The keyword GCONTOL cannot be used in local grid refinement runs.

ECLIPSE 100

2 Keyword WLIMTOL only works for economic limit checks on wells in active local grids. It is ignored in all other limit violation checks.

ECLIPSE 100

3 In parallel runs, LGRs cannot be switched on (using the LGRON keyword) after they have been switched off.

ECLIPSE 100

4 In parallel runs, wells cannot be declared inside of LGRs (using the WELSPECL keyword) after LGRs have been switched off.

Other problems with local grid wells

1 When a well is under rate control, its potential flow rate in a local grid system can be much larger than it would be in the un-refined global grid. This is because the connecting grid blocks, whose pressures are used in the potential calculation, are much smaller in the local grid system. In calculating the potential, the assumption is made that the pressures in the connecting grid blocks does not change if the well flow rate increases from its current value to its potential value. While this may be a reasonable assumption in the global grid, it is not true for highly refined local grids. The smaller the connecting grid blocks are, the more optimistic will be the potential.

ECLIPSE 100

2 When group control targets are applied, there may be significant discrepancies between the actual group flow rate and the imposed rate target when local time-stepping is employed. This is because group control is applied while solving the global grid, but not while solving the local grids. Group control is applied to wells in the usual way while solving the global grid. The target rates allocated to the wells under group control are then held fixed while each local grid system is advanced to the end of the global time step. If the flow rate of any local grid well changes over the length of the global time-step, the target rates of the other wells cannot be adjusted to accommodate the change. Thus the group flow target is not met exactly. The group flow discrepancy may be significant if a local grid well changes its flow rate significantly over the length of the global time step. If this occurs, possible remedies are to limit the size of the global time-steps, use Coarsening instead of Local Grid Refinement, or run all those local grids containing wells in the offending group(s) as in-place LGRs using keyword LGRLOCK.

Parallel load balancing with LGRs

The strategy used to distribute local grids across processors is fundamentally different between ECLIPSE 100 and ECLIPSE 300. In ECLIPSE 300 refined cells are placed on the same processor as the corresponding host cell. An LGR could therefore be distributed across any number of processors. ECLIPSE 100 assigns complete LGRs to processes without regard to the ownership of the host cell. Host and fringe cell information is swapped between processors as required. ECLIPSE 100 instead attempts to balance the number of refined cells on each processor.

Geometry and grid data in LGRs

Transmissibility data

In general, if grid data are not specified for a local refinement (by inserting keywords between CARFIN, RADFIN or RADFIN4 and ENDFIN) then they are derived from the properties of the host cells. You should note the following conventions used for default local permeability and transmissibility data.

Permeabilities

Unless permeabilities are explicitly defined for a local radial grid, global PERMX data are copied to PERMR and global PERMY data are copied to PERMTHT. If the PERMX and PERMY values are different, a warning is issued by ECLIPSE.

Transmissibility multipliers

Transmissibility multipliers (MULTX, MULTY, MULTZ) are applied to the X+, Y+ and Z+ faces of each grid block. For a local refinement, multipliers can be copied from the host grid only if the positive face of the local cell coincides with a host cell face; otherwise, a default value of 1.0 is used. If this is inappropriate, transmissibility multipliers should be explicitly defined for the local grid.

Note For connections between local and global cells, the transmissibility multipliers for the adjoining global block are used for flows in the X-, Y- or Z- directions. The local grid block multipliers are used for flows in the X+, Y+ or Z+ directions.

Pore volumes

The pore volume of a refined global cell may differ from the sum of the pore volumes of the local cells which it contains, either because the local porosities and net-to-gross ratios differ from the values for the host cell or because of discrepancies in geometry.

To avoid significant volumetric errors for radial grids, the geometric volumes of the outer refined blocks are adjusted to fit the host cell. However, local porosities and NTG values are not modified, so differences in global and local pore volumes may still occur.

After computing the local pore volumes, ECLIPSE replaces the pore volume of the host cell with the sum of the refined pore volumes. If a difference of more than 1% is detected, a warning message is issued. Pore volume reports are produced after any modifications have been made.

Active cells

The local cells identification, specified using the ACTNUM keyword, is inherited from the parent global cell. This property can be overwritten by inserting the ACTNUM keyword between CARFIN, RADFIN, or RADFIN4 and ENDFIN only if the value for the global cell is 1.

Radial refinements

1 A local radial grid may be defined in either a single column of cells (using RADFIN), or in a box consisting of four columns of cells (using RADFIN4). The geometry of the local grid may be defined using one of the following methods:

   a Refinements in the R-direction may be defined by user-given DR values. In the Z-direction, refinements can be specified by NZFIN / HZFIN data. Missing data are calculated automatically, so that a radial grid can be defined by a minimum set of one RADFIN or RADFIN4 keyword and one INRAD keyword.

   b In a block-centered host, DZ values may be input directly, with radii defined as for the case (a). Use of user-defined DTHETA values is not recommended, as the local angles are usually restricted by the host cell geometry.

   c In a corner-point host, a local radial grid may be defined by corner-point coordinates. Note that the local COORD data are expected in cylindrical coordinates, relative to an origin at the center of the refinement. (Note: Cartesian local COORD data are defined relative to the global grid origin.)

2 A local radial refinement defined through RADFIN can only be inserted in a single column of host cells, with NTHETA = 1 or NTHETA = 4. Such a radial grid defined by user-specified corner points (in a corner-point host) must be 3-dimensional (that is, NTHETA = 4).

   A local radial refinement defined through RADFIN4 can only be inserted in a box of four columns of host cells, with NTHETA = 4 or NTHETA = 8. Such a radial grid defined by user-specified corner points (in a corner-point host) must be 3-dimensional (that is, NTHETA = 8).

   A local radial grid cell must fit inside a single host cell.

3 The center of a radial refinement defined through RADFIN is assumed to be at the geometric center of the top host block.

   The center of a radial refinement defined through RADFIN4 is assumed to lie at the central node of the top of the refined box.

4 The sides of the inner radial cells are assumed to be vertical.

5 The innermost radius must be defined by INRAD.

6 The radius for the inside of the outer local block can be defined by OUTRAD. Otherwise it is calculated as RMAX/3.0 where RMAX is the maximum radius of a circle contained within the radial refinement.

7 The user may provide some or all DR values. Any missing radii are completed as a geometric series (ECLIPSE 100), as follows:

   No of radii = NR

   No of radii defined by user = NU

   Radius of inside of outer local block = $\mathrm{RREF} = R_{\mathrm{NR}-1}$

   Last radius defined by user = RU

   Then for $i = \mathrm{NU}+1$ to $\mathrm{NR}-2$

   $R_{i+1} = \mathrm{FAC} \cdot R_i$

   where $\mathrm{FAC} = \left(\frac{\mathrm{RREF}}{\mathrm{RU}}\right)^{1/(\mathrm{NR}-\mathrm{NU}-1)}$

8 For local transmissibility calculations using RADFIN, the outer radius is taken as the area equivalent radius of the host grid block, based on the top area of the host cell. When RADFIN4 is used, the equivalent radius is based on the top area of the four host cells. For volumetrics and connections to the global grid, the sides of the local grid are taken from the coordinate lines for the host.

Note that the equivalent radii are shown in the output reports for the local COORD values.

9 Figure 39.7 shows the numbering convention used for connecting local radial grids to the host when RADFIN is used:

Figure 39.7 The numbering convention used by RADFIN for connecting radial LGRs to host cells

The radial cells are numbered so that Theta increases in an anti-clockwise direction, starting from the quadrant corresponding to the Y- face of the global cell. If necessary, the local coordinates are rotated to align the radial cell with its host.

Reported COORD data will show the corrected Theta values.

ECLIPSE 100

For a local radial refinement defined with RADFIN4, the topology is also arranged with theta increasing anti-clockwise, but starting from the middle of the X+ (I+) face of the refinement as shown in Figure 39.8

Figure 39.8 The numbering conventions used by RADFIN4 for connecting radial LGRs to host cells

In the above case, Theta increases anti-clockwise in (I,J) space, but there is of course no reason why the coordinate system should not be right handed, in which case theta increases clockwise.

10 Transmissibility calculations

For flow between local radial blocks, the standard ECLIPSE method based on pressure equivalent radii is used to compute transmissibilities.

Flow between the outer local block and the adjoining host block is treated as linear, but with the transmissibility center for the local block taken at the pressure equivalent radius. With radii computed as described on page 585, and host cells which are approximately square in cross-section, this method corresponds to the approach described by Pedrosa and Aziz [Ref. 22], and Ewing et al. [Ref. 23].

Alphabetical list of GRID section keywords allowed within a local grid

Refer to the "ECLIPSE Reference Manual" for a further description of each keyword.

Table 39.1 GRID Section keywords

Keyword	Description
ACTNUM	Identifies active grid blocks.
ADD	Adds specified constants to specified arrays in the current box.
ADDZCORN	Adds a constant to the cell corner point depths array ZCORN.
BOX	Re-defines the current box.
COORD	Defines the lines that contain all grid block corner points for each (i, j) and for each reservoir in the grid.
COORDSYS	Specifies completion of the circle in both corner point and block-centered geometries. Allocates grid block layers to reservoirs if there is more than one reservoir in the grid.
COPY	Copies data in the current box from one specified array to another.
COPYBOX	Copies data from one box of cells to another box of the same size.
DIFFMMF	Matrix-fracture diffusivity multipliers for current input box.
DIFFMR	R direction diffusivity multipliers for current input box.
DIFFMTHT	Theta direction diffusivity multipliers for current input box.
DIFFMX	X+ direction diffusivity multipliers for current input box
DIFFMY	Y+ direction diffusivity multipliers for current input box.
DIFFMZ	Z+ direction diffusivity multipliers for current input box.
DR	R direction grid block sizes for the current box.
DRV	Vector of R direction grid block sizes.
DTHETA	Theta direction grid block sizes for the current box.
DTHETAV	Vector of theta direction grid block sizes.
DX	X direction grid block sizes for the current box.
DXV	Vector of X direction grid block sizes.
DY	Y direction grid block sizes for the current box.
DYV	Vector of Y direction grid block sizes.
DZ	Z direction (vertical) grid block sizes for the current box.
DZMTRX	The vertical dimension of a typical block of matrix material, for the entire grid.
DZMTRXV	The vertical dimension of a typical block of matrix material, for the current box.
DZNET	Grid block net thicknesses for the current box.

Table 39.1 GRID Section keywords

Keyword	Description
ENDBOX	Re-defines the current box to encompass the whole grid.
EQLZCORN	Resets part of the corner point depth array.
EQUALS	Sets specified arrays to specified constants in the current box.
HXFIN, HYFIN, HZFIN	Local grid cell size ratios.
HRFIN	Radial grid DR ratios.
IMPORT	Import a binary grid file.
INRAD	Sets inner radius of radial grid.
MAXVALUE	Applies a maximum value to specified arrays within the current box.
MINPV	Sets a minimum pore volume that each cell must have to be active.
MINPVV	Sets a vector of minimum pore volumes that the cells in the current box must have to be active.
MINVALUE	Applies a minimum value to specified arrays within the current box.
MULTIPLY	Multiplies specified arrays by constant values in the current box.
MULTPV	Pore volume multipliers for the current box.
MULTR	R+ direction transmissibility multipliers for the current box.
MULTR-	R- direction transmissibility multipliers for the current box.
MULTTHT	Theta+ direction transmissibility multipliers for the current box.
MULTTHT-	Theta- direction transmissibility multipliers for the current box.
MULTX	X+ direction transmissibility multipliers for the current box.
MULTY	Y+ direction transmissibility multipliers for the current box.
MULTZ	Z+ direction transmissibility multipliers for the current box.
MULTX-	X- direction transmissibility multipliers for the current box.
MULTY-	Y- direction transmissibility multipliers for the current box.
MULTZ-	Z- direction transmissibility multipliers for the current box.
NNC	Explicit entry of non-neighbor connections.
NTG	Grid block net-to-gross ratios for the current box.
NXFIN, NYFIN, NZFIN	Number of local grid cells in each host cell.
PERMR	R direction permeabilities for the current box.
PERMTHT	Theta direction permeabilities for the current box.
PERMX	X direction permeabilities for the current box.
PERMY	Y direction permeabilities for the current box.
PERMZ	Z direction permeabilities for the current box.
PORO	Grid block porosities for the current box.
QMOBIL	Controls the mobile end-point correction for a local grid.
SIGMA	Matrix-to-fracture coupling factor for the entire grid.

Table 39.1 GRID Section keywords

Keyword	Description
SIGMAGD	Matrix-fracture coupling for oil-gas gravity drainage (whole grid).
SIGMAGDV	Matrix-fracture coupling for oil-gas gravity drainage (current box).
SIGMAV	Matrix-to-fracture coupling factor for the current box.
SOLVDIRS	Override principal solver directions within a local grid.
THCGAS	Gas phase thermal conductivities for the current box.
THCOIL	Oil phase thermal conductivities for the current box.
THCONR	Specifies rock and fluid thermal conductivities for the current box.
THCROCK	Rock thermal conductivities for the current box.
THCSOLID	Solid phase thermal conductivities for the current box.
THCWATER	Water phase thermal conductivities for the current box.
TOPS	Depths of top faces of grid blocks for the current box.
VEFIN	Controls the Vertical Equilibrium model within a local grid.
ZCORN	Depths of grid block corners.

Local grid coarsening

Grid Coarsening is the complement of local grid refinement. Cells may be amalgamated to reduce the total number of active cells in the global grid.

ECLIPSE 100

To reserve memory for grid coarsening, in keyword LGR in the RUNSPEC section set MCOARS equal to the number of coarsening cells (amalgamations), for example.

```
LGR
--MAXLGR  MAXCLS  MCOARS
    0       0       48  /
```

Because grid coarsening generates non-neighbor connections, the keyword NONNC must not be specified in the RUNSPEC section.

Coarsenings are specified in the GRID data section using keyword COARSEN. For example, with NDIVIX = 10, NDIVIY = 10 and NDIVIZ = 6 we can use:

```
COARSEN
-- I1 I2 J1 J2 K1 K2 NX NY NZ --
    1 10  1  4  1  6  5  2  3  /
    1  4  5 10  1  6  2  3  3  /
/
```

The first coarsening reduces the number of active cells from 10*4*6 = 240 to 5*2*3 = 30. The eight cells at (1,1,1), (2,1,1), (1,2,1), (2,2,1), (1,1,2) (2,1,2), (1,2,2), (2,2,2) are amalgamated into a single cell for simulation. The second coarsening reduces the number of active cells from 4*6*6 = 144 to 2*3*3 = 18, a reduction in the number of active cells by a factor of 8. Thus the number of amalgamated cells is MCOARS = 30+18 = 48.

NX, NY, and NZ are the number of coarsened cells in each direction within each coarsening set. If values are not provided then default values are set to 1. Specified values of NX must be an exact factor of I2 - I1 + 1, and similarly for NY and NZ.

Coarsening sets must not overlap one another, but they may join together as in the above example. Thus the coarsening facility is extremely flexible, allowing the engineer to build a highly complex grid. Local grid refinements cannot be placed within or adjacent to a grid coarsening, that is the local grids may only be applied to non-coarsened cells.

Implementation

Grid coarsenings are set for the duration of a run and may not be switched on or off at different times during a run. The amalgamations are computed automatically by ECLIPSE which also computes the appropriate non-neighbor connections between different coarsenings and between normal cells and coarsened cells.

Within each amalgamation, a “representative cell” is chosen in the center of the amalgamation. All cells within the amalgamation apart from the representative cell are made inactive. The representative cell is given the properties (dimensions, permeability, pore volume, etc.) of the complete amalgamation.

Note Even if this chosen representative cell is supposed to be deemed inactive (due to the use of the ACTNUM array, or having a zero pore volume, for instance), it is re-activated because ECLIPSE uses it as the storage cell for the upscaled quantities.

If a well is completed in an amalgamation, ECLIPSE moves its connections into the representative cell. Thus, if it was completed in several layers of a multi-layer amalgamation, it ends up with only one connection that is situated in the representative cell. The representative cell of course has the DX, DY and DZ of the complete amalgamation, and the connection transmissibility factor is calculated accordingly.

Furthermore, if a cell connects with an aquifer, then the appropriate connection is set up by moving the connection to the representative cell of the coarsening group involved.

Just as refined cells have the decimal point replaced by R in solution reports (specified using RPTSCHED) so coarsened cells have the decimal point replaced by C. Pore volumes of representative cells may be reported using keyword RPTGRID. Other grid properties such as cell dimensions, porosities, permeabilities, etc. are reported for the original cells before amalgamation.

For a run involving dual porosity, the coarsenings should only be defined for fracture blocks. ECLIPSE copies these coarsenings into the matrix blocks.

Upscaling

Within each coarse block the properties are simply upscaled from fine (f) to coarse (c) in a single coarse cell amalgamation $(I1,I2)\times(J1,J2)\times(K1,K2)$ as follows:

Pore Volume

$$PV_c = \sum_f PV_f$$

Depth

$$D_c = \frac{\sum_f PV_f D_f}{PV_c}$$

DX, DY, DZ

$$DX_c = \frac{\sum_f DX_f}{(J2-J1+1)(K2-K1+1)} \quad ,$$

with similar expressions for DY_c and DZ_C.

Permx, Permy, Permz

$$Kx_c = \frac{\sum_f PV_f Kx_f}{PV_c} \quad .$$

Permy, and Permz are calculated in a similar way.

Tops

$$TOPS_c = \frac{\sum_f PV_f \cdot \overline{TOPS_f}}{PV_c}$$

where $\overline{TOPS}$ is the depth corrected to the top surface of the coarse cell amalgamation.

Tranx, Trany, Tranz

The upscaling for transmissibilities is achieved as a simple average. Between the centers of two neighboring coarse cells, $TRANX_c$ is obtained from a harmonic average of $TRANX_f$ in the X-direction and by summing $TRANX_f$ in the Y- and Z- directions so that

$$TRANX_c = \sum_J \sum_K \left[\frac{1}{\sum_I \left[\frac{1}{TRANX_f} \right]} \right]$$

where the summations in J and K are over the X-direction face between the two cells, and the summation $\sum_I \left(\frac{1}{TRANX_f} \right)$ is over all I-indexed fine cells between the cell centers of the coarse cells.

So for two coarse cells next to each other in the X-direction, say $(IA,IB) \times (J1,J2) \times (K1,K2)$ and $(IB+1,IC) \times (J1,J2) \times (K1,K2)$,

then the summations in the J- and K-directions are $\sum_{J=J1}^{J2} \sum_{K=K1}^{K2} \ldots$, and in the I-direction $\sum_{I=IL}^{IU} \ldots$, where (assuming an odd number of fine cells in each direction in each coarse cell), $IL = (IA+IB)/2$ and $IU = (IB+1+IC)/2-1$.

For one-dimensional single-phase flow governed by Darcy's law, this argument produces a continuous flux across each interface.

Within each coarse cell $(I1,I2) \times (J1,J2) \times (K1,K2)$, an active representative cell is chosen to be located at $((II+I2)/2, (J1+J2)/2, (K1+K2)/2)$. The values of upscaled grid quantities (described below and above) are nominally associated with this cell. After processing the grid data, ECLIPSE retains just the active uncoarsened cells in the fine grid and the active representative cells in coarse groups.

Any region quantities such as SATNUM, ROCKNUM, PVTNUM for the coarse cell are therefore associated with the representative cell only. Similarly, any end-point values in a coarse cell, such as SWL, SWCR, SOWCR, etc., are taken from the values in the representative cell. No relative permeability upscaling is done here; take care not to define coarse cells too large, in order to encompass wide relative permeability variations between different rock types.

Any well completed in a coarse cell has its completions nominally moved to the representative cell (for reporting purposes) and the connection factor and drawdown are calculated using the upscaled quantities DX_c, DY_c, DZ_c, Kx_c, Ky_c, Kz_c and $DEPTH_c$. This means that in each coarse cell with a completion on the fine grid, the well is assumed to penetrate fully through the coarse cell in the appropriate direction of penetration.

Note If the connection factor, Kh or r_o (pressure equivalent radius) is given in COMPDAT, then this is not upscaled from fine to coarse grids.

Faults in coarsened regions

Where x- or y-faults exist in coarsened regions, these are represented by non-neighbor connections between cells which are physically adjacent despite having different z-coordinates. When coarsening is defined such that the fault lies between two original cells which will subsequently be in the same column of coarse cells, these connections are no longer sensible, as they connect cells in the z-direction where the geometry of the situation requires a connection in the x- or y-direction.

In this case, the ends of the non-neighbor connection are recalculated so that the connection is still made across the fault in the correct direction. This is geometrically equivalent to moving the fault to the edge of the new coarsened cell.

The transmissibility of the new non-neighbor connection is calculated exactly as for the upscaling example above, except that one of the transmissibilities in the sum will be the old non-neighbor connection transmissibility. Diffusivity and thermal transmissibility, if appropriate, are recalculated in exactly the same way.

Note There can be only one fault per direction in any one coarsened cell. So a coarsened cell can contain both an x-fault and a y-fault, but not two x-faults or two y-faults.

Automatic refinement (ECLIPSE 100)

Introduction

The Automatic Refinement facility permits the grid to be refined as a reverse process to coarsening, with minimal change to the input data. This is useful for the automatic scaling of existing datasets for massively parallel computations, and for performing grid refinement sensitivity studies. There are several caveats on its use. The facility should not be used to create 'in-place' LGRS.

You specify refinement factors (NRX, NRY, NRZ) for each grid dimension by adding the RUNSPEC section keyword AUTOREF to an existing base dataset. For a base dataset with grid dimensions NX, NY, NZ, ECLIPSE converts the input data to the refined grid, obtained by refining each base cell into NRX*NRY*NRZ cells. Each grid cell is simply refined uniformly by the factors NRX, NRY, NRZ in each dimension. It is possible to coarsen an autorefined grid so as to achieve autorefinement only in a part of the grid.

The base dataset should first be run without autorefinement to check for any errors. Error and consistency checking of the dataset with autorefinement is performed with respect to the base grid, but any output files such as GRID, RESTART or FLUX files, etc., refer to the autorefined grid.

Treatment of GRID section data in automatically refined grids

Data for the autorefined grid is inherited from host cells in the base grid. Most data is simply copied to the automatically refined cells from the host cells in the base grid. The exceptions to this rule are the geometric grid data DX, DY, DZ, TOPS, DZNET, DZMTRXV, COORD and ZCORN. These are determined such that the obvious refinement rules are observed. Note that the PINCH and PINCHOUT values are not altered from the base grid and that the MINPVV/MINPV data for the autorefined grid is simply inherited from host cells in the base grid.

By default, data for the multiplier keywords MULTX etc. is also inherited from host cells in the base grid. Therefore in the autorefined grid, MULTX(IB, JB, KB) applies to all autorefined cells (IA, JA, KA) contained within the base grid cell (IB, JB, KB) rather than just on the X+ direction interface between cells (IB, JB, KB) and (IB+1, JB, KB) in the base grid.

If item 4 of the AUTOREF keyword is set to 1, then MULTX, etc., will be applied only between autorefined cells on the X+ interface between these base grid cells.

If item 4 of the AUTOREF keyword is set to 2, then MULTX, etc., assumes its default behavior in the autorefined grid, and additional multipliers SMULTX etc. may also be entered, which only apply between autorefined cells on the X+ interface between base grid cells.

Any numerical aquifers will be represented equivalently in the autorefined grid by the central autorefined cell within the host cell in the base grid. The specified aquifer length and cross-sectional area are copied from the base grid to retain the initial volumes.

Any coarsening boxes specified in the base grid using the COARSEN keyword are autorefined. The keyword AUTOCOAR may be used to specify coarsening boxes in the base grid which are not autorefined; this allows regions of the autorefined grid to be coarsened back to the base grid if required.

Treatment of EDIT section data in automatically refined grids

Data entered in the EDIT section is upscaled from the base grid to the autorefined grid in an appropriate manner. Edited pore volumes in the base grid are applied to autorefined cells by multiplying by the ratio of new (edited) to old pore volumes for host cells in the base grid. In like fashion, any transmissibilities entered in the EDIT section are accounted for by applying a multiplier which is the ratio of new (edited) transmissibility for the host cells in the base grid to the value of transmissibility on the host cells in the base grid obtained by upscaling the calculated transmissibilities on the autorefined grid to the base grid.

Treatment of REGIONS section data in automatically refined grids

Any region data specified in the PROPS, REGIONS or SCHEDULE sections is inherited by the autorefined grid from the host cells in the base grid. This applies also to end-points specified in the PROPS section.

Treatment of SUMMARY section data in automatically refined grids

Any grid cell numbers or completion location indices for the base grid entered in the SUMMARY section are converted to the central autorefined cell within the base cell. The SUMMARY and RSM files exhibit this conversion.

Treatment of wells in automatically refined grids

A vertical well connection in an automatically refined base cell is moved to the central column of refined cells within the base cell. Thus consider a single connection in the base grid for a well penetrating in the Z direction. Within the automatically refined grid there are NRZ connections, corresponding to each layer of the automatically refined grid within the base connected cell. The connections are located in the central column of refined cells, with location $((NRX + 1)/2, (NRY + 1)/2)$. ECLIPSE treats these as independent connections (for example, they will be closed one by one in automatic workovers). If you wish these connections to be lumped together for simultaneous closure in workovers, you should use the COMPLUMP keyword, specifying the base cell coordinates in the lumped connection range. Note that all the well and connection data input in the SCHEDULE section should refer to the cell coordinates in the base grid (thus requiring no change to the data), but the results reported in the output files will refer to the refined grid cell coordinates.

This is illustrated in Figure 39.9 in the case NRX = NRY = 3:

Figure 39.9 Well positioning in automatically refined grids

Similarly, a horizontal connection (COMPDAT item 13 = 'X' or 'Y') in the base grid is replaced by NRX or NRY connections in the central X or Y direction row of the base cell.

Note The above rule is inappropriate for a well located at a corner or an edge of a base grid cell; ECLIPSE still places it in the central column or row of refined cells.

If the connection factor and Kh are not explicitly defined using keyword COMPDAT in the base dataset, then ECLIPSE calculates the connection factors within the autorefined grid in the usual way.

If Kh is defined in keyword COMPDAT then its value in the autorefined grid is set equal to the specified Kh divided by the number of divisions in the direction of penetration of the well. If the connection factor T_{wj} is defined in COMPDAT then an additional skin factor ΔS is computed such that T_{wj} equals the value calculated by the default formula on the base grid when the skin factor is incremented by ΔS. The connection factor on the autorefined grid is then computed from the default formula on the autorefined grid again with the increment ΔS added to the skin factor.

If the well's drainage radius is defaulted in item 7 of keyword WELSPECS, it takes the value of the pressure equivalent radius of the refined cell, rather than that of the base cell. This results in an increase in the reported productivity index.

Current restrictions

1. Local grid refinements may not be used with automatic refinement.
2. The NNC and EDITNNC keywords are not available with automatic refinement.
3. Wellbore friction and multi-segment wells may not be used with automatic refinement.
4. The nine-point schemes are incompatible with autorefinement, and it is not possible to re-specify transmissibility multipliers within the SCHEDULE section when using autorefinement.

Summary of keywords

RUNSPEC section

The keywords are:

Table 39.2 RUNSPEC section

Simulator	Keyword	Description
ECLIPSE 100 only	AUTOREF	sets options for autorefinement.
	LGR	sets options and dimensions for local grid refinement and coarsening.
ECLIPSE 300 only	NOAIMLGR	suppresses making all LGR cells implicit in AIM runs.

GRID section

The keywords are:

Table 39.3 GRID section

Simulator	Keyword	Description
ECLIPSE 100 only	AUTOCOAR	is analogous to the COARSEN keyword and may be used in auto-refined grids.
ECLIPSE 100 only	AMALGAM	defines groups of LGRs which are to be amalgamated.
	CARFIN	sets up a cartesian local grid refinement.
	COARSEN	defines a box of cells for grid coarsening.
	ENDFIN	terminates data for a local grid.
	EXTFIN	specifies an unstructured local grid refinement.
	EXTHOST	used to specify the global host cells for an unstructured local grid refinement.
	EXTREPGL	specifies the replaced global cells for an unstructured local grid refinement.
	GRIDFILE	controls the contents of the Grid Geometry file.
	HRFIN	can be used to dictate the size ratios of DRV values within radial local grids.
	HXFIN	can be used to dictate the size ratios of DXV values within cartesian local grids.
	HYFIN	can be used to dictate the size ratios of DYV values within cartesian local grids.
	HZFIN	can be used to dictate the size ratios of DZV values within cartesian local grids.
	INRAD	specifies the inner radius of the radial refinement.
	NXFIN	sets the number of local cells in each global cell in the x-direction.
	NYFIN	sets the number of local cells in each global cell in the y-direction.

Table 39.3 GRID section

Simulator	Keyword	Description
	NZFIN	sets the number of local cells in each global cell in the z-direction.
ECLIPSE 100 only	OUTRAD	inner radius of the outer annulus in a radial LGR.
ECLIPSE 100 only	QMOBIL	controls the mobile end point correction in an LGR.
	RADFIN	sets up a radial LGR.
ECLIPSE 100 only	RADFIN4	sets up a 4 column radial LGR.
	REFINE	indicates that subsequent data refers to a named local grid.RPTGRIDL controls grid section output for LGRs.
	TRANGL	specifies the global-local connections and associated transmissibilities for an unstructured local grid refinement.
ECLIPSE 100 only	SMULTX, SMULTY, SMULTZ	alternative transmissibility multipliers for autorefinement.
ECLIPSE 100 only	VEFIN	may be used in LGR runs with Vertical Equilibrium in the global grid, to control the Vertical Equilibrium model in the local grid systems.

EDIT section

The EDIT section keywords are:

Table 39.4 EDIT section LGR keywords

Keyword	Description
ENDFIN	terminates data for a local grid.
REFINE	indicates that subsequent data refers to a named local grid.

PROPS section

The PROPS section keywords are:

Table 39.5 PROPS section LGR keywords

Keyword	Description
ENDFIN	terminates data for a local grid.
REFINE	indicates that subsequent data refers to a named local grid.

REGIONS section

The REGIONS section keywords are:

Table 39.6 REGIONS section LGR keywords

Keyword	Description
ENDFIN	terminates data for a local grid.
REFINE	indicates that subsequent data refers to a named local grid.

SOLUTION section

The SOLUTION section keywords are

Table 39.7 SOLUTION section LGR keywords

Keyword	Description
ENDFIN	terminates data for a local grid.
GETGLOB	read only the global restart data.
REFINE	indicates that subsequent data refers to a named local grid.

SUMMARY section

Well, block and connection data for an LGR can be obtained by prefixing the appropriate keywords with an L. Each subsequent data item to which the keyword refers is then prefixed by and LGR name. (See the SUMMARY section overview for further details.)

SCHEDULE section

The SCHEDULE section keywords are:

Table 39.8 SCHEDULE section LGR keywords

Simulator	Keyword	Description
ECLIPSE 300 only	COMPAGHL	sets the gravity head in local grid completion cells.
	COMPDATL	defines well completions in an LGR.
ECLIPSE 300 only	COMPKRIL	sets injection relative permeabilities in local grid completion cells.
	COMPLMPL	lumps well completions for automatic workovers.
ECLIPSE 300 only	COMPMBIL	sets injection mobilities in local grid completion cells.
ECLIPSE 100 only	COMPRPL	scales saturations for local grid well connections.
ECLIPSE 100 only	COMPVEL	sets well connection depth data in local grids.

Table 39.8 SCHEDULE section LGR keywords

Simulator	Keyword	Description
	ENDFIN	terminates data for a local grid.
ECLIPSE 100 only	LGRFREE	switch to local time stepping.
ECLIPSE 100 only	LGRLOCK	switch to global time stepping.
ECLIPSE 100 only	LGROFF	switch a local grid refinement off.
ECLIPSE 100 only	LGRON	switch a global grid on.
ECLIPSE 300 only	LSCRITS	sets WARP linear solver convergence criteria for sub-grids.
ECLIPSE 100 only	NEXTSTPL	sets the next local time-step.
	REFINE	indicates that subsequent data refers to a named local grid.
ECLIPSE 100 only	TUNINGL	sets simulator control parameters for all local grids.
ECLIPSE 100 only	TUNINGS	sets simulator control parameters for specific local grids.
	WELMOVEL	Moves a well into an LGR at a restart.
	WELOPENL	shuts or re-opens wells or well connections in an LGR.
	WELSPECL	defines a well in an LGR.
ECLIPSE 100 only	WFRICTNL	designates a well in an LGR to be a wellbore friction well.
	WPIMULTL	multiplies well connection factors for a well in an LGR.

Global keywords

The Global keywords are:

Table 39.9 The Global LGR keywords

Simulator	Keyword	Description
ECLIPSE 100 only	OPTIONS	Switch 32 controls time-step chops due to wells dying under THP control. Switch 37 replaces local grid solution states with those of the global host cell. Switch 46 controls the interpolation of MINPV or MINPVV to LGRs. Switch 51 controls the first local grid solution following a global time-step. Switch 52 controls the use of corner point depths in VE runs. Switch 56 controls the application of LGRON/LGROFF.

Example LGR problem

This dataset describes the seventh SPE Comparative Solution Project (1991) - a horizontal well with friction. The dataset describes CASE 3A, having a 900 ft. horizontal well completed across 3 global cells in the x-direction. These 3 cells, plus an additional cell at each end (making 5 in all), are refined as follows:

Z-direction: 3 layers with a 2: 1: 2 thickness ratio

Y-direction: 3 rows with a 2: 1: 2 thickness ratio

X-direction: The two global cells at each end were refined into 4, with equal DX values. The central global cell was not refined in the x-direction.

RUNSPEC

`RUNSPEC` section

```
RUNSPEC
TITLE
 7th SPE Benchmark, Case 3a
DIMENS
 9    9    6  /
OIL
WATER
GAS
DISGAS
FIELD
TABDIMS
 1    1   15   15    1   15  /
WELLDIMS
 2    9    1    2 /
LGR
 1  150  /
FRICTION
 1 /
START
 1 'JAN' 1990  /
NSTACK
 25 /
FMTOUT
FMTIN
UNIFOUT
UNIFIN
```

GRID

GRID section

```
GRID        ======================================================
=
DXV
  9*300  /
DYV
  620  400  200  100  60  100  200  400  620  /
DZ
  81*20  81*20  81*20  81*20  81*30  81*50  /
PERMX
  486*300 /
PERMY
  486*300 /
PERMZ
  486*30 /
PORO
  486*0.2 /
EQUALS
'TOPS' 3590  1 9  1 9  1 1  /
/
```

Specify the refined local grid. The global cells I=5-9, J=5, K=1 are refined into a 9x3x3 system.

```
CARFIN
-- name  I1-I2  J1-J2  K1-K2  nx ny nz
  'LGR1' 5  9   5  5   1  1   9  3  3  /
```

Specify the x-direction partition of the 5 global cells into 9. The cells at either end are partitioned into 4 equal-sized cells. The middle cell is not partitioned in the x-direction.

```
NXFIN
 2  2  1  2  2  /
```

Specify the size ratio of the local grid cells in the y-direction. The global cells are divided into 3 rows in the y-direction, with a 2:1:2 thickness ratio.

```
HYFIN
 2.0  1.0  2.0  /
```

Specify the size ratio of the local grid cells in the z-direction. The global cells are divided into 3 layers in the z-direction, with a 2:1:2 thickness ratio.

```
HZFIN
 2.0  1.0  2.0  /
```

End of local grid specification

```
ENDFIN
```

PROPS

Use Stone's Method 2 for 3-phase oil relative permeability.

```
PROPS      ==========================================================
STONE2
SWOF
-- Sw    Krw    Krow     Pcow
  0.22   0.0    1.0000   6.30
  0.3    0.07   0.4000   3.60
  0.4    0.15   0.1250   2.70
  0.5    0.24   0.0649   2.25
  0.6    0.33   0.0048   1.80
  0.8    0.65   0.0      0.90
  0.9    0.83   0.0      0.45
  1.0    1.0    0.0      0.0    /
SGOF
-- Sg    Krg    Krog     Pcgo
  0.0    0.0    1.00     0.0
  0.04   0.0    0.60     0.2
  0.1    0.022  0.33     0.5
  0.2    0.1    0.10     1.0
  0.3    0.24   0.02     1.5
  0.4    0.34   0.0      2.0
  0.5    0.42   0.0      2.5
  0.6    0.5    0.0      3.0
  0.7    0.8125 0.0      3.5
  0.78   1.0    0.0      3.9    /

PVTW
-- Pref    Bw       Cw       Vw    Viscosibility
   3600  1.00329   3.0E-6   0.96         0  /
ROCK
-- Pref     Cr
   3600    4.0D-6   /
DENSITY
--  oil   water   gas
    45.0  62.14   0.0702  /
```

```
PVDG
-- Pgas   Bg   Viscg
    400  5.9   0.013
    800  2.95  0.0135
   1200  1.96  0.014
   1600  1.47  0.0145
   2000  1.18  0.015
   2400  0.98  0.0155
   2800  0.84  0.016
   3200  0.74  0.0165
   3600  0.65  0.017
   4000  0.59  0.0175
   4400  0.54  0.018
   4800  0.49  0.0185
   5200  0.45  0.019
   5600  0.42  0.0195   /
PVCO
-- Pbub   Rs     Bo     Visco    Co     CVo
    400  0.165  1.012   1.17   1.0E-5   0.0
    800  0.335  1.0255  1.14     2*
   1200  0.500  1.038   1.11     2*
   1600  0.665  1.051   1.08     2*
   2000  0.828  1.063   1.06     2*
   2400  0.985  1.075   1.03     2*
   2800  1.130  1.087   1.00     2*
   3200  1.270  1.0985  0.98     2*
   3600  1.390  1.11    0.95     2*
   4000  1.500  1.12    0.94     2*
   4400  1.600  1.13    0.92     2*
   4800  1.676  1.14    0.91     2*
   5200  1.750  1.148   0.9      2*
   5600  1.810  1.155   0.89     2*
  /
PMAX
 6000  /
```

SOLUTION

The specified initial conditions can be obtained by equilibration.

```
SOLUTION  ==========================================================
EQUIL
-- Dep   Pres  Dwoc  Pcwoc  Dgoc  Pcgoc  Rsvd  Rvvd  Fipc
   3600  3600  3700   0.0   3590   0.0    1      0     0  /
RSVD
-- Dep   Rs
   3500  1.41
   3750  1.41  /
RPTSOL
 'PRES'  'SOIL'  'SWAT'  'SGAS'  'RS'  /
```

SUMMARY

SUMMARY section

```
SUMMARY     ==========================================================
FOPR
FGPR
FWPR
FOPT
FGPT
FWPT
FWCT
FGOR
LWBHP
'LGR1' 'PROD' /
/
LCPR
'LGR1' 'PROD' 3 2 2 /
/
FPR
FWIR
FWIT
RUNSUM
```

SCHEDULE

SCHEDULE section

```
SCHEDULE    ==========================================================
RPTSCHED
 'WELLS=2'  'SUMMARY=2'  'WELSPECS'  /
```

Specify producer in local grid LGR1:

```
WELSPECL
-- Name  Group  LGR    I  J  Dbh   Phase
  'PROD' 'G'   'LGR1'  8  5  3600  'OIL' /
/
```

Specify injector in global grid:

```
WELSPECS
-- Name  Group  I  J  Dbh   Phase
   'INJ' 'G'    9  5  3725  'WAT' /
/
```

Specify connections of producer in local grid:

```
COMPDATL
-- Name   I  J  K1-K2  Op/Sh  SatTab  ConFact  Diam  KH S D  Dirn
  'PROD' 3  2  2  2   'OPEN' 2*              0.375   3*    'X'  /
  'PROD' 4  2  2  2   'OPEN' 2*              0.375   3*    'X'  /
  'PROD' 5  2  2  2   'OPEN' 2*              0.375   3*    'X'  /
  'PROD' 6  2  2  2   'OPEN' 2*              0.375   3*    'X'  /
  'PROD' 7  2  2  2   'OPEN' 2*              0.375   3*    'X'  /
/
```

Specify connections of injector in global grid:

```
COMPDAT
-- Name   I  J  K1-K2  Op/Sh  SatTab  ConFact
  'INJ' 1  5  6  6  'OPEN' 1*       24.34  /
  'INJ' 2  5  6  6  'OPEN' 1*       24.34  /
  'INJ' 3  5  6  6  'OPEN' 1*       24.34  /
  'INJ' 4  5  6  6  'OPEN' 1*       24.34  /
  'INJ' 5  5  6  6  'OPEN' 1*       24.34  /
  'INJ' 6  5  6  6  'OPEN' 1*       24.34  /
  'INJ' 7  5  6  6  'OPEN' 1*       24.34  /
  'INJ' 8  5  6  6  'OPEN' 1*       24.34  /
  'INJ' 9  5  6  6  'OPEN' 1*       24.34  /
/
```

Specify the friction data for the producer. The grid block locations refer to the local grid.

```
WFRICTNL
-- Name   Diam   Roughness
  'PROD' 0.375  3.75E-4  /
-- Local Grid   I J K  Tlen1  Tlen2
       1*       7 2 2   0.0    150  /
       1*       6 2 2   150    300  /
       1*       5 2 2   300    600  /
       1*       4 2 2   600    750  /
       1*       3 2 2   750    900  /
/
```

Specify the production well constraints:

```
WCONPROD
-- Name   Op/Sh  CtlMode  Orat  Wrat  Grat  Lrat  Resv  Bhp
  'PROD' 'OPEN' 'LRAT' 3*              9000  1*    1500  /
/
```

Specify the injection well constraints:

```
WCONINJE
-- Name  Phase   Op/Sh  CtlMode  Srat  Resv  Bhp
  'INJ' 'WAT' 'OPEN' 'BHP' 2*         3700  /
/
```

time steps and reports.

```
TSTEP
 4*100  /
RPTSCHED
 'PRES'  'SWAT'  'SGAS'  'FIP'  'WELLS=2'
 'SUMMARY=2'  'CPU=2'  'WELSPECS'  /
TSTEP
 100  /
RPTSCHED
 'WELLS=2'  'SUMMARY=2'  'WELSPECS'  /
TSTEP
 4*100  /
RPTSCHED
 'PRES'  'SWAT'  'SGAS'  'FIP'  'WELLS=2'
 'SUMMARY=2'  'CPU=2'  'WELSPECS'  /
TSTEP
 100  /
RPTSCHED
 'WELLS=2'  'SUMMARY=2'  'WELSPECS'  /
TSTEP
 4*100  /
RPTSCHED
 'PRES'  'SWAT'  'SGAS'  'FIP'  'WELLS=2'
 'SUMMARY=2'  'CPU=2'  'WELSPECS'  /
TSTEP
 100  /
END
```

Miscible Flood Modeling

Chapter 40

Introduction

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL

This chapter describes the ECLIPSE 100 Miscible Flood model.

The Miscible Flood model is an implementation of the empirical treatment suggested by M. Todd and W. Longstaff (see [Ref. 24]).

The ECLIPSE 100 Solvent model also provides a 4-component extension of the black oil model to enable modeling of reservoir recovery mechanisms in which injected fluids are miscible with the hydro-carbons in the reservoir to model gas injection projects without going to the complexity and expense of using a compositional model. The solvent extension implements the Todd and Longstaff empirical model for miscible floods and is flexible enough to model a wide range of gas injection schemes. See "The Solvent Model" on page 955 for more information.

An alternative approach to miscibility is adopted in ECLIPSE 300, which is based on the use of surface tensions based on the Macleod-Sugden correlation; see "Surface Tension Effects" on page 983 for details. This chapter refers only to the Todd-Longstaff model adopted in ECLIPSE 100.

The Todd-Longstaff mixing parameter model

The model we shall consider is a 3-component system consisting of reservoir oil, injection gas (solvent) and water. The reservoir oil component consists of stock tank oil together with the associated solution gas. The solvent and reservoir oil components are assumed to be miscible in all proportions and consequently only one hydrocarbon phase exists in the reservoir. The relative permeability requirements of the model are those for a two-phase system (water/hydrocarbon). The Todd-Longstaff mixing parameter technique requires modification of the viscosity and density calculations in a black oil simulator.

The Todd-Longstaff model is an empirical treatment of the effects of physical dispersion between the miscible components in the hydrocarbon phase. The model introduces an empirical parameter, ω, whose value lies between 0 and 1, to represent the size of the dispersed zone in each grid cell. The value of ω thus controls the degree of fluid mixing within each grid cell. A value of $\omega=1$ models the case when the size of the dispersed zone is much greater than a typical grid cell size and the hydrocarbon components can be considered to be fully mixed in each cell. In this case the miscible components have the same value for the viscosity and density, as given by the appropriate mixing rule formulae. A value of $\omega=0$ models the effect of a negligibly thin dispersed zone between the gas and oil components, and the miscible components should then have the viscosity and density values of the pure components. In practical applications an intermediate value of ω would be needed to model incomplete mixing of the miscible components.

Effect of water saturation

A feature of miscible gas injection processes that may also be modeled is the screening effect of high water saturation on the contact between the miscible gas and the in-place oil in each grid cell. The effective residual oil saturation to a miscible gas drive is found to increase with increasing water saturation and correct modeling of the effect is important since it may reduce the efficiency of the miscible displacement. The process is modeled by introducing an effective residual oil saturation, S_{or}, which depends on the water saturation ($S_{or} = S_{or}(S_w)$). A mobile oil saturation is then calculated by

$$S_o^* = \mathrm{MAX}(S_o - S_{or}, 0.0) \qquad \text{[EQ 40.1]}$$

The mobile oil saturation S_o^* is then used to determine the relative permeabilities of miscible components and the effective gas and oil viscosities and densities in each grid cell. The keyword SORWMIS sets the effective residual saturation.

Relative permeability model

The solvent and reservoir oil are considered to be miscible components of the hydrocarbon (non-wetting) phase. The flow is 2-phase in character and 2-phase relative permeability curves need to be defined for the water and hydrocarbon phases, say $K_{rw}(S_w)$ and $K_{rn}(S_n)$ (where S_w is the water saturation and S_n $(= S_o + S_s)$ is the hydrocarbon phase saturation). K_{rn} can be considered to be the relative permeability of oil as measured in a water flood test. The oil and solvent relative permeabilities are proportional to their local volume ratio in the hydrocarbon phase,

$$K_{ro} = \left(\frac{S_o}{S_o + S_s}\right) K_{rn}(S_n) \qquad \text{[EQ 40.2]}$$

$$K_{rs} = \left(\frac{S_s}{S_o + S_s}\right) K_{rn}(S_n) \qquad \text{[EQ 40.3]}$$

It is possible to modify the straight line miscible relative permeabilities using the `MSFN` keyword. In this case the oil and miscible gas relative permeabilities are:

$$K_{ro} = M_{kro} \left(\frac{S_o}{S_o + S_s}\right) K_{rn}(S_n) \qquad \text{[EQ 40.4]}$$

$$K_{rs} = M_{krs} \left(\frac{S_s}{S_o + S_s}\right) K_{rn}(S_n) \qquad \text{[EQ 40.5]}$$

where the M_{kro} and M_{krs} functions are supplied with the `MSFN` keyword.

Miscibility of oil and solvent implies zero capillary pressure between the solvent and oil and thus $P_{cos} = 0$. However a non-zero capillary pressure is possible between the water and hydrocarbon phases, $P_{cnw}(S_w)$, dependent on the water phase saturation. The component pressures are given by

$$P_s = P_o \qquad \text{[EQ 40.6]}$$

$$P_w = P_o - P_{cnw}(S_w) \qquad \text{[EQ 40.7]}$$

Note that equations [EQ 40.1], [EQ 40.3] and [EQ 40.2] can produce a discontinuity in the oil and solvent relative permeabilities at the oil saturation $S_o = S_{or}$ if the maximum value of the effective residual oil saturation exceeds the residual saturation of the oil to water flooding. ECLIPSE experiences non-linear convergence problems if this situation arises, but it issues a warning message during data checking.

Viscosity model

The following form is suggested by Todd-Longstaff for the effective oil and solvent viscosities to be used in an immiscible simulator:

$$\mu_{o\ \mathrm{eff}} = \mu_o^{1-\omega}\mu_m^{\omega} \qquad \text{[EQ 40.8]}$$

$$\mu_{s\ \mathrm{eff}} = \mu_s^{1-\omega}\mu_m^{\omega} \qquad \text{[EQ 40.9]}$$

If ω=1 then $\mu_{o\ \mathrm{eff}} = \mu_{s\ \mathrm{eff}} = \mu_m$ where μ_m is the viscosity of a fully mixed oil-solvent system. The formula to be used for μ_m is the 1/4-power fluid mixing rule

$$\frac{1}{\mu_m}^{1/4} = \frac{S_s}{S_n}\left(\frac{1}{\mu_s}\right)^{1/4} + \frac{S_o}{S_n}\left(\frac{1}{\mu_o}\right)^{1/4} \qquad \text{[EQ 40.10]}$$

$$\mu_m = \frac{\mu_o\mu_s}{\left(\frac{S_s}{S_n}\cdot\mu_o^{1/4} + \frac{S_o}{S_n}\cdot\mu_s^{1/4}\right)^4} \qquad \text{[EQ 40.11]}$$

$$S_n = S_o + S_s \qquad \text{[EQ 40.12]}$$

The case ω=1 models a large dispersed oil-solvent zone. The Todd-Longstaff model treats this case as a local unit mobility ratio displacement.

If ω=0 then $\mu_{o\ \mathrm{eff}} = \mu_o$, $\mu_{s\ \mathrm{eff}} = \mu_s$ and each component has an effective viscosity equal to its 'pure' value. Such a case corresponds to a local high adverse mobility ratio displacement and models a negligibly thin oil-solvent dispersed zone.

The mixing parameter approach allows the case of a partial mixing zone to be modeled by choosing an intermediate value of ω.

Component density model

The treatment of effective oil and solvent densities in the ECLIPSE model is based on the same 1/4-power rule as the effective viscosities. By default the density calculation will use the same mixing parameter as the viscosity. However, a separate mixing parameter may optionally be specified for the effective density calculation.

The densities are computed by the following procedure. First, the partially mixed or effective viscosities are calculated using equations [EQ 40.8] and [EQ 40.9] and [EQ 40.10], [EQ 40.11] and [EQ 40.12] above. The value of each effective component viscosity is then used in turn in [EQ 40.10], [EQ 40.11] and [EQ 40.12] to yield an effective saturation fraction to be used in oil and solvent density calculations. The effective saturation fractions used for density calculations are

$$\left(\frac{S_o}{S_n}\right)_{oe} = \frac{\mu_{o\,\mathrm{eff}}^{1/4} \cdot \mu_o^{1/4} - \mu_s^{1/4} \cdot \mu_o^{1/4}}{\mu_{o\,\mathrm{eff}}^{1/4} \cdot (\mu_o^{1/4} - \mu_s^{1/4})} \qquad \text{[EQ 40.13]}$$

$$\left(\frac{S_o}{S_n}\right)_{se} = \frac{\mu_{s\,\mathrm{eff}}^{1/4} \cdot \mu_o^{1/4} - \mu_s^{1/4} \cdot \mu_o^{1/4}}{\mu_{s\,\mathrm{eff}}^{1/4} \cdot (\mu_o^{1/4} - \mu_s^{1/4})} \qquad \text{[EQ 40.14]}$$

The effective oil and solvent densities (ρ_{oeff}, ρ_{seff}) are now computed from the effective saturations fractions in expressions [EQ 40.13] and [EQ 40.14] and the pure component densities (ρ_o, ρ_s) using the following formulae:

$$\rho_{o\,\mathrm{eff}} = \rho_o\left(\frac{S_o}{S_n}\right)_{oe} + \rho_s\left[1 - \left(\frac{S_o}{S_n}\right)_{oe}\right] \qquad \text{[EQ 40.15]}$$

$$\rho_{s\,\mathrm{eff}} = \rho_o\left(\frac{S_o}{S_n}\right)_{se} + \rho_s\left[1 - \left(\frac{S_o}{S_n}\right)_{se}\right] \qquad \text{[EQ 40.16]}$$

The ECLIPSE model allows you to specify a different surface density for the solvent gas to that for the solution gas. In equations [EQ 40.15] and [EQ 40.16], the pure solvent density at reservoir conditions, ρ_s, is computed from the solvent surface density. The oil density at reservoir conditions, ρ_o, is calculated from the stock tank oil and solution gas surface densities.

Note that equations [EQ 40.13] and [EQ 40.14] are singular for unit mobility ratio displacements (that is $\mu_o = \mu_s$) and the model should not be used in such cases. For future applications in cases where $\mu_o = \mu_s$ it may prove necessary to implement the alternative density mixing calculation proposed by Todd-Longstaff.

The mixing parameter model would be of limited use unless the mixing parameter could itself be modeled over a wide range of operating conditions. A value of ω=1 results in a piston-like displacement of oil by the injected solvent. If ω=0 the displacement is similar to an immiscible displacement (except for the treatment of relative permeability). An intermediate value of ω results in a continuous solvent saturation increase behind the solvent front. Todd-Longstaff accounted for the effects of viscous fingering in 2-D studies by setting ω=2/3 independent of mobility ratio. For field scale simulations they suggested setting ω=1/3. However, in general history matching applications the mixing parameter may be regarded as a useful history matching variable to account for any reservoir process inadequately modeled.

The control of miscibility with pressure

In many miscible displacements, the gas is only miscible with the reservoir oil at high pressure. Typically the gas-oil capillary pressure reduces with increasing pressure, and only when it has reduced to zero can the two fluids be considered to be miscible. It is possible to model the transition between miscibility and immiscibility as a function of pressure by using the `PMISC` keyword.

The `PMISC` function interpolates between the immiscible and miscible values of the PVT properties, relative permeability data and capillary pressure data.

The PVT properties are interpolated as follows (taking gas properties as an example):

$$\frac{1}{B_g} = \frac{1}{B_{gm}} M_p + \frac{1}{B_{gi}}(1 - M_p) \qquad \text{[EQ 40.17]}$$

$$\frac{1}{B\mu_g} = \frac{1}{B\mu_{gm}} M_p + \frac{1}{B\mu_{gi}}(1 - M_p) \qquad \text{[EQ 40.18]}$$

where

B_g = gas formation volume factor

B_{gi} = immiscible gas formation volume factor

B_{gm} = miscible gas formation volume factor

$B\mu_g$ = gas FVF * viscosity

$B\mu_{gi}$ = gas FVF * viscosity (immiscible)

$B\mu_{gm}$ = gas FVF * viscosity (miscible)

M_p = pressure dependent miscibility function.

The miscible formation volume factor is calculated from the Todd-Longstaff density:

$$B_{gm} = \frac{\mathrm{SD}_s}{R_{s\ \mathrm{eff}}} \qquad \text{[EQ 40.19]}$$

where

SD_s = surface density of solvent

$R_{s\ \mathrm{eff}}$ = effective solvent density in [EQ 40.16].

The relative permeability interpolation algorithm has two steps:

1 Scale the relative permeability end-points by the miscibility function. For example, the residual oil saturation:

$$S_{or} = S_{orm} M_p + S_{ori}(1 - M_p)$$

where

S_{orm} = miscible residual oil

S_{ori} = immiscible residual oil

M_p = pressure dependent miscibility function.

2 Calculate the miscible and immiscible relative permeability scaling for the new end points. Then the relative permeability is again an interpolation between the two using the miscibility function:

$$K_r = K_{rm} M_p + K_{ri}(1 - M_p)$$

where

K_{rm} = scaled miscible K_r

K_{ri} =scaled immiscible K_r.

The capillary pressure is treated as follows:

$$P_{cog} = P_{cog\ (\mathrm{misc})} M_p + P_{cog\ (\mathrm{immisc})}(1 - M_p) \qquad \text{[EQ 40.20]}$$

$$P_{cog\ (\mathrm{misc})} = P_{cog}(S_g)$$

$$P_{cog\ (\mathrm{immisc})} = P_{cog}(S_g + S_s) \qquad \text{[EQ 40.21]}$$

where

S_g = lean gas saturation

S_s = solvent saturation.

Control of numerical diffusion in miscible flow simulation

The relative permeability of the oil and solvent components in the Todd-Longstaff model are specified by equations [EQ 40.3] and [EQ 40.2] or [EQ 40.4] and [EQ 40.5]. The relative permeability of each component is essentially proportional to the saturation of each component for low saturations. In such cases the influence of numerical diffusion in spreading the miscible components can be unacceptably high, and some control of the numerical diffusion errors is required.

A two-point upstream algorithm operating only on the miscible component relative permeabilities is available for use in the ECLIPSE Miscible Flood option to control the numerical diffusion of oil and solvent. The two-point upstream projection technique is illustrated in Figure 40.1.

Figure 40.1 The two-point upstream projection technique

We require to compute the flow of component *C* between cells i and j, which we take to be aligned along the X-axis. Suppose that cell i is the upstream cell for the flow of component *C* between i and j. Let cell k be the second upstream cell to cell j and K_{ck}, K_{ci} and K_{cj} denote the component relative permeabilities in cells k, i and j respectively. DX_k, DX_i and DX_j are the X-direction cell sizes.

For single point upstream weighting, the flow rate of component C across the i-j cell interface is $F_{ij,c}$ where:

$$F_{ij,c} = T_{ij}\frac{K_{ci}}{B_{ci}\,\mu_{ci}}\mathrm{DP}_{ij,c} \qquad \text{[EQ 40.22]}$$

where

T_{ij} is the transmissibility between cells i and j

$\mathrm{DP}_{ij,c}$ is the potential difference between cells i and j for component C

B_c, μ_c are the formation volume factor and viscosity of component C

The two-point upstreaming algorithm operating on component relative permeabilities replaces K_{ci} in the equation [EQ 40.22] by a linearly extrapolated relative permeability, $K_{ci}*$, obtained from cells i and k,

$$K_{ci}* = \frac{2\mathrm{DX}_i + \mathrm{DX}_k}{\mathrm{DX}_i + \mathrm{DX}_k}K_{ci} - \frac{\mathrm{DX}_i}{\mathrm{DX}_i + \mathrm{DX}_k}K_{ck} \qquad \text{[EQ 40.23]}$$

The above expression for $K_{ci}*$ can yield negative values or values greater than unity, and in such cases needs to be restricted to physical values. This is most simply achieved by constraining $K_{ci}*$ to lie between the values K_{ci} and K_{cj}. Thus,

$$\mathrm{MIN}(K_{ci}, K_{cj}) < K_{ci}^* < \mathrm{MAX}(K_{ci}, K_{cj}) \quad \text{[EQ 40.24]}$$

In some cases a single point upstream calculation of K_{ci}^* (that is $K_{ci}^* = K_{ci}$) should be used instead of the two-point upstream value. Such cases are

1. Cell k lies outside the reservoir mesh (that is the k-i cell boundary is an edge of the reservoir.
2. Cell k is inactive.
3. The k-i cell boundary contains a zero transmissibility multiplier.
4. Cells i and j are coupled by a non-neighbor connection.

Using the ECLIPSE miscible flood option

The ECLIPSE Miscible Flood option can only be used in oil-gas and oil-gas-water combinations (that is gas and oil are treated as the miscible components). The keyword MISCIBLE in the RUNSPEC section activates the model. The two-point upstreaming calculation of component relative permeabilities is achieved by setting item 3 in the MISCIBLE keyword to TWOPOINT, to allow comparison of runs with and without numerical diffusion control.

The keyword MISCNUM should be used in the REGIONS section to define the miscibility region to which each grid cell belongs. This keyword is not obligatory, and a default region number of 1 is assumed. The maximum region number that can be defined (NTMISC) must be entered in item 1 of the keyword MISCIBLE in the RUNSPEC section. Echo of the MISCNUM region data can be obtained using the mnemonic MISCNUM in the RPTREGS keyword.

It is possible to override the miscible flow calculation of the gas and oil properties in certain cells in the grid. If a cell is allocated a MISCNUM region number of zero, then the gas and oil will be assumed to be immiscible under all conditions in the cell.

The mixing parameter, ω, should be input in the PROPS section under the keyword TLMIXPAR. A single real number in the range 0 to 1 must be supplied for each miscible region. If a different mixing parameter is required for the effective densities, a second mixing parameter can be supplied for each miscible region. If the grid contains any cells whose MISCNUM region is zero, the mixing parameter is automatically set to zero in those cells.

For gas-oil runs, the keywords SGFN and SOF2 should only be entered if there is at least one cell which has a MISCNUM region number of zero. For those cells that are allocated a positive MISCNUM region number, ECLIPSE assumes that the gas-oil capillary pressure is zero and the relative permeabilities are equal to their respective saturations.

In oil-water-gas runs the keywords SOF2 and SWFN are obligatory. In the miscible regions, the SWFN data defines the water relative permeability and water-hydrocarbon capillary pressure in the usual way. The SOF2 keyword represents the relative permeability of the hydrocarbon phase with respect to hydrocarbon phase saturation (the function $K_{rn}(S_n)$ in equations [EQ 40.3] and [EQ 40.2]). If there are any immiscible regions in the grid, the SOF3 and SGFN keywords must be used to enter the immiscible gas-oil relative permeabilities.

The hysteresis model is available in both the miscible and the immiscible zones. In the miscible zones, a two phase treatment (water/hydrocarbon) of the hysteresis effect is invoked on the water-hydrocarbon capillary pressures and relative permeabilities.

The Saturation Table Rescaling option is also available in both the miscible and immiscible zones of the grid. In the miscible regions, the saturation table endpoints for the hydrocarbon phase should be entered as oil-in-water data using the appropriate scaling keywords. Data entered for gas end-points will be ignored in the miscible zones.

The miscible residual oil saturation and critical gas saturation versus water saturation tables can be defined in the PROPS section using the keywords SORWMIS (and SGCWMIS when the Solvent model is used). A maximum of NTMISC tables, each with a maximum of NSMISC rows of data, are allowed. The SORWMIS/ SGCWMIS keywords are not obligatory and an effective residual oil saturation of 0.0 and critical gas value of 0.0 will be assumed if the appropriate keyword is omitted. The miscible solvent-oil relative permeability curves can modified using the MSFN keyword. If the MSFN keyword is omitted, straight line relative permeability curves are used.

The calculation of the initial reservoir state in the miscible zones is performed using capillary pressures defined in equations [EQ 40.6] and [EQ 40.7] with end points implied from equations [EQ 40.3] and [EQ 40.2] or [EQ 40.4] and [EQ 40.5].

The surface densities of the stock-tank oil, water and solution gas components should be defined using the existing DENSITY or GRAVITY keyword. The surface density of the injection gas must be entered using the SDENSITY keyword in the PROPS section. In the immiscible zones, the injection gas surface density is taken from the DENSITY or GRAVITY keyword to equal that of the solution gas. An echo of the surface density of each component can be obtained using the mnemonic DENSITY in the RPTPROPS keyword.

Specification of pressure function data for the pure components is unaltered from the usual data requirements of ECLIPSE. If the oil is defined as dead (no DISGAS keyword in RUNSPEC) then the initial solution gas is omitted from the GOR reports in the results file and the GOR refers to the solvent-oil ratio. It is possible to include the initial solution gas in the GOR reports by defining the oil component to be live (keyword DISGAS in RUNSPEC). In either case, the reported saturation values for oil and solvent refer to the relative volume occupied by each component as if they were not mixed.

The pressure dependence of miscibility can be controlled by using the PMISC keyword. This keyword is optional; if it is not present then miscibility is assumed at all pressures.

If DISGAS has been specified in the RUNSPEC section, the value of DRSDT refers to any immiscible zones in the grid. Its default value in these regions will be infinity. In the miscible zones, the value of DRSDT is always zero.

No new keywords have been added in the SUMMARY file section. The existing graphics facilities allow line graphs to be obtained of total gas injection / production rates for connections, wells, groups and the field; also for cumulative totals, inter-region gas flows, gas-in-place, etc. If the injected gas needs to be distinguished from the initial solution gas, then the existing tracer option allows separate line graphs of the above quantities to be produced to distinguish the two types of gas.

The two-point upstreaming technique is intended for use in dispersed flow models and is not compatible with the calculation of relative permeability in the Vertical Equilibrium option. The Todd-Longstaff model is based on a dispersed fluid distribution throughout each grid cell. It is therefore suggested that the Vertical Equilibrium model should not be used with the Miscible Flood option.

The two-point upstreaming algorithm produces substantial reductions in the level of numerical diffusion associated with component relative permeabilities. However, the algorithm inevitably involves an increase in the property evaluation and matrix element computation time as well as an increase in the work performed by the linear solver. Experience on small problems has indicated an approximate increase of around 50% in run times due to the use of two-point upstreaming.

Mixed Water Relative Permeability Modeling

Chapter 41

Mathematical Model

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

An alternative model, that takes into account the dissolution of gases, may be used to calculate the water relative permeability during a CO2SOL, H2SSOL or GASSOL simulation. This optional mixing model is activated using the keyword MIXWPERM.

In this case, the standard water relative permeability is corrected to take into account the hydrocarbon moles dissolved in the water phase, using the expression:

$$\tilde{K}_w = \frac{m_w K_w + \sum_i (m_i)_{Aq} [\alpha_i(P, T) K_o + \beta_i(P, T) K_g]}{m_w + \sum_k (m_k)_{Aq}}$$

in which $\tilde{K}_w$ denotes the new water relative permeability obtained following the mixing calculation, K_w denotes the standard water relative permeability prior to mixing, K_o and K_g denote, respectively, the oil and gas phase permeabilities m_w represents the water component molar density, meanwhile $(m_k)_{Aq}$ and $(m_i)_{Aq}$ denote, respectively, the aqueous molar density of components k and i defined by CNAMES, indices k and i each spanning all of the soluble hydrocarbon components. Finally, denoting pressure and temperature using P and T respectively, the oil and gas activity functions for component i are represented, respectively, by:

$$\alpha_i(P, T) = a_{i1}(P) + a_{i2}(P)T + a_{i3}(P)T^2$$

$$\beta_i(P, T) = b_{i1}(P) + b_{i2}(P)T + b_{i3}(P)T^2$$

The coefficients a_{i1}, a_{i2} and a_{i3} are interpolated, respectively, from columns 2, 3 and 4 of the MIXACTO keyword input table for component i. Similarly, coefficients b_{i1}, b_{i2} and b_{i3} are interpolated, respectively, from columns 2, 3 and 4 of the MIXACTG keyword input table for component i.

Note that the sum $\alpha_i + \beta_i$ must not exceed unity for the given pressure and temperature, for any component i. This ensures that the new water relative permeability following mixing is bounded by unity. If $\alpha_i + \beta_i$ exceeds unity, the simulation will stop and the activity function coefficients must be reduced accordingly.

Multi-phase flash

Chapter 42

Introduction

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

Multi-phase components can be used in EoS runs for asphaltene prediction. These components can appear in the solid, liquid or gas phases.

Multi-phase keywords

RUNSPEC section keywords

The RUNSPEC keywords are:

Table 42.1 RUNSPEC keywords

Keyword	Description
MULTIPHA	Requests the multi-phase flash which can be used to predict asphaltene precipitation.
SOLID	Requests Solids option

PROPS section keywords

The PROPS keywords are:

Table 42.2 PROPS keywords

Keyword	Description
HYDRO	Define multi-phase components

SUMMARY section keywords

Solid properties can be output to the summary file. Keywords are given in Table 42.3

Table 42.3 Solid properties that can be output to the summary file

Keyword	Output
BBSOL	Solid reservoir molar density
BDENS	Solid Density
BSDEN	Solid DENsity (same as BDENS)
BSSAT	Solid SATuration
BSSOLID	Solid Saturation (same as BSSAT)

SCHEDULE section keywords.

Table 42.4 Solid properties that can be output with RPTRST, RPTSCHED and RPTSOL

Keyword	Output
BSOL	Solid reservoir molar densities
DENS	Solid reservoir density
SSOLID	Solid saturations

Using multi-phase

The SOLID keyword must be used if multi-phase components are present.

Multi-phase components

The simulator can be used to predict when asphaltene precipitation first occurs. At the end of each time step a multi-phase flash is performed to determine whether a stable asphaltene phase exists. The multi-phase flash first determines whether one or more liquid phases are stable. A liquid phase is then identified as an asphaltene phase if:

1 the heaviest hydrocarbon is an aromatic (see the HYDRO keyword), and

2 the heaviest hydrocarbon represents more than a third of the phase (by weight).

The saturation, density and molar density of the asphaltene phase are stored in the simulator as solid properties. At present the asphaltene phase is ignored in the flow calculations, so this model should only be used as an indicator of whether asphaltene precipitation occurs in the reservoir.

Asphaltene prediction is carried out in compositional runs using an Equation of State, if the SOLID keyword is used, and if the heaviest hydrocarbon is an aromatic (see the HYDRO keyword).

Multi porosity

Chapter 43

Introduction

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

In ECLIPSE 300 it is possible to set up a multi porosity model as an extension to the dual porosity model. We then have the fracture system plus a given number of matrix pore systems.

There are two ways to supply the grid properties for the matrix sub cells:

1 Using the keyword NMATOPTS the matrix sub cells are assigned transmissibilities and pore volumes as described in the "Discretized matrix model (ECLIPSE 100)" on page 263. The grid is geometrically partitioned as a “russian doll” of nested matrix subcells. This enables grid data to be supplied only for the dual porosity part of the grid and the nested matrix subgrid cells data are generated automatically.

2 Input over the grid. The data is supplied for the entire simulation grid, similarly to the dual porosity model, but with grid data such as permeabilities also for the matrix-matrix connections.

In either case the number of pore systems is input using the NMATRIX keyword. This general setup gives a variety of possibilities for studying the transient behavior in the matrix. In particular the option can be used for

1 Coal bed methane. See "Adsorption model in ECLIPSE 300" on page 163.

2 Shale gas reservoirs. See "Shale Gas" on page 921.

The special case of triple porosity systems to represent vuggy carbonates uses the same underlying multiporosity model, but restricts the number of porosities to 3 (fracture, matrix, vugs with fracture and matrix connections) or 4 (with additional “isolated” vugs connected only to the matrix). This model is activated using the TRPLPORO keyword, which also specifies if isolated vugs are to be used. The data is supplied for the entire simulation grid, as with the multiporosity option (2) above. This model is described in more detail in the Triple Porosity section below. Triple porosity is also available in ECLIPSE 100 but the general multi-porosity feature is **not** available.

Using the Multi porosity option

RUNSPEC Section

The number of simulation cells in the Z-direction, as specified by the DIMENS keyword, needs to be a multiple of the number of pore systems. The number of pore systems is equal to the number of matrix pore systems, set by the NMATRIX keyword, increased by one to account for the fracture system (NPOROS=NMATRIX+1).

The DUALPORO keyword is also required. If a dual permeability calculation is to be made where matrix to matrix connections are also used, then the DUALPERM keyword is also required. For multi-porosity, this will create a multi-permeability model - the sub-matrices will also be connected together. In most cases, this may not be the required solution, as the primary job of the sub-matrices is usually to model transients in the matrix-to-fracture flow. If this is the case, the user should make the multi-porosity model a dual-permeability model by using multipliers of size zero for the z-direction permeabilities or transmissibilities.

Note It is important to note that the ECLIPSE 300 model requires the full grid dimensions to be set by DIMENS, unlike the discretized matrix model in ECLIPSE 100.

Grid section

In order to be compatible with standard dual porosity input, the first NDIVIZ/NPOROS layers are associated with the outermost matrix cells connecting to the fractures. The next layers are associated with the fracture cells. Thereafter the matrix sub cells follow as illustrated in the figure below.

Figure 43.1 Multi porosity example having three pore systems.

The connections between the porosity systems are automatically generated. It is possible to supply the grid for the outermost primary matrix grid cells only, and using the keyword `DPGRID` the fracture and sub matrix grid are copied from the outermost matrix grid if they are not specified.

By default properties such as porosities and permeabilities together with `SIGMAV` should be input for the submatrix grid cells. The transmissibilities between the subgrid cells are then calculated as for the matrix- fracture transmissibilities as described in "Dual Porosity".

Matrix discretization (NMATOPTS)

When the `NMATOPTS` option is invoked, the matrix subgrid cell properties are copied from the outer primary matrix grid cells. Only the permeabilities and porosities of the primary matrix grid cells are used to generate the matrix subgrid cells tranmissibilities and pore volumes. The pore volume and transmissibility calculation is described in the "Discretized matrix model (ECLIPSE 100)" on page 263. In addition the ECLIPSE 300 model can be used with an UNIFORM partitioning. The pore volumes are evenly distributed between the matrix subgrid cells and the transmissibilities are calculated following a 1D computation where the distance DX of the matrix cells are assumed to be partitioned as DX / NMATRIX.

Figure 43.2 Structure of the matrix discretization and nomenclature

In the figure above the uniform partitioning is used with NMATRIX=4. The pore volume of the primary matrix cell is divided into

$$PV1 = PV2 = PV3 = PV4 = \frac{PVM}{4}$$

The transmissibility of the primary matrix-fracture connection is used to compute the matrix subgrid cells transmissibilities as

$$TR1 = TR2 = TR3 = TR4 = 2TRM \times 4$$

Coal in multiporosity

Using the NMATOPTS matrix discretization feature with coal or shale gas the definition of coal volumes needs to be considered, see "The Coal Bed Methane model" and for the partitioning into the matrix subgrid cells see "Shale Gas" on page 921. Only the input of the primary matrix subgrid cells are used to generate the subcells coal and pore volumes.

Triple Porosity

When the `TRPLPORO` keyword is used, only 3 or 4 porosities are allowed. The matrix and fracture porosities are specified as porosities 1 and 2 as usual. Porosity 3 is used to represent vugs (larger voids in the rock which are not part of the fracture system) which are connected to the matrix in the same way as the fracture is connected to the matrix, and additionally to the fracture through micro fracture connections. If used, porosity 4 represents additional vugs with connection to the matrix only (isolated vugs).

Unlike the standard multiporosity connections, a porosity in triple porosity may be connected to more than 2 other porosities, as the matrix can be connected to the fracture and two different vug porosities (isolated and connected). There is also a cyclic path in the porosity system, formed by the matrix, fracture and connected vug porosities. These relationships are illustrated in the figure above. Note that the logical order, with the fracture as the outer porosity, is shown, but in the grid description the fracture is still the second porosity.

As for the multiporosity model, the DUALPORO and, where needed, the DUALPERM, keywords are required. The input allows for the possibility of a triple permeability model, where the vug porosities form a connected continuum. However, in most if not all cases, only a dual permeability model will be required for triple porosity. To make the vug cells connected only to the local matrix and, for connected vugs, to the local fracture, you should add permeability or transmissibility multipliers of size zero in the z-direction for the vug porosities.

Figure 43.3 Triple porosity connectivity

Note NMATRIX and NMATOPTS should not be used with TRPLPORO.

Transmissibilities

In the current version of the input for this model, transmissibilities between vugs porosities and the matrix and fracture porosities are still described by Kazemi-type multipliers using the SIGMA and SIGMAV keywords. However, the LTOSIGMA keywords cannot be used and the interpretation of the SIGMA multiplier as an inverse squared matrix length is not applicable. In particular the connection between fracture and connected vugs is through micro fractures, and the connection between the matrix and vugs is through the typical inter-vug distance in the matrix and the size of the vugs. The SIGMA values used for all connections to vugs, whether through micro fractures from the fracture, or through matrix rock from the matrix, should honor this difference from standard matrix-fracture connection values.

An exact Kazemi-type equivalent to the LTOSIGMA input cannot be provided. However, for matrix-vug sigmas, a useful rule of thumb is that the product $V\sigma$ can be replaced by $\beta A/L$ where A is the cross-sectional area of the vug, L is half the distance between two vugs, and β is the fraction of total volume occupied by the vugs. There is no simple formula to represent the connected-vug/fracture sigma, and you should consider general transmissibility theory for the micro fractures characterizing this connection.

Input values for SIGMAV are applied to porosities in the following way:

- matrix/fracture connection sigma values are input through the matrix cells;

- fracture/connected-vug sigma values are input through the fracture cells;
- matrix/connected-vug sigma values are input through the connected-vug cells (third porosity);
- matrix/isolated-vug sigma values are input through the isolated-vug cells (fourth porosity, if present).

PROPS, REGION and SOLUTION Sections

This follows the input as for a standard single or dual porosity model. However, simulation grid cell data must be specified on a cell-by-cell basis, regardless of the use of NMATOPTS. Only data entered in the grid section will be processed by the NMATOPTS keyword. Examples of user data that needs to be specified for the entire simulation grid are regional data such as SATNUM, EQLNUM and FIPNUM.

Output

Summary output follows the usual syntax, where block solutions can be obtained by specifying a cell index corresponding to the matrix subcell.

On the restart file the subcell matrix solutions are only output if a flexible restart file is requested.

Output to the PRT file using the RPTGRID, RPTSOL, RPTSCHED keywords will output the requested quantity for all cells.

Examples

The following data sets come with the ECLIPSE installation:

Table 43.1 Multi porosity example files

File	Description
`MPORO.DATA`	Multi porosity example, gas-water case
`SHALEGAS1.DATA`	Shale gas

Limitations

The multi porosity option can currently not be used with:

1 Coarsening

2 Block to block connections (BTOBALFA or BTOBALFV)

ECLIPSE 100

3 The triple porosity option can currently not be used with parallel in ECLIPSE 100.

Note The restriction on using LGRs with multi- or triple-porosity has been removed.

Multi-segment Wells

Chapter 44

Introduction

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL

The Multi-segment Well model is a Special Extension, which is available in both ECLIPSE 100 and ECLIPSE 300. It provides a detailed description of fluid flow in the well bore. The facility is specifically designed for horizontal and multi-lateral wells, although it can of course be used to provide a more detailed analysis of fluid flow in standard vertical and deviated wells. Like the standard well model, the equations are solved fully implicitly and simultaneously with the reservoir equations, to provide stability and to ensure that operating targets are met exactly. A description of the model, with two example applications, is published in [Ref. 82].

The detailed description of the fluid flowing conditions within the well is obtained by dividing the well bore (and any lateral branches) into a number of 1-dimensional segments. Each segment has its own set of independent variables to describe the local fluid conditions. In ECLIPSE 100 there are four variables per segment: the fluid pressure, the total flow rate and the flowing fractions of water and gas. In ECLIPSE 300 there are $N_c + 3$ variables per segment (where N_c is the number of hydrocarbon components): the fluid pressure, the total molar flow rate and the flowing molar densities of water and each hydrocarbon component. Each segment may have completions in one or more reservoir grid blocks, or none at all if there are no perforations in that location. The variables within each segment are evaluated by solving material balance equations for each phase or component and a pressure drop equation that takes into account the local hydrostatic, friction and acceleration pressure gradients. The pressure drop may be calculated from a homogeneous flow model where all the phases flow with the same velocity, or a ‘drift flux’ model that allows slip between the phases. Alternatively, the pressure drop may optionally be derived from pre-calculated VFP tables, which can potentially offer greater accuracy and provide the ability to model chokes.

Benefits

The Multi-segment Well model has all the capabilities of the Wellbore Friction option, and provides several additional features and greater flexibility. The benefits of using this well model include:

1. Improved handling of multi-lateral topology. Data input has been designed for ease of use with multi-lateral wells. There are two separate keywords to define the segment structure and the perforated lengths. Individual branches are identified by their branch number. Branches may, if required, have sub-branches.
2. Improved modeling of multi-phase flow. Use of the drift flux model or pre-calculated pressure drop tables can produce more accurate pressure gradients than the homogeneous flow treatment used in the other well models. The pressure gradient can vary from segment to segment throughout the well; it is calculated fully implicitly in each segment using the local flowing conditions.
3. Flexibility. The variable number of grid block completions per segment allows the choice between using many segments for greater accuracy or fewer segments for faster computation.
4. The Multi-segment Well model contains several facilities for modeling advanced wells. Specific segments can be configured to model flow control devices such as chokes, by providing a pressure drop table that describes their pressure loss characteristics as a function of flow. Various ‘built in’ flow control device models are also provided:
    a. Variable pressure loss multipliers can be used to represent ‘smart’ devices which react to isolate high GOR or WOR regions.
    b. Flow limiting valves can also be used to represent ‘smart’ devices which react to limit the flow of oil, water or gas through a segment.
    c. Other advanced well components such as sub-critical valves, ‘labyrinth’ devices and down-hole separators can also be modeled.
5. Crossflow can be modeled more realistically, as the fluid mixture can vary throughout the well. Complex crossflow regimes may potentially occur in multi-lateral wells, including branch-to-branch crossflow and crossflow within individual branches. In the other well models, if crossflow occurred, the mixture flowing back into the reservoir would reflect the average contents of the well bore, regardless of which parts of the well were crossflowing.
6. Wellbore storage effects can be modeled more accurately by dividing the well bore into several segments. The drift flux model allows phases to flow in opposite directions at low flow rates, so that phase redistribution within the well bore can occur during shut-in well tests.

The multi-segment well model

Segment structure

A multi-segment well can be considered as a collection of segments arranged in a gathering tree topology, similar to the node-branch structure of a network in the Network option. A single-bore well will, of course, just consist of a series of segments arranged in sequence along the well bore. A multi-lateral well has a series of segments along its main stem, and each lateral branch consists of a series of one or more segments that connects at one end to a segment on the main stem (Figure 44.1). It is possible, if required, for lateral branches to have sub-branches; this may be useful when modeling certain inflow control devices as part of the network of segments.

The segment network for each well may thus have any number of 'generations' of branches and sub-branches, but it must conform to gathering-tree topology. Segments within the gathering tree topology can be connected in order to form loops - this facility is described in "Looped flowpaths" on page 642.

Figure 44.1 A multi-lateral, multi-segment well

Each segment consists of a **node** and a **flowpath** to its parent segment's node. (We use the term 'flowpath' here rather than 'branch', as the latter term is reserved for lateral branches in multi-lateral wells, each of which may contain several segments.) A segment's node is positioned at the end that is furthest away from the wellhead (Figure 44.2). Each node lies at a specified depth, and has a nodal pressure which is determined by the well model calculation. Each segment also has a specified length, diameter, roughness, area and volume. The volume is used for wellbore storage calculations, while the other attributes are properties of its flowpath and are used in the friction and acceleration pressure loss calculations. Also associated with each segment's flowpath are the flow rates of oil, water and gas, which are determined by the well model calculation.

Note The top segment is a special case. It does not have a parent segment so no pressure drop is calculated along this segment's flowpath. Hence there is no need to specify diameter, roughness and area for this segment. The equation for the pressure drop along the flowpath is replaced by a boundary condition appropriate to the well's mode of control. See "The top segment" on page 641 for details.

Figure 44.2 Well segments

A segment node must be positioned at each branch junction. Flow from the formation through grid-block-to-well connections also enters the well at segment nodes. A segment (or to be more precise, its node) can accept flow from any number of grid block connections (Figure 44.3). If you wish to have a separate segment node for each connection (for greatest accuracy) then it is best to position the nodes to lie at the center of their corresponding grid block connections. But it is also possible to reduce the number of segments by allocating two or more grid block connections to each segment. In this case it best to position each segment node to lie at the center of the perforated interval that is allocated to the segment.

ECLIPSE allocates each grid block connection to the segment whose node lies nearest to it. If necessary, you can over-ride this by explicitly specifying the segment to which the connection should be allocated in the `COMPSEGS` (or `COMPSEGL`) keyword.

Figure 44.3 Allocating connection flows to segments

It is also possible to position segment nodes at intermediate points along the well bore, for example where the tubing properties or inclination angle change. Additional segments can be defined to represent chokes or inflow control devices. In common with other aspects of numerical simulation, the optimum number of segments required to model a particular well depends on your preferred compromise between speed and accuracy.

Inflow performance

The flow of fluid between a grid block and its associated segment's node is given by the inflow performance relationship

$$q_{pj} = T_{wj} M_{pj} (P_j + H_{cj} - P_n - H_{nc}) \qquad \text{[EQ 44.1]}$$

where

q_{pj} is the volumetric flow rate of phase p in connection j at stock tank conditions in ECLIPSE 100 or reservoir conditions in ECLIPSE 300. The flow is taken as positive from the formation into the well, and negative from the well into the formation.

T_{wj} is the connection transmissibility factor.

M_{pj} is the phase mobility at the connection. If the node is injecting fluid into the formation, the ratio of the phase mobilities reflects the mixture of phases in the segment.

P_j is the pressure in the grid block containing the connection.

H_{cj} is the hydrostatic pressure head between the connection's depth and the center depth of the grid block.

P_n is the pressure at the associated segment's node n.

H_{nc} is the hydrostatic pressure head between the segment node n and the connection's depth.

Note the two components of the hydrostatic head, H_{cj} and H_{nc}. In multi-segment wells the connection depths are not forced to be equal to the grid block center depths as they are in wells (Figure 44.4). They may be set independently in keyword `COMPSEGS` (or `COMPSEGL`), or defaulted to allow ECLIPSE to calculate them from the segment depths and their position within the segment. The hydrostatic head H_{nc} between a connection and its corresponding segment node is calculated implicitly in the usual way from the density of the mixture in the segment. The hydrostatic head H_{cj} between the connection and the grid block center is calculated from an average of the fluid densities in the grid block.

ECLIPSE 100

The average is weighted according to the fluid relative permeabilities, to give a smooth variation as the fluid in the block is displaced by another phase. This average density is calculated explicitly at the beginning of each time step. If the use of explicit quantities causes oscillations from one time step to the next, they can be damped by setting the 63rd switch of the `OPTIONS` keyword, which averages the head term with its value at the previous time step.

ECLIPSE 300

The average is weighted according to the implicit fluid saturations, to give a smooth variation as the fluid in the block is displaced by another phase.

Figure 44.4 Hydrostatic head components

The pressure loss calculation

The following options are available for calculating the pressure drop:

1 Using a built-in homogeneous flow model.

This assumes there is no slip between the phases. The hydrostatic term depends on the density of the fluid mixture within the segment, which is simply a flow-weighted average of the phase densities. The frictional pressure drop calculation is the same as the method used by the Wellbore Friction option (see "Calculating the frictional pressure loss" on page 1163). The acceleration pressure drop across the segment is the difference between the velocity heads of the inflowing and outflowing mixture (see "Acceleration pressure loss" on page 655).

2 Interpolating the pressure drop from pre-calculated VFP tables.

VFP tables can be constructed (using, for example, the program VFP*i*) to describe the pressure drop along a certain length of tubing at the appropriate angle of inclination. The pressure drop along a segment is interpolated from the respective VFP table and can be scaled according to the length or depth span of the segment. Optionally, the table can be constructed to contain just the friction pressure drop, and ECLIPSE computes the hydrostatic pressure drop from the properties of the fluid within the segment (an option useful for undulating sections of tubing). VFP tables can also be used to describe pressure losses across chokes (the program VFP*i* contains the Sachdeva choke model).

3 Using a built-in ‘drift flux’ slip model.

This is a simple correlation that is continuous across the range of flowing conditions. It includes the effects of slip as two components: a local vertical slip velocity superimposed on a velocity ratio resulting from the profiles of holdup fraction and flow velocity across the tubing cross-section. The local vertical slip velocity will produce counter-current flow (phases flowing in opposite directions) when the mixture velocity is small enough.

4 Using a built-in model for a specific flow control device.

There is a built-in model to represent sub-critical flow through a valve with a specified throat cross-section area. See keyword `WSEGVALV` for further details.

There is a built-in ‘flow limiting valve’ model, which is a hypothetical device that reacts dynamically to limit the flow rate of oil, water or gas (at surface conditions) through a segment to a specified maximum value, by sharply increasing the frictional pressure drop across the segment if the limit is exceeded. See keyword `WSEGFLIM` for further details.

There is also a built-in model that corresponds to a ‘labyrinth’ flow control device, which may be used to control the inflow profile along a horizontal well or branch by imposing an additional pressure drop between the sand face and the tubing. See keyword `WSEGLABY` for further details.

Another built-in model calculates the pressure drop across an inflow control device that is placed around a section of the tubing and diverts the inflowing fluid from the adjacent part of the formation through a sandscreen and then into a spiral constriction before it enters the tubing. See keyword `WSEGSICD` for further details.

There is also a built-in model that corresponds to an ‘autonomous’ inflow control device. This works in a similar way to the ‘spiral’ inflow control device. See keyword `WSEGAICD` for further details.

The homogeneous flow model is the default method; it is simple to compute, and gives results similar to the Wellbore Friction option (when there is no crossflow). VFP tables provide the most flexible, and potentially the most accurate, method of calculating pressure losses. They can be constructed (for example with VFP*i*) using an appropriate choice of multi-phase flow correlation. The drift flux model is less accurate than using properly calculated VFP tables, because of the need for simplicity and continuity in the built-in correlation. The main reason for its presence is to provide a means of modeling phase separation and counter-current flow at low flow velocities, such as in shut-in well tests. Various parameters in the drift flux model that govern the slip velocity can be adjusted with the keyword `WSEGFMOD`.

The frictional pressure loss across individual segments can be adjusted by specifying a scaling factor with the keyword `WSEGMULT`. Optionally, the scaling factor can be defined to vary with the water-oil ratio and gas-oil ratio of the mixture flowing through the segment. A variable multiplier may be applied, for example, to model the operation of an adjustable flow control device whose purpose is to choke back production from high WOR or GOR regions.

The top segment

The top segment of the well is special. Its node corresponds to the well's bottom hole reference depth, so that the top segment's pressure is the same as the well's BHP. Since there is no parent segment above it, no pressure losses are calculated across the top segment. Its pressure loss equation is replaced by a boundary condition appropriate to the well's mode of control. The top segment may, however, have a volume, which is used in wellbore storage calculations.

It is natural to consider whether there is any advantage in modeling the whole well bore in segments up to the tubing head, in which case the well's reported BHP is really its THP. In general, this is not recommended. It is best to position the top segment just above the perforations and any lateral branch junctions, and to model the pressure loss between this point and the tubing head in the standard way with a VFP table. One reason is that, if the well cannot operate at a fixed THP because its IPR lies below the bottom of the 'U' shaped VFP curve, the standard logic in ECLIPSE recognizes this situation and shuts the well in cleanly; whereas if the whole well were modeled as a series of segments up to the tubing head, the multi-segment well model may simply not converge.

A situation where it may be useful to model the whole well bore as a series of segments is when a well test is being simulated and you need to model wellbore storage effects in detail. Here the drift flux model could be applied to model the process of phase separation in the well bore during a shut-in test.

External sources and sinks

It is possible to import water or gas into a segment from a source that is external to the reservoir grid. The import rate may be defined either as a constant value or as a function of the segment's pressure (see keyword `WSEGEXSS`). The imported fluid is added to fluid flowing through the segment. There are a number of possible applications for this facility. Gas import may be used to model the effects of lift gas injection, to examine the kick-off process for example. Water import may be used to take account of the water used to power a down-hole hydraulic pump if this is injected into the wellbore.

ECLIPSE 300

There is also an option to remove fluid from a segment into an external sink, at a rate that is a function of the segment's pressure. This facility is intended for use with the ECLIPSE 300 Thermal option to model recirculating wells.

Secondary Wells

ECLIPSE 300

As an extension to the facilities afforded by the keyword WSEGEXSS, further controls may be exerted over well segments by defining one or more segments to be secondary wells. This is accomplished using the keyword WSEGWELL which associates a well segment with a secondary well name. Once this association has been established, the secondary well keywords SCONPROD, SCONINJE and STEST may be employed to control the production from and injection into the segments. The keywords WSEGEXSS and WSEGWELL are mutually exclusive for a given well segment. With the exception of the top segment, any segment in the well topology may be associated with a secondary well name.

There are a number of possible applications for this facility. For example, a dual-tubing well may be modeled in terms of a multi-segment well where the conventional (primary) well can be used to represent one tubing and a well segment can be used to represent the other tubing by defining this as a secondary well. There are, however, a number of factors to consider when constructing this form of model. For example, if the primary well shuts, the secondary well will also shut. Moreover, the multi-segment well topology can become complex if both secondary producers and injectors are modeled via secondary wells. In addition, the controls available with a secondary well are more limited than those available with a conventional well.

Looped flowpaths

The gathering tree topology required by multi-segmented wells normally prevents the formation of looped flowpaths within the well itself. However, it is possible to add looped flowpaths as follows:

- The basic topology of the well must be specified in the normal manner using the WELSEGS keyword. One or more pairs of nodes of different segments must be made physically coincident. (In practice, this just means that the segment nodes should be specified as being at the same measured depth in the WELSEGS keyword.) Note that this in itself is not sufficient to form a looped flowpath and these node pairs will still act as though they are unconnected in any way.
- Each such pair of segment nodes must be "linked" using the WSEGLINK keyword. This is the key step that instructs each pair of segment nodes to communicate and form looped flow-paths. At least one of these segments must be at the end of a branch.

Figure 44.5 Simple looped multi-segmented well

A pair of linked segment nodes is shown in Figure 44.5. This link forms a looped flowpath between the upper and lower branch in this multi-segmented well. Flow from the lower branch can pass into the main stem of the well directly or via the upper branch, and flow from the upper branch can pass into the main stem directly or via the lower branch. The pressure variations in the upper and lower branches will determine the direction of the flow.

The pressure drop equations for the segments whose nodes are linked are unaffected, but the material balance equations for one of the segments are replaced instead by equations that enforce equality of the solution (pressure, flowing fraction of water and flowing fraction of gas) at the linked segment nodes.

Handling of linked segment properties

In the `WSEGLINK` keyword, the first segment of each pair specified is known as the **primary linked segment**, while the second segment of the pair is known as the **secondary linked segment**. (If the first segment of the pair is at the end of a branch, but the second is not, then the first segment will be treated as the secondary link segment instead.) It is the secondary linked segment that has its material balance equations replaced by the solution equality equations.

- If connection flows or external source /sink flows are specified for both the linked segments (using either the `COMPSEGS` or `COMPSEGL` keywords), then these flows will all be assigned to the primary linked segment (as it is the only one of the pair that has the material balance equations which incorporate these terms).
- Any volume associated with the secondary linked segment will be allocated as additional volume in the primary linked segment.
- Apart from the replacement of the material balance equations by linked segment node solution equality equations in the secondary linked segment, each of these segments is exactly the same as any other segment used to define a multi-segmented well. Hence the pressure drop equation in either of the linked segments can be modified in any of the ways described in "The pressure loss calculation" on page 640.

Example application: Modeling an unpacked annulus

As an example of an application of looped flowpaths within multi-segmented wells, consider the modelling of flow in an unpacked annulus.

One branch of the well can be used to model the tubing and may contain multiple segments, each in direct communication with its immediate neighbors. This tubing branch is shown as a dashed line in Figure 44.6.

Another branch of the well can be used to model the annulus, the segments on this branch are also in direct communication with their immediate neighbors. This annulus branch is shown as a dotted line in Figure 44.6. Flow only enters the well via connections at annulus segments.

Flow can move freely along the annulus and pass into the tubing via control valves at intervals along the well. Without the ability to model looped flowpaths, the flow from a certain annulus segment would have been constrained so that it would have to flow into the tubing via a specific valve. But due to the presence of looped flowpaths, flow in an annulus segment is free to pass through any of the valves into the tubing.

Figure 44.6 Schematic of a multi-segmented well with an unpacked annulus

Impact of looped flowpaths on the well linear equations

From [Ref. 82], in the absence of such linked segment nodes, the linear system arising from the well equations can be solved very efficiently due to the special structure of the well matrix. However, as soon as loops are added, this special structure is lost and an alternative method for solving the well system must be used. ECLIPSE uses an iterative method called GMRES [Ref. 51] to solve the well linear system arising from the presence of loops, with an incomplete LU factorization pre-conditioner used to accelerate the convergence. If required, the parameters that control the GMRES solver and the pre-conditioner can be modified using the `WSEGSOLV` keyword.

Restrictions

1. One of the segments linked using the `WSEGLINK` keyword must be at the end of a branch.
2. It is not permissible for the secondary linked segment to have zero effective absolute roughness. That is, item 8 in the `WELSEGS` record defining the secondary linked segment properties must be non-zero.
3. A segment cannot be both a secondary link segment and a labyrinth inflow control device with non-positive configuration number (specified using the `WSEGLABY` keyword). If the need for this particular situation arises, the specification of the primary and secondary link segments can be reversed in the `WSEGLINK` keyword.
4. The top segment of the well cannot be part of a loop so it cannot appear in a row of the `WSEGLINK` keyword.

Pressure drops from VFP tables

VFP tables offer the most flexible, and potentially the most accurate, means of determining the pressure drop across each segment. Interpolating the pressure drop from a table is also considerably faster than calculating it from a multi-phase flow correlation. It is not necessary to supply a separate table for each segment. Segments of similar diameter, inclination and roughness can share the same table; ECLIPSE scales the pressure drop according to the length or depth span of each segment.

VFP tables should be constructed (by, for example, the program VFP*i*) for a representative length of tubing at the appropriate angle of inclination, using a suitable multi-phase flow correlation. The tables should be in the form of production well VFP tables, and are entered in the `SCHEDULE` section with the keyword `VFPPROD`. Standard VFP tables give the BHP as a function of the flow rate, the THP, the water and gas fractions, and optionally the ALQ. When applied to a well segment, the BHP should be interpreted as the segment's nodal pressure (which is its inlet pressure), and the THP should be interpreted as the pressure at the node of its neighboring segment towards the wellhead (that is the segment's outlet pressure). This is similar to the interpretation used for VFP tables constructed to model branch pressure drops in the Network option.

Pressure drop scaling

The second item in the first record of a VFP table (see keyword `VFPPROD`) normally contains the bottom hole datum depth of the table. However, when the table is used to supply segment pressure drops, it should contain either the length or the depth span (bottom depth - top depth) of the representative length of tubing for which the table was calculated. When ECLIPSE interpolates a segment's pressure drop from the table, it will scale the pressure drop in proportion to the segment's length or depth span.

Instructions to scale the pressure drop according to the segment's length or depth span are entered in the seventh item of keyword `WSEGTABL`. The choice should be consistent with the quantity supplied in the second item in the first record of the VFP table. The choice of scaling quantity will only make a difference if the same table is to be used for segments having a range of inclinations. The friction pressure drop is proportional to the length, while the hydrostatic pressure drop is proportional to the depth span; the choice should be based on whichever is the dominant component of the overall pressure drop. Obviously, horizontal segments should be scaled according to length.

There is also the option not to scale the interpolated pressure drop at all, irrespective of the length or depth span of the segment. This choice would be appropriate when the segment represents a choke, since the pressure drop across a choke does not vary in proportion to the overall length of the device.

As well as the automatic scaling according to the segment's length or depth span, a user-defined scaling factor can be applied to the interpolated pressure drop. This is defined in keyword `WSEGMULT`. A typical use of this would be to inhibit manually the flow through a segment representing a flow control device by increasing the pressure drop across it. Optionally, the scaling factor can be defined to vary with the water-oil ratio and gas-oil ratio of the mixture flowing through the segment, for example to model the operation of an adjustable flow control device whose purpose is to choke back production from high WOR or GOR regions. Alternatively the variation with WOR and GOR could be built directly into the VFP table.

Pressure drop components

VFP tables normally describe the combined effect of friction and hydrostatic pressure losses along the representative length of tubing. The tables do not supply the information necessary to resolve the combined pressure drop into these separate components. Accordingly, the combined pressure drop obtained from a VFP table is reported as the friction pressure drop; the hydrostatic pressure loss is reported as zero.

The tables also usually include an acceleration pressure loss due to expansion of the gas phase in its passage down the pressure gradient. However, they cannot take into account any acceleration pressure loss caused by additional fluid entering the segment (for example through the perforations). But ECLIPSE calculates the total acceleration pressure loss and add it to the overall pressure loss interpolated from the table, if this is requested in Item 6 in Record 1 of keyword `WELSEGS`. Ideally, in this case, the VFP tables ought to be constructed without including any acceleration pressure loss, or the component due to gas expansion is accounted for twice. The VFP*i* program has a switch in the Calculation Control panel to turn off the acceleration pressure loss calculation. The acceleration component, however, is generally small in comparison with the overall pressure drop, and in the majority of cases can be neglected.

There is an option to supply VFP tables containing the friction pressure drop only, and have ECLIPSE calculate the hydrostatic pressure drop across the segment and add it to the interpolated pressure drop. Friction-only pressure drop tables can be constructed, of course, by defining the representative length of tubing used for calculating the table to be horizontal. This option is useful to apply to gently undulating sections of tubing, to overcome the need for separate tables for each inclination angle. The hydrostatic pressure drop calculated by ECLIPSE is based on a homogeneous flow model, so slip effects are neglected.

Reverse flow

A complication arises when fluid is flowing through the segment in the opposite direction expected for a production well, that is, away from the well head. The sign of the flow is then negative. But in production wells this situation may arise only when crossflow occurs, and the flow velocities are not likely to be large. If reverse flow is likely to occur, it is possible to extend the VFP table into the negative flow region. VFP*i* can produce tables whose first few flow values are negative, if requested. The table must not, however, have a point at zero flow, since the water and gas fraction values are singular there.

There are three options for handling cases where the flow is less than (or more negative than) the first flow value in the VFP table. The option for each segment is set in the sixth item in keyword `WSEGTABL`.

1. Reverse the flow, look up the VFP table, then reverse the pressure drop.

   This option is recommended for use in horizontal segments or segments representing chokes, or with ‘friction only’ VFP tables (see "Pressure drop components" on page 646), in which the table’s pressure drop has no hydrostatic component. In these cases the pressure drop can simply be reversed when the flow reverses. The first flow value in the VFP table should be positive. For negative flows, the ‘mirror image’ of the table is used. For flows in between + and - the first value, the pressure loss is linearly interpolated between these two points.

2. Fix the lookup value of the flow rate at the first flow point in the table.

This option is recommended for use in vertical or inclined segments in which the friction pressure loss in reverse flow is expected to be small compared with the hydrostatic pressure loss. (The 'reverse flow' option above is not suitable in these cases as the hydrostatic pressure loss does not reverse when the flow reverses.) If the first flow value in the VFP table is positive, it should be low enough to give a negligible friction pressure drop, so that the calculated friction pressure drop will remain negligible during reverse flow.

3 Extrapolate the pressure drop linearly with the flow.

This should be used with caution, as substantial extrapolation (especially into the negative flow region) may result in stability problems.

Chokes

VFP tables can also be constructed to model the pressure loss across flow control devices such as chokes. For example, VFP*i* can produce a table based on the Sachdeva choke model, incorporating both critical and sub-critical flow regimes. The effects of varying the choke diameter can also be incorporated into the table, by using the ALQ variable. VFP*i* can create a table of pressure losses across several different choke diameters, with the diameters represented by the ALQ variable in the appropriate units. The selected ALQ definition for this purpose should be '`BEAN`'. It is then possible to model the effects of varying the choke diameter at any time in the simulation by changing the look-up ALQ value of the VFP table (the eighth item in keyword `WSEGTABL`).

Segments representing chokes should **not** have their pressure losses scaled according to length or depth span (`WSEGTABL` item 7), and the most appropriate option for dealing with reverse flow is to reverse the pressure loss (`WSEGTABL` item 6).

Injection wells

Injection wells would normally have single-phase water or gas flowing through their segments, so there should be less need to use VFP tables to model the pressure losses. But multi-phase flow can occur if there is crossflow with reservoir fluid entering at one or more perforated intervals. If you wish to use VFP tables to model the pressure loss across segments in an injection well, the possibility of multi-phase flow requires the VFP tables to be of the **production** type (keyword `VFPPROD`). They should therefore be prepared as for a production well, but with negative flow values since the predominant direction of flow is away from the well head.

The drift flux slip model

Introduction

A slip model enables the phases to flow with different velocities. In general, in all but downward flow, the gas phase will flow with a greater velocity than the liquid phase. Slip thus increases the liquid holdup fraction for a given flowing gas-liquid ratio, thereby increasing the hydrostatic head component.

The program VFP*i* contains several one-dimensional slip correlations which it uses to compute pressure losses in the tubing. These, however, are not suitable for implementing directly in ECLIPSE. A model suitable for use in multi-segment wells ought to have the following attributes:

Simplicity

The calculation has to be done many times in each segment at every time step. We should look for a model that offers rapid and efficient calculation.

Continuity

The model must cover the complete range of flowing conditions without any first order discontinuities. Any such discontinuities would prevent the iterations of the well solution from converging. This requirement rules out methods involving flow regime maps, because the calculated pressure gradient changes discontinuously from one flow regime to another, unless some form of smoothing is applied.

Differentiability

The fully implicit solution of the Multi-segment Well model requires the calculation of derivatives of the phase flow rates and the pressure drop for the Jacobian matrix. Ideally these derivatives should be calculated analytically, as calculating numerical derivatives with respect to each main variable would be costly in CPU time.

In addition to these requirements, it would be useful for the model to work in the countercurrent flow regime, allowing the heavy and light phases to flow in opposite directions when the overall flow velocity is small. This would enable ECLIPSE to model the separation of phases within the well bore that occurs when a well is shut-in at the surface, and also the accumulation of water in undulating sections of a horizontal well.

Model formulation for gas-liquid slip

Drift flux models express the gas-liquid slip as a combination of two mechanisms. The first mechanism results from the non-uniform distribution of gas across the cross-section of the pipe, and the velocity profile across the pipe. The concentration of gas in the gas-liquid mixture tends to be greater nearer the center of the pipe and smaller near the pipe wall. The local flow velocity of the mixture is also greatest at the center of the pipe. Thus, when integrated across the area of the pipe, the average velocity of the gas tends to be greater than that of the liquid. The second mechanism results from the tendency of gas to rise vertically through the liquid due to buoyancy.

A formulation that combines the two mechanisms is

$$v_g = C_0\, j + v_d \qquad \text{[EQ 44.2]}$$

where

v_g is the flow velocity of the gas phase, averaged across the pipe area

C_0 is the profile parameter (or distribution coefficient) resulting from the velocity and gas concentration profiles

j is the volumetric flux of the mixture

v_d is the drift velocity of the gas.

The volumetric flux of the mixture is same as the average mixture velocity, and is the sum of the gas and liquid superficial velocities

$$j = v_{sg} + v_{sl} = \alpha_g v_g + (1 - \alpha_g) v_l \qquad \text{[EQ 44.3]}$$

where α_g is the gas volume fraction, averaged across the pipe area.

The average liquid flow velocity is thus

$$v_l = \frac{1 - \alpha_g C_0}{1 - \alpha_g} j - \frac{\alpha_g}{1 - \alpha_g} v_d \qquad \text{[EQ 44.4]}$$

Several of the one-dimensional slip correlations take the general form of [EQ 44.2]. The task here is to develop relationships for C_0 and v_d that are simple, continuous and differentiable, and which extend into the countercurrent flow regime. The derivation of these relationships is described in [Ref. 29]. The relationships contain some adjustable parameters that can be tuned to fit observations. In [Ref. 29] the parameter values are optimized to fit a series of experiments performed in 15 cm (6 in) diameter tubing. In [Ref. 30] the optimization process was improved and the model was extended to allow for the observation that the presence of free gas can disrupt the slip between oil and water, depending on the pipe inclination. The default values of the parameters in the model in ECLIPSE are those of the original implementation described in the subsequent sections of this chapter, but you may request ECLIPSE to use the optimized parameters from [Ref. 29] or [Ref. 30] with the keyword `WSEGDFMD`.

If you choose to apply the drift flux model to any segment in a well, ECLIPSE employs a different set of main variables for all the well's segments. In particular, the first variable is the volumetric flux j instead of the total flow rate.

The profile parameter

Zuber and Findlay [Ref. 25] have shown that the value of C_0 can range from 1.0 to 1.5. But a number of 1-D flow correlations take a value of 1.2 to apply in the bubble and slug flow regimes (for example, the models of Aziz, Govier and Fogarasi, Ansari et al. [Ref. 27], and Hasan and Kabir. Accordingly we set C_0 to a constant value of 1.2 at low values of α_g and j.

At high velocities, however, the profiles flatten out and C_0 approaches 1.0. Moreover, C_0 must approach 1.0 as α_g approaches 1.0. In fact the product $\alpha_g C_0$ must never exceed 1.0.

Accordingly we employ the following relation for C_0

$$C_0 = \frac{A}{[1+(A-1)\beta^{*2}]} \qquad \text{[EQ 44.5]}$$

The coefficient A is defaulted to 1.2, but can be set by you to any value not less than 1.0. The purpose of the term β^* is to make C_0 reduce to 1.0 at high values of α_g and j. It is evaluated from the formula

$$\beta^* = \frac{(\beta - B)}{(1-B)} \text{ subject to } 0 \le \beta^* \le 1 \qquad \text{[EQ 44.6]}$$

β is a parameter that approaches 1.0 as α_g approaches 1.0, and also as the mixture velocity approaches a particular value. We choose the velocity of the onset of the annular flow regime to be the velocity at which profile slip vanishes. The transition to annular flow occurs when the gas superficial velocity v_{sg} reaches the 'flooding' value that is sufficient to prevent the liquid from falling back against the gas flow, v_{sgf}. An expression for v_{sgf} is given in the next section, with [EQ 44.12]. We thus choose the following expression for β

$$\beta = \max\left(\alpha_g, F_v \frac{\alpha_g |j|}{v_{sgf}}\right) \qquad \text{[EQ 44.7]}$$

The parameters B and F_v are user-definable constants. B represents the value of the gas volume fraction or the flooding velocity fraction at which the profile parameter C_0 will start to reduce from the value A. It defaults to 0.3. F_v is a multiplier on the flooding velocity fraction. It defaults to 1.0, but the profile flattening can be made more or less sensitive to the velocity by respectively increasing or decreasing its value.

The default values for the parameters A and B can be changed by requesting either of the optimized parameter sets from [Ref. 29] or [Ref. 30] with the keyword `WSEGDFMD`. Note that in the `SHI-04` model there is no profile slip, $A = 1.0$, making the value of B irrelevant.

There is a limit on the maximum value that the parameter B may have. Intuitively we would expect the gas superficial velocity $v_{sg} = \alpha_g v_g$ to increase with both α_g and j. Thus we require

$$\frac{\partial}{\partial \alpha_g}(\alpha_g C_0) > 0 \text{ and } \frac{\partial}{\partial j}(j C_0) > 0 \qquad \text{[EQ 44.8]}$$

Both these criteria will be satisfied if

$$B < \frac{2-A}{A} \qquad \text{[EQ 44.9]}$$

With the default value of $A = 1.2$ the constraint is $B < 0.6667$.

The drift velocity

The derivation of the expression for the drift velocity v_d is described in [Ref. 29]. The expression is derived by combining data on the limits of counter-current flow made under a variety of flowing conditions and interpolating between them to avoid discontinuities. The method honors observations of gas-liquid relative velocities at low and high gas volume fractions, and joins them with a 'flooding curve'.

The rise velocity of gas through a stationary liquid is related to v_d by

$$v_g(v_l = 0) = \frac{v_d}{1 - \alpha_g C_0} \qquad \text{[EQ 44.10]}$$

As α_g approaches zero, the gas velocity should approach the rise velocity of a single bubble, for which Harmathy recommended the value of $1.53\ v_c$ where v_c is the characteristic velocity

$$v_c = \left[\frac{\sigma_{gl}\, g\, (\rho_l - \rho_g)}{\rho_l^{\,2}}\right]^{1/4} \qquad \text{[EQ 44.11]}$$

where σ_{gl} is the gas-liquid interfacial tension, g is the gravitational acceleration, ρ_l and ρ_g are the liquid and gas phase densities.

At high values of α_g, the gas velocity should approach the 'flooding velocity', which is just sufficient to support a thin annular film of liquid and prevent it from falling back against the gas flow. Wallis and Makkenchery obtained the relation

$$v_g(v_l = 0) = K_u \left(\frac{\rho_l}{\rho_g}\right)^{1/2} v_c \qquad \text{[EQ 44.12]}$$

where K_u is the 'critical Kutateladze number', which is related to the dimensionless pipe diameter

$$D^* = \left[\frac{g\,(\rho_l - \rho_g)}{\sigma_{gl}}\right]^{1/2} D \qquad \text{[EQ 44.13]}$$

by the function in Table 44.1.

Table 44.1 The relationship between the dimensionless pipe diameter and the critical Kutateladze number

D^*	K_u
≤ 2	0
4	1.0
10	2.1
14	2.5
20	2.8
28	3.0
≥ 50	3.2

To interpolate between these two extremes we make use of the flooding curve described by Wallis to define the limit of the counter-current flow regime. The curve must be ramped down at low gas fractions in order to match Harmathy's bubble rise velocity. We apply a linear ramp between two values of the gas volume fraction, a_1 and a_2. The resulting relation for the drift velocity is

$$v_d = \frac{(1-\alpha_g C_0)\, C_0 K(\alpha_g) v_c}{\sqrt{\frac{\rho_g}{\rho_l}}\, \alpha_g C_0 + 1 - \alpha_g C_0} \qquad \text{[EQ 44.14]}$$

where

$$K(\alpha_g) = 1.53/C_0 \quad \text{when } \alpha_g \le a_1$$

$$K(\alpha_g) = K_u \quad \text{when } \alpha_g \ge a_2$$

and a linear interpolation of these values when $a_1 < \alpha_g < a_2$. The default values of a_1 and a_2 are set to 0.2 and 0.4 respectively, to match data used by Zuber and Findlay to demonstrate the transition from the bubble flow regime. Their default values can be changed by requesting either of the optimized parameter sets from [Ref. 29] or [Ref. 30] with the keyword `WSEGDFMD`.

The formulation described above is intended to model the behavior of gas-liquid mixtures in a vertical pipe. To scale the drift velocity for inclined flow it is multiplied by an inclination factor

$$v_d(\theta) = m(\theta)\ \ v_d\big|_{\theta=0} \qquad \text{[EQ 44.15]}$$

where θ is the angle of inclination from the vertical and $v_d\big|_{\theta=0}$ is given by [EQ 44.14]. By default ECLIPSE uses the inclination factor given by Hasan and Kabir [Ref. 34]

$$m(\theta) = (\cos\theta)^{0.5}\,(1+\sin\theta)^2 \qquad \text{[EQ 44.16]}$$

Hasan and Kabir present the formula as valid for oil/water flow for $\theta < 70°$, but we shall apply this for all inclination angles. For horizontal segments the drift velocity becomes zero.

Item 2 of keyword `WSEGDFMD` allows you to override this default formula for the inclination factor. You may input your own table of m vs. θ with keyword `WSEGDFIN`. (Note that if you supply your own table it is not written to the `RESTART` file, so it must be present in all restarted data sets.) Alternatively, you can ask ECLIPSE to use a built-in table that reproduces the `SHI-03` model described in [Ref. 29]. This is the recommended option if you select the `SHI-03` model's parameter set in item 1 of keyword `WSEGDFMD`. Note that the built-in table has $m > 1$ for vertical pipes, since it was necessary to rescale v_d at all values of the inclination angle, including $\theta = 0$, to fit the experimental observations. If you opt for the `SHI-04` model described in [Ref. 30] you are recommended to select the corresponding relation for the inclination factor, which is

$$m(\theta) = 1.85\,(\cos\theta)^{0.21}\,(1+\sin\theta)^{0.95} \qquad \text{[EQ 44.17]}$$

Note that this also has $m > 1$ for vertical pipes.

Three-phase flow and oil-water slip

The drift flux model described above was formulated for two-phase gas-liquid mixtures. For three-phase mixtures flowing in the well bore, and for oil-water mixtures, we apply the following model.

For three-phase mixtures, we first use the above formulation to model the total slip between the gas phase and the combined liquid phase. The properties of the combined liquid phase (density, viscosity) are calculated as an average of the oil and water properties, weighted according to the volume fraction α of each phase. This treatment neglects any tendency of the oil-water mixture to form emulsions, but it is in common practice as a way of applying two-phase flow correlations to oil-water-gas mixtures in a well. The gas-liquid interfacial tension is calculated as

$$\sigma_{gl} = \frac{\alpha_o \sigma_{go} + \alpha_w \sigma_{wg}}{\alpha_o + \alpha_w} \qquad \text{[EQ 44.18]}$$

σ_{go} and σ_{wg} are the gas-oil and water-gas interfacial tensions. They can either be supplied as tabulated functions of pressure with the keywords STOG and STWG respectively (in ECLIPSE 100), or calculated internally from built-in correlations obtained from Beggs [Ref. 35]. In ECLIPSE 100 the reservoir fluid temperature is unknown and is arbitrarily set to 160 ^{o}F (71.1 ^{o}C) for these correlations. In ECLIPSE 300 the temperature is set to the temperature of the EOS region corresponding to the well. Then, having obtained the velocity of the combined liquid phase, we use an oil-water slip model to resolve the velocities of the oil and water phases.

By default, ECLIPSE uses the formulation of Hasan and Kabir [Ref. 34] to model the slip between oil and water in the liquid phase. This takes the form of

$$v_o = C_0' \, v_l + v_d' \qquad \text{[EQ 44.19]}$$

They suggest a value of 1.2 for the profile parameter, which becomes unity when oil is the continuous phase at $\alpha_o > 0.7$. We implement this as a continuous function of the holdup fraction of oil in the liquid phase, $\alpha_{ol} = \alpha_o / (\alpha_o + \alpha_w)$

$$\begin{aligned} C_0' &= A \quad \text{when } (\alpha_{ol} \le B1) \\ C_0' &= 1 \quad \text{when } (\alpha_{ol} \ge B2) \\ C_0' &= A - (A-1)\left(\frac{\alpha_{ol} - B1}{B2 - B1}\right) \quad \text{when } (B1 < \alpha_{ol} < B2) \end{aligned} \qquad \text{[EQ 44.20]}$$

The values of the coefficients default to $A = 1.2$, $B1 = 0.4$ and $B2 = 0.7$. A condition equivalent to [EQ 44.8], that the oil superficial velocity should increase with the oil holdup fraction, requires

$$B1 < (2 - A)\, B2 \qquad \text{[EQ 44.21]}$$

The oil-liquid drift velocity is calculated as

$$v_d' = 1.53 v_c' \, (1 - \alpha_{ol})^2 \qquad \text{[EQ 44.22]}$$

where

$$v_c' = \left[\frac{\sigma_{ow}\, g\, (\rho_w - \rho_o)}{\rho_w^{\,2}}\right]^{1/4} \qquad \text{[EQ 44.23]}$$

σ_{ow} is the oil-water interfacial tension. This can either be supplied as a tabulated function of pressure with the keyword STOW (in ECLIPSE 100) or calculated internally as the difference between the values of σ_{go} and σ_{wg} obtained from the above correlations. Hasan and Kabir state that the drift velocity should become zero when oil is the continuous phase, but we keep the form of [EQ 44.22] because when $\alpha_{ol} = 0.7$ the drift velocity will have diminished by an order of magnitude from its value at $\alpha_{ol} = 0$.

If you select the SHI-03 or SHI-04 model in item 1 of keyword WSEGDFMD, ECLIPSE will use a different formulation for oil-water slip based on the results described in [Ref. 29] or [Ref. 30] respectively. In these experiments no profile slip was observed between the oil and water phases, and accordingly the parameter A in [EQ 44.20] is set equal to 1.0. The parameters $B1$ and $B2$ are redundant since C_0' is always 1.0. In place of [EQ 44.22] the oil-liquid drift velocity is calculated as

$$v_d' = 1.53 v_c' (1 - \alpha_{ol})^{n'} \qquad \text{[EQ 44.24]}$$

where n' is set equal to 0.95 or 1.0 respectively.

The oil-water drift velocity in inclined segments is scaled in a similar way as described in the previous section for the gas-liquid drift velocity, using an inclination factor $m(\theta)$ as shown in [EQ 44.15]. By default ECLIPSE uses Hasan and Kabir's relation [Ref. 34] as shown in [EQ 44.16]. Alternatively you can request ECLIPSE in keyword WSEGDFMD to use a table of m vs. θ instead. You may then input your own inclination factor table with keyword WSEGDFIN or let ECLIPSE use its built-in table, which is optimized to fit the experimental results described in [Ref. 29]. This is the recommended option if you select the optimized set of model parameters in keyword WSEGDFMD. Note that the built-in table has different inclination factors for gas-liquid and oil-water drift velocities. If you opt for the SHI-04 model described in [Ref. 30] you are recommended to select the corresponding relation for the inclination factor, which is

$$m(\theta) = 1.07\cos\theta + 3.23\sin 2\theta - 2.32\sin 3\theta \qquad \text{[EQ 44.25]}$$

This relation is valid up to 88° from the vertical. Between 88° and 90° (horizontal) ECLIPSE will extrapolate $m(\theta)$ linearly to zero.

The effect of gas on oil-water slip

The presence of free gas entrained in the liquid can disrupt the slip between the oil and water phases, as [Ref. 30] concludes. It proposes that the inclination factor $m(\theta)$ for oil-water slip should be multiplied by a factor $m_g(\theta, \alpha_g)$ that describes this effect as a function of the pipe inclination and the gas volume fraction:

$$\begin{aligned} m_g(\theta, \alpha_g) &= 1 - \frac{\alpha_g}{a_3(\theta)} \quad \text{when} \quad \alpha_g < a_3 \\ m_g(\theta, \alpha_g) &= 0 \quad \text{when} \quad \alpha_g > a_3 \end{aligned} \qquad \text{[EQ 44.26]}$$

The oil-water drift velocity decreases to zero as the gas volume fraction increases from zero to the value a_3, and remains at zero at all higher values of α_g. The cut-off value a_3 is a function of the pipe inclination. At angles between 0 and 70 degrees from the vertical only a small amount of gas is required to eliminate the drift between oil and water, while for near horizontal flows gas has a much smaller effect (due to segregation from the liquid). The following equation approximates the observations of [Ref. 30]

$$a_3(\theta) = 0.017\exp(\theta_r^{3.28}) \qquad \text{[EQ 44.27]}$$

where θ_r is the angle from the vertical in radians $(= \pi\ \theta°/180)$.

You can include the gas effect in the calculation of the drift velocity by selecting the SHI-04 option in item 3 of keyword WSEGDFMD. You are recommended to select this option whenever you select the SHI-04 model in item 1. However, the SHI-04 gas effect option may be used with any of the drift flux models available in item 1, but note that it will not reduce the profile slip which is present in the ORIGINAL model where $A = 1.2$.

Friction

The calculation of the friction pressure loss is based on the formulation used in the correlation of Hagedorn and Brown [Ref. 36]

$$\delta P_f = \frac{C_f \; f \; L \; w^2}{A^2 \; D \; \rho} \qquad \text{[EQ 44.28]}$$

where

f is the Fanning friction factor

L is the length of the segment

w is the mass flow rate of the fluid mixture through the segment

A is the segment's area of cross-section for flow

D is the segment's diameter

ρ is the in-situ density of the fluid mixture

C_f is a units conversion constant

2.679E-15 (`METRIC`), 5.784E-14 (`FIELD`), 1.523E-13 (`LAB`), 2.644E-15 (`PVT-M`).

The Fanning friction factor is calculated using the same method as in the Wellbore Friction Option (see "Calculating the frictional pressure loss" on page 1163), but with the Reynolds number formulated as

$$R_e = \frac{C_r \; D \; w}{A \; \mu} \qquad \text{[EQ 44.29]}$$

C_r is a units conversion constant

0.01158 (`METRIC` and `PVT-M`), 0.01722 (`FIELD`), 0.02778 (`LAB`).

The viscosity μ is calculated as an average of the phase viscosities weighted according to their volume fraction.

The above formulation becomes equivalent to the homogeneous model used in the Wellbore Friction option when there is no slip between the phases.

Acceleration pressure loss

The acceleration pressure loss across a segment is the difference between the velocity head of the mixture flowing across the segment's outlet junction and the velocity heads of the mixture flowing through all its inlet junctions (its main inlet and any side branches connecting with the segment).

$$\delta P_a = H_{v\text{out}} - \sum_{\text{inlets}} H_{v\text{in}} \qquad \text{[EQ 44.30]}$$

The velocity head of the mixture flowing through a junction is

$$H_v = \frac{0.5 \; C_f \; w^2}{A^2 \; \rho} \qquad \text{[EQ 44.31]}$$

For the outlet junction flow, A is the cross-section area of the segment. For inlet junction flows, A is the maximum of the cross-section areas of the segment and the inlet segment. Thus the acceleration pressure loss will give the Bernoulli effect at restrictions but no pressure recovery at expansions. Note also that, by including the full velocity head of any inflow from side branches, we are assuming that the inflow is directed into the segment with no momentum loss.

Inflow of reservoir fluid into the segment through grid block connections is **not** included in the inflow velocity head. The acceleration pressure loss thus takes account of the need to accelerate this fluid from zero velocity to the segment's outflow velocity.

Down-hole separators (ECLIPSE 100)

Down-hole separators are available in the ECLIPSE 100 version of multi-segment wells.

A down-hole separator can separate water or free gas from the flowing mixture within the well, and send it along a lateral branch to reinject into the formation.

Defining a down-hole separator

A segment is designated as a down-hole separator with the keyword `WSEGSEP`. The segment must have three segments connecting with it:

1. An inlet segment, from which the wellstream enters the separator.
2. A water or gas offtake segment, which should be the beginning of the lateral branch that injects the separated fluid back into the formation.
3. An outlet segment, through which the remaining fluid flows to continue its journey towards the wellhead; this should normally be the separator segment's outlet segment on the same branch.

A down-hole pump is required to maintain the flow through the separator and re-inject the offtake fluid into the formation. Pumps may be placed in any segment except the separator itself, but the best location for the pump to control the separation process is the separator offtake segment, or another segment in the offtake branch (a 'pullthrough pump'). There is a choice of two methods for modeling the pump:

1. For a pullthrough pump for a down-hole water separator, there is a simple device model with an additional term to limit the carry-over of oil into the water offtake branch (keyword `WSEGPULL`).
2. Pumps can be modeled by VFP tables (keyword `WSEGTABL`). Note that a pump in the offtake branch should be modeled by a VFP table with negative flow values, since the flow is in the direction away from the wellhead.

Figure 44.7 shows schematically a multi-segment well with a down-hole separator extracting water to reinject into the formation. The segments show the direction of positive flow (towards the wellhead) to illustrate that the offtake branch has negative flow.

Figure 44.7 Segment diagram for a well with a down-hole water separator

The separator model

The flow rate of the fluid along the offtake branch depends on the pressure in the branch (which may be controlled by a pump), the pressure in the formation, the pressure losses along the branch and its injectivity index. The action of the separator is to modify the content of the mixture flowing into the offtake branch through the separator offtake, increasing the fraction of the offtake's preferential phase. This is done by transforming the free oil, water and free gas holdup fractions in the offtake segment.

The holdup fractions α^{of} of the mixture entering the offtake are calculated as a function of the holdup fractions α^{sep} of the remaining fluid within the separator segment and a separation efficiency E. For a water separator offtake the transformation is

$$\alpha_w^{of} = E + (1-E)\alpha_w^{sep}$$

$$\alpha_g^{of} = (1-E)\alpha_g^{sep} \qquad \text{[EQ 44.32]}$$

$$\alpha_o^{of} = (1-E)\alpha_o^{sep}$$

Obviously, water cannot be taken off unless it is present within the separator. Thus E must approach zero as α_w^{sep} approaches zero. This functionality is provided by keeping E as a user-defined constant when α_w^{sep} is above a certain value α_{lim}, and letting E linearly approach zero when α_w^{sep} falls below the value α_{lim},

$$E = E_{max} \qquad \text{when } \alpha_w^{sep} > \alpha_{lim}$$

$$E = \frac{\alpha_w^{sep}}{\alpha_{lim}} E_{max} \qquad \text{when } \alpha_w^{sep} < \alpha_{lim} \qquad \text{[EQ 44.33]}$$

E_{max} and α_{lim} are user-defined constants for the offtake, specified in keyword WSEGSEP.

The local volumetric flow rate of each free phase through the offtake is proportional to its holdup fraction α^{of}. Free oil (and free gas) may contain dissolved gas (and vaporized oil) according to the R_s (and R_v) of the phase within the separator segment. The flows at surface conditions are thus

$$Q_o^{of} = A\,j\,(\alpha_o^{of} b_o^{sep} + R_v^{sep} \alpha_g^{of} b_g^{sep})$$

$$Q_w^{of} = A\,j\,\alpha_w^{of} b_w^{sep} \qquad \text{[EQ 44.34]}$$

$$Q_g^{of} = A\,j\,(\alpha_g^{of} b_g^{sep} + R_s^{sep} \alpha_o^{of} b_o^{sep})$$

where A is the area of the offtake, j is the volumetric flux of the mixture (that is its flow velocity) and b is the reciprocal of the phase formation volume factors. The separator segment uses the homogeneous flow model, so all phases flow with the same velocity.

The flows through the outlet junction (towards the wellhead) are similar functions of α^{sep}, the holdup fractions of the fluid remaining within the separator.

Gas separators act in a similar manner to remove free gas (at the separator segment's pressure) and send it preferentially into the offtake segment. They have an equivalent transformation of holdup fractions between the separator and the offtake

$$\alpha_g^{of} = E + (1 - E)\alpha_g^{sep}$$

$$\alpha_w^{of} = (1 - E)\alpha_w^{sep} \qquad \text{[EQ 44.35]}$$

$$\alpha_o^{of} = (1 - E)\alpha_o^{sep}$$

where

$$E = E_{max} \qquad \text{when } \alpha_g^{sep} > \alpha_{lim}$$

$$E = \frac{\alpha_g^{sep}}{\alpha_{lim}} E_{max} \qquad \text{when } \alpha_g^{sep} < \alpha_{lim}$$

[EQ 44.36]

Behavior of the model

An oil/water separation system powered by electric submersible pumps is described in [Ref. 83]. A pump is necessary to boost the pressure to re-inject the water into the formation. Typically, the pump is placed either at the separator inlet to boost the feed pressure (a 'pushthrough pump') or at the water offtake to boost the injection pressure (a 'pullthrough pump'). As we shall see, a pullthrough pump allows a more selective control of the offtake flow rate and the overall separation efficiency of the system. Optionally, a second pump may be placed at the separator outlet to lift the produced oil to the surface.

The overall separation efficiency of the system is governed by the 'flow split', the fraction of the inflow that exits through the oil outlet. Let us consider the behavior of the model for a water separator (with $E_{max} = 1.0$) as the flow split is reduced by steadily increasing the flow rate through the water offtake. At low offtake rates, when the flow through the offtake is less than the inflow rate of water to the separator, the offtake flow is purely water. The remainder of the water exits with the oil through the oil outlet. As the offtake flow increases towards the value of the water inflow rate, the water fraction flowing through the oil outlet decreases to a residual value that depends on α_{lim}. As the offtake flow increases further to exceed the water inflow rate, the water fraction in the oil outlet remains at a residual value but there is an increasing carry-over rate of oil through the water outlet. The residual water fraction in the oil outlet stream can be made arbitrarily small by decreasing the input value of α_{lim}, but in general the smaller this value is, the more difficult it is for the well solution to converge.

For optimum separation, the flow rate through the water offtake should be kept equal to the inflow rate of water into the separator. But the flow rate in the offtake branch depends on the pressure in the separator, the pressure losses along the branch, its injectivity index and the pressure in the formation. However, the offtake flow can be controlled with a pullthrough pump. If the pump is too weak, the flow rate of water along the offtake branch may not be able to keep up with the flow of water into the separator from the formation. In this case the excess water remains in the production stream that goes to the surface. On the other hand, if the pump is too strong the flow of fluid along the offtake branch may exceed the flow of water into the separator. In this case the flow in the outlet branch also includes some of the oil from the formation, which is re-injected along with the water. The choice of pump power depends on whether the primary objective is to re-inject clean water (at the expense of leaving some water in the production stream to the surface), or to remove all the water from the production stream (at the expense of re-injecting some of the produced oil).

ECLIPSE offers a number of ways of automatically cutting back the pressure differential from an over-powered pullthrough pump to maintain optimum separation:

1. Use the built-in device model for a pullthrough pump for a down-hole water separator (keyword WSEGPULL). This consists of a simple formula for the pump pressure differential as a function of flow rate, with an additional term to reduce the pressure differential if there is excessive carry-over of oil. The maximum power of the pump is set by the user, and the actual applied power (determined from the reduced pressure differential) can be written to the Summary file with the SUMMARY section mnemonic SPPOW.
2. Represent the pump pressure versus flow characteristics by a VFP table (using keyword WSEGTABL). Note that the VFP table must have negative flow values, since the flow in the offtake branch is in the direction away from the wellhead. Further down the offtake branch, include a flow limiting device (keyword WSEGFLIM) with a small negative oil rate limit to restrict the rate of oil carry-over.
3. Represent the pump pressure versus flow characteristics by a VFP table (using keyword WSEGTABL) as above, but instead of adding a flow limiting device, modify the table to reduce the pump pressure differential when the water cut drops below 1.0. An example of this type of pump table is shown below. The outlet pressure versus flow curve for a WCT (watercut) = 1.0 represents the pump's full power, delivering a maximum 500 psi pressure boost which drops off as the negative flow increases. But the pump pressure also drops off rapidly with decreasing WCT. As the WCT decreases from 1.0 to 0.9 the pressure boost reduces by a factor of 10. The curve for WCT = 0.0 shows no pressure boost at all, except at very low flow rates. In fact the maximum pressure boost is applied at low negative flow rates and all positive flow rates, for all values of WCT, to prevent reverse flow occurring in the offtake branch. Note also that the illustrated pump curve is insensitive to the gas fraction and the inlet pressure, as there is only one value for each of these quantities.

```
VFPPROD
 1  20.0  LIQ  WCT  GLR  THP  ' '  FIELD  BHP /

 -8000  -6000  -2000  -10  -1  10 /  flow values

 3000 /  THP (pump inlet pressure) values

 0.0  0.9  1.0 /  WCT values

 1.0 /  GOR values

 0 /  ALQ values

 1 1 1 1  3000  3000  3000  3000  3500  3500 /  WCT = 0.0
 1 2 1 1  3000  3030  3040  3050  3500  3500 /  WCT = 0.9
 1 3 1 1  3000  3300  3400  3500  3500  3500 /  WCT = 1.0
s/
```

Whatever model you select for the pump, the nonlinearities in the separator and pump models can make it difficult for the well solution to converge. You are advised to use the WSEGITER keyword to increase the maximum number of well iterations, and if necessary, also to invoke the option to have multiple iteration cycles with variable pressure loss weighting (see "Performance" on page 669).

Thermal multi-segment well model

ECLIPSE 300 Thermal option

In thermal simulations an extra variable is used, representing the energy density in the segment. Energy is stored in both the fluid and the pipe wall, which are assumed to be at the same temperature. A description of the model and some example applications are published in [Ref. 84].

Energy inflow

The flow of energy between a grid block and its associated segment's node is given by the inflow performance relationship

$$q_{ej} = T_{wj} \cdot \sum_p M_{epj} \cdot (P_j + H_{cj} - P_n - H_{nc}) \qquad \text{[EQ 44.37]}$$

where

M_{epj} is the energy mobility in each phase at the connection. If the node is injecting fluid into the formation, the ratio of the phase mobilities reflects the mixture of phases in the segment.

The other terms in this equation are the same as used in the flow equation [EQ 44.1].

Conductive heat transfer

Conductive heat transfer can occur along the well, across the well (between the well tubing and the well casing), and between the well and the formation, which may be inside or outside the simulation grid.

Heat transfer across a small volume is given by the formula

$$Q = -K \cdot A \cdot \frac{\partial T}{\partial X} \qquad \text{[EQ 44.38]}$$

where

K is the thermal conductivity and

A is the cross-sectional area.

This formula is used to calculate heat conduction along the well.

For a length of cylindrical annulus, formula [EQ 44.38] can be integrated over the radius to give

$$Q = \frac{L}{R} \cdot (T_i - T_o) \qquad \text{[EQ 44.39]}$$

where the specific thermal resistance of the annulus is given by

$$R = \frac{1}{2\pi} \cdot \frac{1}{K} \cdot \ln\frac{r_o}{r_i}, \qquad \text{[EQ 44.40]}$$

L is the pipe length

T_i is the temperature at the inner radius r_i and

T_o is the temperature at the outer radius r_o.

The resistance of a thin annulus is calculated by substituting

$$r_o = r_i + \Delta r$$

into [EQ 44.40] to give

$$R = \frac{1}{2\pi} \cdot \frac{\Delta r}{K} \cdot \frac{1}{r_i} \qquad \text{[EQ 44.41]}$$

Heat transfer along the well

[EQ 44.38] is used to model heat transfer along the well. You can specify a thermal conductivity and a cross-sectional area. Values are usually chosen to represent the well walls, since conduction in the walls is usually much larger than conduction in the fluid.

Heat transfer between the well and the reservoir or to an external temperature

[EQ 44.39] is used to calculate heat conduction from the well to the formation. For heat losses from a segment to a reservoir grid, heat transfer occurs from a bulk flow in the segment across a pipe wall, insulation, annulus, casing wall, cement, altered formation to unaltered formation. Heat loss from a segment to rocks outside the simulation grid can be modeled in exactly the same way if you specify the external temperature.

Figure 44.8 Heat conduction between a well and the formation

The thermal resistance for these cases is defined by summing terms similar to [EQ 44.40] and [EQ 44.41], see Prats, Chapter10:

$$R_h = \frac{1}{2\pi}\left[\frac{1}{h_f \cdot r_i} + \frac{1}{h_{P_i} \cdot r_i} + \frac{\ln(r_o/r_i)}{\lambda_P} + \ldots\right.$$

$$\ldots + \frac{1}{h_{P_o} \cdot r_o} + \frac{\ln(r_{ins}/r_o)}{\lambda_{ins}} + \frac{1}{h_{rc_{an}} \cdot r_{ins}} + \ldots$$

$$\left.\ldots + \frac{\ln(r_{co}/r_{ci})}{\lambda_P} + \frac{\ln(r_w/r_{co})}{\lambda_{cem}} + \frac{\ln(r_{ea}/r_w)}{\lambda_{ea}} + \frac{f(t_D)}{\lambda_E}\right]$$

where

h_f is the film heat transfer coefficient between the fluid inside the pipe and the pipe wall

h_{P_i} is the heat transfer coefficient across any deposits of scale or dirt at the inside wall

λ_P is the thermal conductivity of the pipe wall

h_{P_o} is the heat transfer coefficient across the contact between pipe and insulation

λ_{ins} is the thermal conductivity of the insulation

$h_{rc_{an}}$ is the heat transfer coefficient due to radiation and convection in the annulus

λ_P is the thermal conductivity of the casing wall

λ_{cem} is the thermal conductivity of the cement

λ_{ea} is the thermal conductivity of the altered earth

$f(t_D)$ is the time function that reflects the thermal resistance of the earth

λ_E is the thermal conductivity of the unaltered earth.

Since R_h can only be specified as a constant, the time dependent function $f(T_D)$ must be approximated.

Heat transfer from the tubing to the casing

[EQ 44.39] is used to calculate heat conduction losses from a segment to another segment. This case is only applicable when the multi-segment well has been configured with active inner tubing and outer annulus segments. Heat transfer takes place from the bulk flow in one segment across the tubing wall to the bulk flow in the adjoining segment.

Figure 44.9 Heat conduction between the tubing and the casing

The thermal resistance is

$$R_h = \frac{1}{2\pi}\left[\frac{1}{h_f \cdot r_i} + \frac{1}{h_{P_i} \cdot r_i} + \frac{\ln r_o / r_i}{\lambda_P} + \frac{1}{h_{P_o} \cdot r_o} + \frac{1}{h_f \cdot r_o}\right]$$

where

h_{P_o} is the heat transfer coefficient across any deposits of dirt or scale on the outer wall.

Using the multi-segment well model

If the run contains any multi-segment wells, the dimensioning data must be supplied in the RUNSPEC section keyword WSEGDIMS. A run cannot contain both multi-segment wells and friction wells; any friction wells must be converted to multi-segment wells (see below).

Defining multi-segment wells

A multi-segment well, like any other well, must first be introduced with the WELSPECS keyword. The connection data must also be specified in the usual way with the COMPDAT (or COMPDATL) keyword. The well must then be defined as a multi-segment well with the keywords WELSEGS and COMPSEGS (or COMPSEGL; keywords ending in L should be used for wells in local grids).

ECLIPSE 100

An alternative to using WELSEGS and COMPSEGS is to use the keyword WFRICSEG (or WFRICSGL). The data for these keywords is identical to WFRICTN (and WFRICTNL), and they are primarily intended to provide a means of converting a friction well into a multi-segment well having one segment per grid block connection.

The WELSEGS keyword defines the segment structure of a well. The first record defines its top segment, which must be segment number 1. The top segment's node is taken to be the bottom hole reference depth of the well, so that the well's BHP is equal to that segment's pressure. The remaining records define the rest of the well's segments, working down the main stem and, for multi-lateral wells, outwards along each branch. Branches are identified by their branch number; the main stem must be branch number 1.

The COMPSEGS (or COMPSEGL) keyword defines the location of the grid block connections within the well, by the branch number and the distance down the well to the start and end of the perforations. **All** connections in a multi-segment well must have their locations defined with this keyword. ECLIPSE automatically allocates each grid block connection to the segment whose node lies nearest to the center of the connection. A segment may contain any number of grid block connections. If you wish to define the segments so that there is a single grid block connection within each segment, it is best to position the segment nodes to be at the center of each grid block connection (Figure 44.10), since the flow is modeled as entering the well segment at its node.

Figure 44.10 Segment node positions with one segment per grid block connection

In multi-segment wells the grid block connections are not forced to be at the same depth as the corresponding grid block centers. Since ECLIPSE knows the depth of the beginning and end of each segment and the position of the center of the connection along the segment, it can work out the depth of the center of the connection. ECLIPSE sets the connection depths this way by default, although it is possible to redefine the connection depths in the COMPSEGS keyword. The hydrostatic head between the connection depth and the grid block center is calculated explicitly from the fluid contents of the grid block (see "Inflow performance" on page 639).

ECLIPSE 100

It is possible to define the initial conditions (the pressure and the fluid contents) in each segment manually using keyword WSEGINIT. This overrides the standard initialization that ECLIPSE performs, which depends in the type of well and the conditions in the connecting grid blocks.

If required, looped flowpaths within multi-segmented wells can be specified using the WSEGLINK keyword. See "Looped flowpaths" on page 642 for further information.

Defining the flow model

The WELSEGS keyword also defines the default flow model for the well: homogeneous flow or the drift flux model. When the segments are first defined, they adopt the flow model specified in WELSEGS. But it is possible to change the flow model for each segment individually using the keyword WSEGFMOD.

If any of the segments in a well are designated to use the drift flux model, that model must be selected as the well's default flow model in WELSEGS item 7, so that the well uses the set of variables required by that model. The drift flux model contains a number of adjustable parameters that you can set (see "The drift flux slip model" on page 648). You can change the model definition, which dictates the default values of these parameters for all well segments, using keyword WSEGDFMD. If you wish to give some of the parameters different values from their model defaults, you can do so by **subsequently** entering the WSEGDFPA keyword. You can set selected parameters for individual segments with keyword WSEGFMOD. The drift flux model also requires values of the interfacial tensions; these may either be input as tables of pressure with keywords STOG, STOW and STWG (in ECLIPSE 100) or calculated from built-in correlations (see "Three-phase flow and oil-water slip" on page 652).

Individual segments may be designated to have their pressure drops determined from VFP tables, using the keyword WSEGTABL (see "Pressure drops from VFP tables" on page 645). The tables must be structured as production well VFP tables, and must have previously been entered with the keyword VFPPROD.

Specific segments may also be designated to represent

- a flow limiting valve by using the keyword WSEGFLIM
- or a general sub-critical valve by using keyword WSEGVALV
- or a 'labyrinth' flow control device by using the keyword WSEGLABY
- or a 'spiral' flow control device by using the keyword WSEGSICD
- or an 'autonomous' flow control device by using the keyword WSEGAICD.

ECLIPSE uses built-in formulae to model the pressure losses across these devices.

Modifying pressure drops and segment properties

The WSEGMULT keyword applies a scaling factor to the frictional pressure drop along specified segments. This could be used, for example, to reduce the flow from a branch by imposing a high frictional pressure drop in the segment nearest the main stem. It could also be used to account for pressure loss in bends, if their 'equivalent length' is known, as they increase the effective length of the segment for the frictional pressure drop.

Optionally, the pressure drop multiplier can be made to vary with the water-oil ratio and gas-oil ratio of the mixture flowing through the segment. A variable multiplier may be applied, for example, to model the operation of an adjustable flow control device whose purpose is to choke back production from high WOR or GOR regions.

The WSEGPROP keyword can be used to alter the geometric properties of a segment. The segment's length, however, cannot be altered because the grid block connections are located by their overall distance down the tubing.

Workovers

When a workover with an action '+CON' is performed on a multi-segment well, that is "close the worst-offending connection and all below it" (see Item 7 of the WECON keyword, for example), this is interpreted as meaning "close the connection and all connections **upstream** of it". Thus instead of selecting the connections to close according to their depth, they are selected according to their length down the tubing—all connections further from the wellhead are closed. In multi-lateral wells, the branch topology is taken into account so that only branches subordinate to the worst-offending connection are closed off.

Heat loss

ECLIPSE 300 Thermal Option

Heat storage in the tubing and casing can be specified using the WELSEGS keyword. This keyword can also be used to specify longitudinal conduction in the tubing and casing between successive segments.

Heat conduction to the reservoir, and heat loss outside the reservoir can be specified using keyword WSEGHEAT. Heat conduction to the reservoir occurs between segments and completion grid cells. Therefore it may be necessary to define completions in each grid cell that the well passes through. These completions can be shut if no flow occurs between the grid cell and the segment. The length of well in each completion cell can be defined using the COMPSEGS keyword.

WSEGHEAT can also be used to model radial heat conduction between non-successive segments, for example between tubing segments and casing segments.

For multi-segment well source terms, the keyword WSEGEXSS can be used to specify the temperature of imported gas.

Thermal contact lengths

Heat transfer occurs to or from a segment node. Since each segment node is at the end of a segment, the heat transfer to and from each node represents half the heat transfer in the two segments either side of the node. For heat transfer between a completion and a segment the thermal contact length is the length of well within the completion cell. This should be specified using the COMPSEGS keyword. For heat transfer between segments, or between a segment and an external temperature, the thermal contact length is the length of well closest to the segment node.

Figure 44.11 Thermal contact lengths

Output

ECLIPSE 100

The segment structure and connection locations are written to the PRINT file in tabular form if the 14th switch of RPTSCHED (mnemonic WELSPECS) is set. It is advisable to check your data by examining this output, as an error can easily be made while entering complicated multi-lateral well data.

At each report time when well reports are requested (ninth switch of RPTSCHED, mnemonic WELLS), an additional table is printed containing information about conditions within each segment.

Information on flows, pressures and pressure drops within individual segments can be written to the SUMMARY file (see the SUMMARY section overview).

The facility in the Wellbore Friction option to output information that GRAF uses to produce specialized plots is also available with multi-segment wells (see "Graphical presentation of friction effects in horizontal wells" on page 1172). GRAF recognizes the presence of multi-segment wells, and produces the same series of prompts as it does for friction wells.

More detailed information on the flowing conditions inside multi-segment wells can be written to the RFT file at specified times (see the WRFTPLT keyword).

ECLIPSE 300

The segment structure is written to the PRINT file under the WELLS heading in tabular form if the sixth switch of RPTPRINT is set to 1, which is the default. This occurs whenever the multi-segment well is updated.

A completion summary that gives details on connections to the segments is given under the COMPLETIONS heading if the seventh switch of RPTPRINT is set to 1, which is the default. This occurs whenever the multi-segment well is updated.

At each report time when well reports are requested (sixth switch of RPTPRINT), an additional table is printed containing information about conditions within each segment.

Information on flows, pressures and pressure drops within individual segments can be written to the SUMMARY file (see the SUMMARY section overview).

The facility in the ECLIPSE 100 Wellbore Friction option to output information which GRAF uses to produce specialized plots is also available with multi-segment wells (see "Graphical presentation of friction effects in horizontal wells" on page 1172). GRAF recognizes the presence of multi-segment wells, and produces the same series of prompts as it does for friction wells. Note that the SUMMARY section keywords for connection flows differ from the corresponding ECLIPSE 100 keywords.

Performance

If the multi-segment well contains looped flowpaths (see the WSEGLINK keyword), then a different solver is used for the well linear equations with each well iteration. The WSEGSOLV keyword can be used to modify the behavior of this looped well linear solver if required.

ECLIPSE 100

Multi-segment wells can require substantially more well iterations to converge than standard wells, especially when being opened or while switching into or out of crossflow. By default, each well is allowed up to MXWSIT iterations to converge, which is set in item 5 of the third record of keyword TUNING (or TUNINGL or TUNINGS) and defaults to 8. If it cannot converge in this number of iterations, a ‘problem’ message is issued and the latest (unconverged) well solution is used. This may hamper the convergence of the non-linear iterations of the time step. For multi-segment wells, it is recommended to use the WSEGITER keyword to give the well solution more opportunity to converge. The keyword can increase the maximum allowed number of iterations, as an alternative to using TUNING. Furthermore it can invoke a sequence of iteration cycles which traverses ‘difficult’ regions of the residual function by taking smaller steps in the pressure variable. Refer to the WSEGITER keyword documentation for a more detailed explanation of the process.

Other facilities that influence the convergence of the well solution can be controlled using the OPTIONS keyword. The 30th OPTIONS keyword switch governs methods aimed at improving convergence by chopping the increment in all the well’s solution variables. The 63rd OPTIONS keyword switch can also be set if the explicit calculation of the hydrostatic head between the well connections and the corresponding grid block centers causes oscillations.

If there are problems encountered with multi-segment wells under THP control, especially when on the point of dying, the wells should be switched to using explicit water cut and GOR values for VFP table lookup (keyword WVFPEXP).

ECLIPSE 300

Multi-segment wells can require more well iterations to converge than standard wells require. There is a CPU expense if run in compositional mode with many multi-segment wells and many segments per well. To gain the best performance, it is recommended that only a few wells be run as multi-segment wells and that the number of active segments not be large. There are tuning parameters available that affect the convergence, but it is not generally recommended to change their values. The 23rd item of CVCRIT changes the maximum number of iterations for convergence of the well, with a default of 40. Item 24 changes the convergence tolerance for the molar densities, with a default of 0.005 moles/segment volume. Item 25 changes the convergence tolerance for the total molar flow rate, with a defaulted of 100 moles/day (or 100 moles/hour if LAB units are chosen). Item 26 sets the convergence tolerance for energy density that is only applicable to Thermal runs; this defaults to 500 energy units/segment volume.

The complexity of the branches does not significantly affect performance of the multi-segment wells. Neither does the number of completions that are connected.

See also "The top segment" on page 641.

If a multi-segment well is crossflowing, there may be multiple solutions and consequent difficulty in converging. If the well exhibits these symptoms, you have the option of setting the 34th argument of OPTIONS3 to be greater than zero. If this argument is set to the number of the particular well in question (wells are numbered in the order they are input), two extra steps are taken to help converge when cross-flowing. Firstly, the multi-segment well primary variables at the start of each well calculation is initialized to the solution at the end of the last time-step. Secondly, during the solution of the flow control rate constraint, the search for the BHP puts extra weight on bisection rather than Newton. This may result in more iterations being taken to converge the well, but it helps convergence in difficult situations. If the 34th argument of OPTIONS3 is set to 10000 then all multi-segment wells use the above.

Restrictions

ECLIPSE 100

Multi-segment wells cannot at present be used in conjunction with the following facilities:

1. The Wellbore Friction option
2. The Polymer Flood model
3. The Temperature option
4. The Solvent model
5. The Gi model
6. Automatic Refinement in the LGR option

In addition, it is strongly recommended that multi-segment wells are not used in conjunction with the following facilities:

- The COMPFLSH keyword.
  - In both the conventional well and the multi-segment well models, the inflowing fluid is transformed to flash conditions when it enters the wellbore through the connections to the reservoir grid. To calculate the fluid properties (for example for the hydrostatic head) a back transformation is applied based on the overall inflow; this is consistent with the treatment of the well as a “mixing tank” and is thus appropriate for the conventional well model. With a multi-segment well, this back transformation (based on the overall inflow) is still applied, even though the local fluid properties may vary with position in the well. Thus the local fluid properties are not as accurately represented as they should be for a discretized wellbore model such as a multi-segment well. These fluid properties influence the hydrostatic and frictional pressure gradients, the outflowing mixture properties when crossflow occurs, and slip (when using the drift-flux model).

ECLIPSE 300

Multi-segment wells in ECLIPSE 300 cannot at present be used in conjunction with the following facilities:

1. velocity dependent gas relative permeability contribution to D-factor (VDKRG)
2. changing well dimensions in the SCHEDULE section (CHANDIMS)
3. k-value tables in compositional mode (KVALUES)
4. tracers (TRACER).

Converting friction wells to multi-segment wells

ECLIPSE 100

The keywords WFRICSEG and WFRICSGL provide an additional way of defining a multi-segment well, as an alternative to the WELSEGS and COMPSEGS (or COMPSEGL) keywords. Their primary purpose is to offer you a very simple way of converting existing datasets that use the Wellbore Friction model (see "The Wellbore Friction Option" on page 1163) into datasets that use multi-segment wells. The data for each keyword is identical to that of the WFRICTN and WFRICTNL keywords, but instead of creating a friction well each keyword creates a multi-segment well with the same branch structure and connection locations. ECLIPSE automatically divides the well into segments such that there is one segment per grid block connection, plus an additional segment at the well's bottom hole reference point. The segments are distributed so that the nodal point of each segment is located at the center of the corresponding grid block connection. The segment lengths are set equal to the tubing length between adjacent connections. The depths of the segment nodes are set equal to the center depths of the connecting grid blocks, as there is no depth data in WFRICTN(L).

The WFRICSEG keyword does not offer the flexibility of WELSEGS and COMPSEGS in defining multi-segment wells, so they are not generally recommended for uses other than converting friction well data. The disadvantages of using the WFRICSEG keyword are:

1. There is an enforced one-to-one correspondence between segments and grid block connections. This may result in an excessively large number of segments.
2. The segment nodes have their depths set equal to the center depths of the connecting grid blocks. Thus, unless the grid is aligned with the well trajectory, a horizontal well may have spurious undulations or deviations from its true trajectory.
3. Branch junctions defined in WFRICTN have to be located at a grid block connection, so extra connections have to be defined at branch junctions; if there is no inflow from the formation at these locations the connections should be declared shut in COMPDAT.
4. When WFRICSEG is used to define a multi-lateral well you cannot dictate the branch numbers; ECLIPSE creates the branches and numbers them itself. However, ECLIPSE is unable to recognize whether the original main stem continues past a branch junction. Instead, **two** new branches are created at each branch junction, one of these being the continuation of the main stem. It is important to allow for this when setting the maximum number of branches in WSEGDIMS. For example, a well consisting of a main stem and a single lateral branch will be converted to have three branches: the upper part of the main stem, the lateral branch and the section of the main stem below the branch junction.

Converting a dataset from using friction wells to using multi-segment wells

1. Remove the FRICTION keyword from the RUNSPEC section, replacing it with the WSEGDIMS keyword.
   a. The maximum number of multi-segment wells in the run is the same as the maximum number of friction wells.
   b. The maximum number of segments per well is be 1 + the maximum number of grid block connections per well.
   c. The maximum number of branches per well should allow for the 'extra branches' created by ECLIPSE in the main stem below lateral branch junctions.

2 Change all WFRICTN keywords to WFRICSEG. In LGR runs, any WFRICTNL keywords should be changed to WFRICSGL. The keyword data should remain unchanged.

3 Optionally the SUMMARY section can include the mnemonics associated with multi-segment wells (SOFR etc.).

When ECLIPSE converts a friction well into a multi-segment well, it prints out its equivalent WELSEGS and COMPSEGS (or COMPSEGL) keyword data in the PRINT file, at the place where WFRICTN (or WFRICTNL) is read. The generated keyword data can be copied and used in subsequent runs in place of the original friction well keyword. It is then possible to edit the data, for example to renumber the whole of the main stem to be branch 1, to remove spurious undulations by adjusting the segment depths to agree with the well trajectory, and to reduce the number of segments if desired.

Summary of keywords

RUNSPEC section

The keywords are:

Table 44.2 RUNSPEC section

Keyword	Description
WSEGDIMS	Sets array dimensions for multi-segment wells.
WSEGSOLV	Sets parameters for the multi-segment well iterative linear solver.

PROPS section

The keywords are:

Table 44.3 PROPS section

Simulator	Keyword	Description
ECLIPSE 100 only	STOG	Oil-gas surface tension versus pressure (used in the drift flux model).
ECLIPSE 100 only	STOW	Oil-water surface tension versus pressure (used in the drift flux model).
ECLIPSE 100 only	STWG	Water-gas surface tension versus pressure (used in the drift flux model).

SUMMARY section

The following keywords control output of segment data specific to multi-segment wells:

Table 44.4 Multi-segment wells SUMMARY output controls

E100	E300	Segment	Information
x	x	SOFR	Segment Oil Flow Rate
x		SOFRF	Segment Free Oil Flow Rate
x		SOFRS	Segment Solution Oil Flow Rate (vaporized oil)
x	x	SWFR	Segment Water Flow Rate
x	x	SGFR	Segment Gas Flow Rate
x		SGFRF	Segment Free Gas Flow Rate
x		SGFRS	Segment Solution Gas Flow Rate
	x	SCFRn	Segment Component n Flow Rate
	x	SHFR	Segment Enthalpy Flow Rate (Thermal option)
x	x	SWCT	Segment Water Cut
x	x	SGOR	Segment Gas Oil Ratio
x	x	SOGR	Segment Oil Gas Ratio
x	x	SWGR	Segment Water Gas Ratio
x	x	SPR	Segment PRessure
x	x	SPRD	Segment PRessure Drop
x	x	SPRDF	Segment PRessure Drop component due to Friction
x	x	SPRDH	Segment PRessure Drop component due to Hydrostatic head
x	x	SPRDA	Segment PRessure drop due to Acceleration head
x	x	SPRDM	Segment frictional PRessure Drop Multiplier (keyword WSEGMULT)
x		SPPOW	Working power of a pullthrough pump (keyword WSEGPULL)
x	x	SOFV	Segment Oil Flow Velocity
x	x	SWFV	Segment Water Flow Velocity
x	x	SGFV	Segment Gas Flow Velocity
x	x	SOHF	Segment Oil Holdup Fraction
x	x	SWHF	Segment Water Holdup Fraction
x	x	SGHF	Segment Gas Holdup Fraction
x	x	SDENM	Segment fluid mixture density
x	x	SGLPP	Segment Gas-Liquid Profile Parameter, C0 (drift flux slip model)
x	x	SGLVD	Segment Gas-Liquid Drift Velocity, Vd (drift flux slip model)
x	x	SOWPP	Segment Oil-Water Profile Parameter, C0 (drift flux slip model)
x	x	SOWVD	Segment Oil-Water Drift Velocity, Vd (drift flux slip model)

Table 44.4 Multi-segment wells SUMMARY output controls (Continued)

E100	E300	Segment	Information
	x	SOIMR	Segment Oil Import Rate (keyword WSEGEXSS)
x	x	SGIMR	Segment Gas Import Rate (keyword WSEGEXSS)
x	x	SWIMR	Segment Water Import Rate (keyword WSEGEXSS)
	x	SHIMR	Segment Enthalpy Import Rate (keyword WSEGEXSS) (Thermal option)
	x	SORMR	Segment Oil Removal Rate (keyword WSEGEXSS)
	x	SGRMR	Segment Gas Removal Rate (keyword WSEGEXSS)
	x	SWRMR	Segment Water Removal Rate (keyword WSEGEXSS)
	x	SHRMR	Segment Enthalpy Removal Rate (keyword WSEGEXSS) (Thermal option)
	x	SOIMT	Segment Oil Import Total (keyword WSEGEXSS)
x	x	SGIMT	Segment Gas Import Total (keyword WSEGEXSS)
x	x	SWIMT	Segment Water Import Total (keyword WSEGEXSS)
	x	SHIMT	Segment Enthalpy Import Total (keyword WSEGEXSS) (Thermal option)
	x	SORMT	Segment Oil Removal Total (keyword WSEGEXSS)
	x	SGRMT	Segment Gas Removal Total (keyword WSEGEXSS)
	x	SWRMT	Segment Water Removal Total (keyword WSEGEXSS)
	x	SHRMT	Segment Enthalpy Removal Total (keyword WSEGEXSS) (Thermal option)
x		SAPI	Segment API value
x		STFR	Segment tracer flow rate
x		STFC	Segment tracer concentration
	x	SPSAT	Segment Psat
	x	STEM	Segment Temperature
	x	SENE	Segment Energy (Thermal option)
	x	SSQU	Segment Steam Quality (Thermal option)
	x	SCVPR	Segment Calorific Value Production Rate
	x	SGQ	Segment Gas Quality
	x	SCSA	Segment Cross Sectional Area. This is specified using the WSEGVALV keyword, and can change if the cross sectional area is used as a reservoir optimization parameter using the OPTPARS keyword.

The segment flow rates reported are the flow rates of each phase through the tubing past the segment's end nearest the well head. Positive values represent flow towards the well head (production), while negative values represent flow away from the well head (injection). The flow rates are reported at surface conditions (the sum of free and dissolved phases in ECLIPSE 100), equivalent to the reported well rates. The flow velocities refer to the local flow velocities of the free phases through the segment. The flow velocities of all the phases should be equal unless you are using a built-in multi-phase flow model that allows slip. The holdup fractions refer to the local volume fraction of the segment occupied by each free phase.

ECLIPSE 100

The tracer flow rate and concentration keywords should be compounded with the name of the tracer to which they refer (for example: STFROL1 and STFCOL1 output the flow rate and concentration of the tracer OL1).

The segment quantity keywords all begin with the letter S. They must be followed by one or more records each containing the well name and the segment number and terminated with a slash. The list of records must be terminated with an empty record just containing a slash. The segment number following a well name may optionally be defaulted, which causes the appropriate quantity to be output for all the segments in the well.

Note If the segment number is defaulted, as for well PROD2 below, ECLIPSE reserves a number of SUMMARY vectors equal to the maximum number of segments per well (NSEGMX, set in RUNSPEC section keyword WSEGDIMS). If this number is much greater than the actual number of connections in well PROD2 then defaulting the segment numbers can become quite wasteful.

For example,

```
SOFR
 PROD1   1 /
 PROD1   4 /
 PROD1   5 /
 PROD2     /
/
```

The following keywords control the output for segments which have been designated as secondary wells.:

Table 44.5 Multi-segment wells SUMMARY output controls for secondary wells

E100	E300	Secondary Well	Information
	x	WDOPR	Secondary well oil production rate.
	x	WDWPR	Secondary well water production rate.
	x	WDGPR	Secondary well gas production rate.
	x	WDLPR	Secondary well liquid production rate.
	x	WDVPR	Secondary well reservoir volume production rate.
	x	WDOPT	Secondary well oil production total.
	x	WDWPT	Secondary well water production total.
	x	WDGPT	Secondary well gas production total.
	x	WDLPT	Secondary well liquid production total.
	x	WDVPT	Secondary well reservoir volume production total.
	x	WDWIR	Secondary well water injection rate.
	x	WDGIR	Secondary well gas injection rate.
	x	WDWIT	Secondary well water injection total.
	x	WDGIT	Secondary well gas injection total.
	x	WDOGR	Secondary well oil-gas ratio.
	x	WDGOR	Secondary well gas-oil ratio.
	x	WDGLR	Secondary well gas-liquid ratio.
	x	WDWGR	Secondary well water-gas ratio.

Table 44.5 Multi-segment wells SUMMARY output controls for secondary wells (Continued)

E100	E300	Secondary Well	Information
	x	WDWCT	Secondary well water cut.
	x	WDTEM	Secondary well temperature.
	x	WDPR	Secondary well pressure.

The following keywords control the outputs for groups which include the contributions from secondary wells:

Table 44.6 Multi-segment wells SUMMARY output controls for groups including secondary well contributions

E100	E300	Secondary Well	Information
	x	GDOPR	Oil production rate including secondary well contributions.
	x	GDWPR	Water production rate including secondary well contributions.
	x	GDGPR	Gas production rate including secondary well contributions.
	x	GDLPR	Liquid production rate including secondary well contributions.
	x	GDVPR	Reservoir volume production rate including secondary well contributions.
	x	GDOPR	Oil production total including secondary well contributions.
	x	GDWPR	Water production total including secondary well contributions.
	x	GDGPR	Gas production total including secondary well contributions.
	x	GDLPR	Liquid production total including secondary well contributions.
	x	GDWIR	Water injection rate including secondary well contributions
	x	GDGIR	Gas injection rate including secondary well contributions.
	x	GDWIT	Water injection total including secondary well contributions.
	x	GDGIT	Gas injection total including secondary well contributions.
	x	GDOGR	Oil-gas ratio including secondary well contributions.
	x	GDGOR	Gas-oil ratio including secondary well contributions.
	x	GDGLR	Gas-liquid ratio including secondary well contributions.
	x	GDWGR	Water-gas ratio including secondary well contributions.
	x	GDWCT	Water cut including secondary well contributions.

The following keywords control the outputs for the field which include the contributions from secondary wells:

Table 44.7 Multi-segment wells SUMMARY output controls for field including secondary well contributions

E100	E300	Secondary Well	Information
	x	FDOPR	Oil production rate including secondary well contributions.
	x	FDWPR	Water production rate including secondary well contributions.
	x	FDGPR	Gas production rate including secondary well contributions.
	x	FDLPR	Liquid production rate including secondary well contributions.
	x	FDVPR	Reservoir volume production rate including secondary well contributions.
	x	FDOPT	Oil production total including secondary well contributions.
	x	FDWPT	Water production total including secondary well contributions.
	x	FDGPT	Gas production total including secondary well contributions.
	x	FDLPT	Liquid production total including secondary well contributions.
	x	FDWIR	Water injection rate including secondary well contributions
	x	FDGIR	Gas injection rate including secondary well contributions.
	x	FDWIT	Water injection total including secondary well contributions.
	x	FDGIT	Gas injection total including secondary well contributions.
	x	FDOGR	Oil-gas ratio including secondary well contributions.
	x	FDGOR	Gas-oil ratio including secondary well contributions.
	x	FDGLR	Gas-liquid ratio including secondary well contributions.
	x	FDWGR	Water-gas ratio including secondary well contributions.
	x	FDWCT	Water cut including secondary well contributions.

SCHEDULE section

The keywords are:

Table 44.8 SCHEDULE section

Simulator	Keyword	Description
	COMPSEGL	Defines the location of multi-segment well completions in a local grid.
	COMPSEGS	Defines the location of completions in a multi-segment well.
ECLIPSE 300 only	CVCRIT	Items 23 - 26 set convergence criteria for multi-segment wells.
ECLIPSE 100 only	OPTIONS	Switch 30 controls the use of a binary chop sequence, which restricts the size of the solution increments if the residuals get worse. However, you are recommended to try the WSEGITER keyword before changing this switch. Switch 63 controls the calculation of the hydrostatic head between each connection of a multi-segment well and its corresponding grid block. Switch 77 restores back-compatibility with the pre-2000A drift flux model calculations involving oil-water slip and inclined tubing.

Table 44.8 SCHEDULE section

Simulator	Keyword	Description
ECLIPSE 300 only	OPTIONS3	Switch 34 may help convergence in difficult situations involving crossflow.
ECLIPSE 300 only	SCONPROD	Secondary well production controls.
ECLIPSE 300 only	SCONINJE	Secondary well injection controls.
ECLIPSE 300 only	STEST	Secondary well reopen test control.
	WELSEGS	Defines the segment structure of a multi-segment well.
ECLIPSE 100 only	WFRICSEG	Converts WFRICTN keyword data for a friction well into data for a multi-segment well.
ECLIPSE 100 only	WFRICSGL	Converts WFRICTNL keyword data for a friction well into data for a multi-segment well.
	WSEGAICD	Defines segments to represent an 'autonomous' inflow control device.
	WSEGDFIN	Inputs a table of inclination scaling factors versus deviation angle for the Drift Flux slip model
	WSEGDFMD	Defines the Drift Flux slip model
	WSEGDFPA	Changes parameter values in Drift Flux slip model.
	WSEGEXSS	Specifies fluid import/removal through an external source/sink.
	WSEGFLIM	Defines segments to represent a flow limiting valve.
	WSEGFMOD	Specifies the multi-phase flow model and related parameters.
ECLIPSE 300 *Thermal only*	WSEGHEAT	Specifies heat loss from a segment.
ECLIPSE 100 only	WSEGINIT	Sets initial pressure and fluid contents of well segments.
ECLIPSE 100 only	WSEGITER	Activates the robust iteration scheme for solving multi-segment wells.
	WSEGLABY	Defines segments to represent a 'labyrinth' inflow control device.
	WSEGLINK	Defines the chord segment links for specifying looped flowpaths
	WSEGMULT	Imposes constant or variable multipliers for segment frictional pressure drops.
	WSEGPROP	Modifies the properties of individual well segments.
ECLIPSE 100 only	WSEGPULL	Defines a pullthrough pump for a down-hole water separator.
ECLIPSE 100 only	WSEGSEP	Defines a segment to represent a down-hole separator.
	WSEGSICD	Defines segments to represent a 'spiral' inflow control device.
	WSEGTABL	Designates segments to have their pressure drops calculated from a VFP table.
	WSEGVALV	Defines segments to represent a sub-critical valve.
ECLIPSE 300	WSEGWELL	Define the association between segment number and secondary well name.

The Network Option

Chapter 45

Introduction

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL

The Network option is designed to provide variable tubing head pressure (THP) limits to groups of wells, which depend on the groups' flow rates according to a set of pipeline pressure loss relationships. The option calculates the well THP limits dynamically by balancing the flow rates and pressure losses in the network.

Consider a group of wells flowing into a common manifold which is connected by a pipeline to an outlet at a fixed pressure (Figure 45.1). For example, the group may represent the wells flowing into a particular sub-sea completion manifold, and the fixed pressure outlet may be a separator or stock tank. The pressure in the manifold will vary with the overall production rate of the group, according to the relationship between the pressure loss through the pipeline and the flow rate, water cut and GOR of the group. This relationship is represented schematically in Figure 45.2 by the Pipeline curve. However, the group's production rate will itself vary with the manifold pressure if at least one of its wells is on THP control, since the manifold pressure is applied to each well as a THP limit. The combined production rate of the wells vs. the THP limit is represented by the Wellbore curve in Figure 45.2. The actual manifold pressure and flow rate of the group will be given by the intersection of these two curves.

Figure 45.1 Common wellhead manifold and pipeline

Figure 45.2 Well THP versus Group Flow Rate for Wellstream and Pipeline

Using the ECLIPSE multi-level grouping hierarchy, a pipeline network can be built up, with a pipeline connecting each group to its 'parent' in the grouping tree. A group of wells connected to a common manifold passes its production stream through a pipeline to its parent group, which gathers the production from a number of common well manifolds and sends it along another pipeline to its own parent group, and so on up the grouping tree until a group with a fixed-pressure separator or stock tank is encountered. Each group is a 'node' in the network. The group with the separator is a 'fixed-pressure node', and the groups subordinate to this have their own 'nodal pressures,' which are determined from the pipeline pressure losses and the production rates. The nodal pressures of each well-group at the bottom of the tree are applied to their wells as THP limits. (Note that any fixed THP limits specified for the wells will be over-written by the calculated values.) The nodal pressure of each group is printed in the well reports in the THP column, and can also be written to the `SUMMARY` file using the mnemonic GPR.

For example, consider the grouping tree shown in Figure 45.3. The groups on the bottom level (GR-A1 etc.) may be sub-sea completion manifolds, connected to their respective platforms PLAT-A and PLAT-B by pipelines. The two platforms can be declared as fixed-pressure nodes, with pressures set equal to their separator pressures. Alternatively the FIELD can be given a fixed pressure, and the two platforms connected to it by additional pipelines. It is not essential for the network to cover the whole grouping tree. For example group GR-A2 can be isolated from the network (by not giving it a pipeline connection to PLAT-A), and THP limits for its wells set or defaulted independently.

Figure 45.3 An example network

Note that the network structure coincides with the grouping hierarchy built using the GRUPTREE keyword. If you require the network to have a different topology from the group control hierarchy, then you must use the Extended Network model which is described in a later section (see "The extended network model" on page 693).

Group production control in the network option

Group control targets and limits can be applied to any group regardless of whether it is part of a pipeline network. A production rate target applied to a group in the network will be met as long as its subordinate wells have sufficient potential. The wells' potentials will of course depend on the nodal pressures in the network, since these determine the wells' THP limits.

Consider, for example, a target production rate of 10,000 STB/DAY of oil applied to the group GR-A1. The THP limits of each of its wells are set equal to the group's nodal pressure in the network. The standard procedure for group control will apportion the rate target among the group's wells in proportion to their oil phase guide rates (which by default are set equal to their production potentials). If some wells cannot produce at their allocated rate targets, they will produce as much as they can under their own flow and pressure limits, while the remaining wells under group control make up the rest of the group's flow target.

When a well is under group control, it will be producing at the rate target allocated to it. Its THP has a value greater than the group's nodal pressure in the network (otherwise the well will have changed to THP control). The THP values of the group-controlled wells in this group are in general all different. However, if the group represents a sub-sea completion manifold, in reality all its wells are operating at the same THP, that is the pressure in the manifold. The standard method of group control by guide rates is therefore not appropriate for groups in which all the wells share a common manifold.

ECLIPSE 100

An alternative method of group control is available for groups representing sub-sea completion manifolds. This method calculates the THP required to make the group produce at the desired rate. Thus it allows the group to produce at a specified rate while its wells remain on THP control operating at a common THP. Physically this method is equivalent to applying a choke to the group's outlet flow line, to increase the upstream pressure and reduce the production rate, as opposed to choking back each well separately. The group is able to produce at its target rate as long as the calculated value of its wells' common THP remains above the group's (unchoked) nodal pressure in the network. If the calculated THP value falls below the group's nodal pressure in the network, the wells operate at the group's nodal pressure and the group is no longer able to produce its target rate. At this point the production rules (see keyword PRORDER) are activated to increase the productivity of the group and bring the group's production back to its target rate. This alternative method of group control may be invoked by declaring the group to be a sub-sea completion manifold in keyword GRUPNET or NODEPROP.

ECLIPSE 100

In the Standard Network model, this method of group control can only be applied to well-groups (that is the groups containing wells, not higher level groups that have subordinate groups). However, when using the Extended Network model, a similar treatment can be applied to production rate targets at any level below the fixed pressure node by invoking the automatic choke option. (See "Automatic chokes" on page 695.) In the Standard Network model a target rate applied to a higher level group will be apportioned among its subordinate wells, or groups with guide rates, according to the standard group control procedure. For example, consider a rate target of 20,000 stb/day of oil applied to group PLAT-A. This is apportioned among the wells in groups GR-A1 and GR-A2 in proportion to their guide rates. The wells under group control operate at their allocated rates, and in general they all have different THP values. Defining groups GR-A1 and GR-A2 as manifold groups has no effect, as the rate target applies to a higher level group. However, if the manifold groups are also given guide rates (in keyword GCONPROD), the alternative method of group control can be applied and their wells operate at

their common manifold pressures. This is because the rate target given to PLAT-A is passed down to the groups GR-A1 and GR-A2 in proportion to their guide rates. If these two groups are given equal guide rates, for example, they both have a target of 10,000 stb/day of oil. In each of the two groups, the common THP required to produce this rate is calculated and applied to their wells as THP limits. If one group cannot make its target without falling below its network nodal pressure, the other group tries to make up PLAT-A's production to 20,000 stb/day. To summarize, if a rate target is to be applied to a group that is superior to a manifold group, the subordinate manifold groups should be given guide rates. A warning message is printed of this is not so.

ECLIPSE 300

ECLIPSE 300 does not use this method of applying rate constraints to manifold groups. Instead it uses automatic chokes to apply rate constraints at any level in the network, not just to well groups. (See "Automatic chokes" on page 695.) The method is the same whether using the Standard or Extended Network models. ECLIPSE calculates the pressure drop across the choke that is necessary to reduce the flow rate to the required target value. The choke must be represented by a branch with a VFP table number of 9999, so that the only pressure loss in this branch is that of the choke. If the branch represents a pipeline having its own pressure losses, an intermediate node must be inserted to distinguish the choke from the rest of the pipeline.

Pumps and compressors

Pumps and compressors in the production network's flow lines can be modeled using the Artificial Lift Quantity, in a similar way to the modeling of a pump or gas lift in a well. The artificial lift quantity for each flow line can be set and re-set at any time in the simulation.

ECLIPSE 100

There is an option to switch on a pump or compressor in the network if a group's oil or gas production rate falls below a specified limit. For example, consider the pipeline between PLAT-A and the FIELD. The production from PLAT-A declines when its subordinate wells are operating with their THP values equal to their groups' nodal pressures. When the production rate falls below a specified limit, a pump or compressor in the pipeline from PLAT-A can be switched on automatically. This reduces the pressure losses in the pipeline and reduce PLAT-A's nodal pressure, consequently reducing the nodal pressures in the subordinate groups and boosting their wells' production.

ECLIPSE 100

Additional, more versatile, facilities for automatic pump or compressor switching are available when using the Extended Network model, which is described in a later section (see "Automatic compressors" on page 694).

ECLIPSE 100

An alternative use for the ALQ is to define it to represent the average surface density of either the oil phase or the gas phase flowing along a pipeline. ECLIPSE then automatically sets it equal to this value when it looks up the VFP table. The VFP table must have been prepared using the same definition of the ALQ, and should span the expected range of surface density values. This option is useful when a mixture of oil or gas with different surface densities flows into the network. Obviously the ALQ cannot then also be used to represent a pump or compressor, and the pipeline cannot contain an automatic pump/compressor.

Injection network

In addition to the production network, a separate network can also be defined for water or gas injection. Each network exists independently of the other. A particular group may be a node in both networks, if required, but in this case the group will have two independent nodal pressures, one for each network to which it belongs.

Pumps can be modeled in the injection network flow lines, but the artificial lift quantity cannot be varied.

As an example of an injection network, consider a pump that takes water at a fixed inlet pressure and pumps it into a manifold supplying a number of water injectors. The outlet pressure of the pump, and thus the pressure in the manifold, varies with the total injection rate. The injection wells operate under THP control, with the THP equal to the pressure in the manifold. Using the grouping tree shown in Figure 45.3, let the injection pump be situated on platform PLAT-B, and the injection wells belong to group GR-B2. Declare PLAT-B as a fixed-pressure node in the water injection network, with a pressure equal to the pump inlet pressure, and declare GR-B2 as a variable-pressure node connected to PLAT-A by a pipeline containing the pump. It makes no difference whether or not PLAT-A and GR-B2 are also nodes in the production network.

ECLIPSE 300

In ECLIPSE 300 it is possible to use the Extended Network model for injection networks, allowing group injection rate constraints to be handled by automatic chokes

Using the standard network option

This section describes how to use the Standard (non-extended) Network option, in which the network structure must be the same as the GRUPTREE hierarchy. If you require the network to have a different topology from the group control hierarchy, then you must use the Extended Network option which is described in a later section (see "The extended network model" on page 693).

ECLIPSE 300

In ECLIPSE 300 you are recommended to use the Extended Network option as this is more versatile than the Standard option. The Standard option keywords, however, are available to assist the porting of ECLIPSE 100 datasets.

Constructing the network

The multi-level grouping tree must first be constructed using the keyword GRUPTREE (but this can be omitted if there is only one level of groups below the field).

The production network is specified using the keyword GRUPNET. With this keyword, one or more groups (or the field itself) can be declared as fixed-pressure nodes, while groups below them in the network are given pipeline connections to their parent groups. The pressure losses in each pipeline are obtained from production VFP tables, in a similar way to the treatment of THP calculations in wells. Thus each pipeline must be associated with a given production VFP table, which is supplied as input data to ECLIPSE in the usual way. The top node of each network must be a fixed-pressure node. It is possible for subordinate groups also to be fixed-pressure nodes; these form 'sub-networks' whose pressures are independent of the main network but whose flows will be added into it.

Water and gas injection networks are specified using the keyword GNETINJE. Again, one or more groups can be declared as fixed-pressure nodes, and groups below them in the network are given pipeline connections from their parent groups. Each pipeline must be associated with an **injection** VFP table for water or gas injection, from which the pressure losses will be calculated.

The pipeline VFP tables can be constructed using the program VFP*i*, which handles horizontal and undulating flowlines as well as vertical and deviated well bores. For a pipeline connecting a parent group and its child, the 'tubing head' end is the parent group and the 'bottom hole' end is the child group. Thus a pipeline VFP table gives the nodal pressures of the child group for selected values of the nodal pressure of the parent group and the flow rate through the pipeline (and also the water and gas fractions in a production network pipeline). The VFP tables should cover the expected range of these quantities in the pipeline. To allow for the case when a group has zero flow rate, the lowest flow rate in the table should be small, for example, 1 stb/day, since the VFP*i* program does not accept a zero flow rate.

Pumps, compressors and gas lift

The VFP tables for the production network pipelines may also include a non-zero value of the Artificial Lift Quantity (ALQ), to model the effects of a pump or compressor on the pipeline. In this case the quantity may be identified with the pump or compressor power. The program VFP can include the effects of a pump or compressor in the pipeline when calculating the pressure losses. The acting pump or compressor power for the pipeline, with which the VFP table is to be interpolated, is set with the keyword GRUPNET. The specified power should correspond with the respective Artificial Lift Quantity value in the VFP tables. An additional option allows a pump or compressor to be switched on automatically in a pipeline when its group's oil or gas production rate falls below a set limit. The option is selected using the keyword GNETPUMP.

Injection network pipelines can also contain pumps, if their VFP tables are constructed to include them. But injection VFP tables do not have the Artificial Lift Quantity as a variable, so a pump cannot be turned on or off by varying this quantity. Instead, two VFP tables may be constructed for the pipeline, one which includes the effect of the pump, and one which does not. Thus the pump can be turned on and off manually during the simulation by changing the pipeline's VFP table number in keyword GNETINJE.

If wells in a production network are subject to gas lift, the flow of lift gas can be included in the network branch pressure drop calculations by setting item 6 in keyword GRUPNET to 'FLO' or 'ALQ' for the pipelines that carry lift gas along with produced fluids. If 'FLO' is selected, the lift gas flows of the subordinate producers will be added to the gas flow rate along the pipeline. The lift gas flow is assumed to be equal to the sum of the ALQ values of the subordinate producers, multiplied by their efficiency factors. If 'ALQ' is selected, the gas flow rate will not be adjusted, but the ALQ value used to look up the pipeline's pressure drop will be set equal to the sum of the ALQ values of the subordinate producers, multiplied by their efficiency factors. If either option is selected, the Artificial Lift Quantity for the wells (WCONPROD item 12) and the pipeline (for the 'ALQ' option) must be defined as the rate of lift gas injection. This definition must be selected in the VFP*i* program if it is used to construct the VFP tables.

Balancing the network

The nodal pressures of the groups in the network are determined by a 'network balancing' calculation. This uses an iterative procedure to calculate the pressures at each node (or group) in the network consistent with the group flow rates and the pipeline pressure losses. At each iteration of the balancing calculation the well and group production rates are determined using the reservoir conditions existing at the beginning of the time step, and the latest iterate of the nodal pressures. The pipeline pressure losses generated by these flow rates are then calculated, and the nodal pressures are updated at each level in the grouping tree from the fixed-pressure node(s) downwards using a secant method. The iterations are terminated when all the nodal pressures agree with the pipeline pressure losses, to within a certain tolerance. In ECLIPSE 100 the number of iterations required to balance the network each time will be written to the PRINT file if either the 10th or the 15th RPTSCHED keyword switch is set (mnemonic VFPPROD or NEWTON). In ECLIPSE 300 the number of iterations coupling the network to the reservoir will automatically be printed in the time step summary table in the column headed NIT.

At each network balancing iteration, the well and group production rates are determined at the latest iterate of the nodal pressures. If a group production target cannot be met, actions may be taken to increase production, such as drilling a new well (see keyword PRORDER). However, it is possible that an intermediate balancing iteration with high nodal pressures and consequentially low well productivities may cause such an action to be taken prematurely. This

can be prevented in ECLIPSE 100 by using the WAITBAL keyword to request that these actions should not be performed until the network is properly balanced. Then if more flow is required once the network has converged, an appropriate action is performed and the network must be balanced again. Using this option therefore requires more network balancing iterations in total. In ECLIPSE 300 there is no need to use WAITBAL as it is automatically enabled; the simulator always waits until the network is properly balanced before drilling a new well.

ECLIPSE 100

You can control the frequency with which the network balancing calculations are performed. The keyword NETBALAN is used to force a network balancing calculation to take place at the beginning of the next time step, and to specify the 'balancing interval' for subsequent time steps. Subsequent balancing calculations are performed at the beginning of each time step that starts after the specified interval has elapsed since the previous calculation. The nodal pressures (which are used to set the well THP limits) are kept constant until updated by the next balancing calculation. The default value of the balancing interval is zero, which results in a balancing calculation being performed at the beginning of each time step. The keyword NETBALAN can also be used to adjust the convergence tolerance and the maximum number of iterations in the calculation.

ECLIPSE 100

By default, the network is balanced using the grid block conditions existing at the beginning of the time step. By the end of the time step, the grid block conditions and well flow rates may have changed, and the flows in the network branches may no longer be exactly consistent with the nodal pressures. The longer the time step or interval between balancing calculations, the greater the balancing error will be at the end of the time step. ECLIPSE 100 checks the balancing error in each branch of the network at the end of each time step, and prints out the largest error in any branch. The balancing error is defined as the difference between the pressure drop along the branch when the network was last balanced, and the pressure drop calculated using the current flow rates at the end of the time step.

ECLIPSE 100

By setting the balancing interval negative in keyword NETBALAN, ECLIPSE 100 can be made to balance the network in each of the first NUPCOL Newton iterations of every time step. (The value of NUPCOL may be set using either keyword NUPCOL or keyword GCONTOL.) This option, in general, results in much smaller balancing errors at the end of the time step, as the reservoir conditions at each successive Newton iteration more closely resemble the converged conditions at the end of the time step. However, the option may cause the time steps to require more Newton iterations to converge, as the well THP limits vary at each balancing calculation.

ECLIPSE 300

In ECLIPSE 300 the network is always balanced in each of the first NUPCOL Newton iterations of every time step. (This is equivalent to setting a negative balancing interval in ECLIPSE 100.)

ECLIPSE 100

You can supply a target value and a maximum permitted value for largest branch balancing error, in the NETBALAN keyword. ECLIPSE 100 attempts to adjust the time step length so as to keep the balancing error at approximately its target value, subject of course to any other constraints on the time step. Time steps limited by the target balancing error can be identified by the mnemonic 'NETW' printed in the time step information line. If the maximum permitted balancing error is exceeded, ECLIPSE 100 chops the time step and rebalance the network. Since time step chops can severely increase the running time of a simulation, a maximum permitted balancing error should only be used with caution, and its value should be several times larger than the target error. A large balancing error is sometimes unavoidable during the time step in which a well dies. You can set a minimum time step size to prevent a large reduction in step size when wells die.

Sub-sea completion manifolds

ECLIPSE 100

If group production control is going to be applied, and some groups represent sub-sea completion manifolds in which the wells all operate at the same THP, then these groups should be declared as manifold groups in keyword GRUPNET. If group control is going to be applied to any group superior to a manifold group, the subordinate manifold groups should be given guide rates in keyword GCONPROD. If a manifold group has a rate target (either imposed directly or allocated as a share of a superior group's rate target in proportion to its guide rate), ECLIPSE calculates the manifold pressure required to make the group produce at its target rate. The calculation is performed iteratively, using a secant method. The convergence tolerance and the maximum number of iterations allowed for this calculation can be set in keyword NETBALAN. The manifold pressure is recalculated at each of the first NUPCOL Newton iterations of each time step, and the wells' THP limits are set equal to the manifold pressure or the group's nodal pressure in the network obtained from the latest network balancing calculation, whichever is the greater. If the time step takes more than NUPCOL Newton iterations to converge, the well THP limits are kept constant for the remaining iterations after the first NUPCOL, and the group may not meet its flow target within the allowed convergence tolerance. This treatment is in keeping with the standard group control method (see keyword GCONTOL).

ECLIPSE 300

The treatment of manifold groups in ECLIPSE 300 is different, in that they are regarded as inlets to automatic chokes (see "Automatic chokes" on page 695). The group's outlet branch contains the choke, and must be given a VFP table number of 9999 so its only pressure drop is that of the choke. ECLIPSE adjusts the pressure drop across the choke to keep the production rate at the required target, provided that it has enough potential.

Keywords required

The keywords needed in the SCHEDULE section to run the network option are summarized below:

Table 45.1 SCHEDULE section keywords

Notes	Keyword	Description
	VFPPROD	Enter the VFP tables for the production wells and the production network pipelines.
	VFPINJ	Enter the VFP tables for the injection wells and the injection network pipelines.
	WELSPECS	Enter the specification data for all wells.
	COMPDAT	Specify the connection data for all wells.
	WCONPROD	Set the control targets and limits for the production wells. THP limits should not be set for wells belonging to a group which is a node in the production network, as they will be determined at each balancing calculation.
	WCONINJE	Set the control targets and limits for the injection wells. THP limits should not be set for wells belonging to a group which is a node in the injection network, as they will be determined at each balancing calculation.
	GRUPTREE	Construct the grouping tree, if there is more than one level of groups below the field. The tree shown in Figure 45.3, for example, is constructed by the following records

Table 45.1 SCHEDULE section keywords

Notes	Keyword	Description
	GRUPNET	Define the production network. For example, in Figure 45.3, let PLAT-A and PLAT-B have fixed pressures of 300 PSIA and the bottom level groups be sub-sea completion manifolds connected to the platforms by pipelines
	GNETINJE	Define an injection network. For example, in Figure 45.3, let PLAT-B have a fixed pressure of 14.7 PSIA and GR-B2 contain the water injectors. The water injection pipeline between these two nodes is represented by injection VFP table number 2, which includes the pump performance characteristics
	GCONPROD	You can, optionally, use this keyword to set production rate targets and limits for any group. For example, the FIELD has an oil production target of 50,000 STB/DAY, while the two platforms have water handling limits of 10,000 STB/DAY. Because the four well-groups are manifold groups subordinate to a group under rate control, they should be given guide rates. Here they are each required to produce the same amount of liquid, so their liquid guide rates are set to the same nominal value - say 100
ECLIPSE 100 only	GNETPUMP	Use if automatic pump/compressor switching is required in the production network. For example, the pipeline from GR-B1 to PLAT-B will be interpolated with an artificial lift quantity of 50 if the oil production rate of GR-B1 falls below 500 STB/DAY:
	NETBALAN	Use of this keyword is optional. It sets the balancing interval, convergence tolerances and iteration limits, and instructions to limit the balancing error by reducing the time step size. In this example, we set a negative balancing interval so that ECLIPSE 100 balances the network in each of the first NUPCOL Newton iterations of every time step (ECLIPSE 300 always does this)
ECLIPSE 100 only	WAITBAL	Optional keyword to prevent Production Rule actions being performed to maintain group production targets until the network is balanced (ECLIPSE 300 always prevent this happening)
	WNETDP	Optional keyword to apply an additional fixed pressure drop between a well's tubing head and its group's node in the network

Examples

GRUPTREE example.

```
GRUPTREE
 'GR-A1' 'PLAT-A' /
 'GR-A2' 'PLAT-A' /
 'GR-B1' 'PLAT-B' /
 'GR-B2' 'PLAT-B' /
/
```

GRUPNET example

```
GRUPNET
-- GROUP   FIXED    VFP  PUMP   MANIFOLD     INCLUDE    ALQ=
-- NAME   PRESSURE  TAB  POWER   GROUP?     LIFT GAS?   DENS?
 'PLAT-*'  300 /
 'GR-A1'   1*        1    1*      'YES'  /
 'GR-A2'   1*        2    1*      'YES'  /
 'GR-B1'   1*        3    1*      'YES'  /
 'GR-B2'   1*        3    1*      'YES'  /
/
```

Note Note that in ECLIPSE 300 the manifold groups must have VFP table numbers of 9999, so a set of intermediate groups must be declared whose outlet pipelines have their pressure drops determined by the VFP tables 1, 2 and 3

GNETINJE example..

```
GNETINJE
-- GROUP  NETWORK  FIXED     VFP
-- NAME   PHASE    PRESSURE  TAB
 'PLAT-B' 'WATER'  14.7          /
 'GR-B2'  'WATER'  1*         2  /
/
```

GCONPROD example

```
GCONPROD
--
 GROUP   CNTL   OIL   WATER   GAS   LIQU   LIMIT  AVAIL  GUIDE  G.R.
--
 NAME    MODE   RATE   RATE    RATE   RATE  ACTION  FLD  RATE  PHASE
  'FIELD'  'ORAT'  50000   /
  'PLAT*'  'NONE'   1*    10000    2*           'RATE'   /
  'GR*'    'NONE'   5*                                   'YES'  100
  'LIQ' /
  /
```

GNETPUMP example:.

```
GNETPUMP
-- GROUP   RATE   PHASE    NEW VFP   NEW LIFT  NEW GAS
-- NAME    LIMIT           TAB NO    QUANTITY  CONSUMPTION
   'GR-B1' 500    'OIL'    1*        50                /
   /
```

NETBALAN example

```
NETBALAN
-1.0 /
```

WAITBAL example

```
WAITBAL
'YES' /
```

WNETDP example:

```
WNETDP
-- WELL      PRESSURE
-- NAME        DROP
  'PROD1'      20.0  /
  'PROD2'      25.0  /
  /
```

The extended network model

The Extended Network model contains several enhancements over the Standard model for production networks:

1 The surface network can be structured independently of the group control hierarchy defined with keyword GRUPTREE. However, it must still have a “gathering tree” topology.

ECLIPSE 100

2 The automatic compressor/pump facilities are more versatile.

Compressors are turned on whenever a nominated group fails to meet its production rate target. Multiple compression levels can be defined.

3 An automatic choke facility can adjust the pressure loss across a choke in a designated network branch to meet a group’s production rate target. This provides an alternative method to guide rate control or prioritization for meeting group production rate targets.

4 It provides more flexibility in handling well and group downtime.

5 It is possible to remove a specified rate or fraction of the water flowing through a node.

ECLIPSE 100

In ECLIPSE 100 the Extended Network model applies to the production network only. However, it is possible to use it in conjunction with the Standard model for an injection network.

ECLIPSE 300

In ECLIPSE 300 the Extended Network model applies to both production and injection networks. The two types of model cannot be mixed, so either all networks must use the Extended model or all must use the Standard model.

The network structure

It is sometimes necessary to impose a group control hierarchy that is different from the physical structure of the surface network. For example, consider a case where production from several groups feeds into a main trunkline at different points. This is shown schematically in Figure 45.4. GA1, GB1, GA2 and GB2 are all well-groups feeding into the trunkline at nodes N1, N2, N3 and N4 respectively. Let us assume that FIELD is the fixed pressure node.

Figure 45.4 The network structure

Now suppose that groups GA1 and GA2 belong to Company A whilst groups GB1 and GB2 belong to Company B. Clearly both companies may need to apply targets or limits to their own total production rate. This would be achieved by defining a group hierarchy with GRUPTREE as shown in Figure 45.5, which is different from the network structure.

Figure 45.5 The group control hierarchy

The group control hierarchy should be constructed as usual with keyword GRUPTREE, and the relevant production targets set with GCONPROD or GCONPRI. But by using the Extended Network Model, a separate network structure can be constructed with keywords BRANPROP and NODEPROP. The network consists of a gathering tree structure of nodes and branches. The nodes at the bottom level of the tree are "source nodes", which inject flow into the network. The source nodes must be well-groups or satellite groups (or master groups in Reservoir Coupling master runs), and all source nodes should correspond to groups in the GRUPTREE hierarchy; in ECLIPSE 100 the corresponding nodes and groups must have the same name in both structures but this is not necessary in ECLIPSE 300. The remaining network nodes and GRUPTREE groups need not coincide. The top node of the network tree (the terminal node) must be a fixed pressure node.

In the example above, GA1, GA2, GB1 and GB2 are common to both the network structure and the GRUPTREE hierarchy. Also the fixed pressure node corresponds to the FIELD group (although this is not compulsory). But nodes N1, N2, N3 and N4 only appear in the network structure, and groups COMP-A and COMP-B only appear in the group control hierarchy.

The network may consist of two or more separate gathering trees, in which case each tree must have its own fixed pressure terminal node at the top. It is possible for subordinate groups also to be fixed-pressure nodes; these form 'sub-networks' whose pressures are independent of the main network but whose flows will be added into it. For example, in the network shown in Figure 45.4 the nodes FIELD and N3 could both be fixed-pressure nodes. Node N3 then acts as a source into node N2 of the main network, while keeping its own pressure independent of the pressures in the main network.

A consequence of separating the group control hierarchy from the network structure is that the location of any gas consumption or import must be identified in both structures. ECLIPSE must know where consumed gas is extracted from the network (to compute downstream pressure losses correctly), and also where it is extracted from the GRUPTREE hierarchy (to perform the group production accounting). Accordingly the GCONSUMP keyword in ECLIPSE 100 has an additional item (Item 4) to identify the node at which the gas is extracted from or added to the extended network structure. Alternatively the keyword NCONSUMP can be used to define the consumption at a network node, and optionally to assign the consumption to a particular group. The corresponding keyword in ECLIPSE 300 for removing gas from the network is NGASREM.

Automatic compressors

ECLIPSE 100

Automatic compressors or pumps are defined with keyword NETCOMPA instead of GNETPUMP. Each is located at a specified branch of the network. They offer a much wider range of features than the GNETPUMP compressors of the standard network model. Their principal features are:

Activation by group target

Each compressor (or pump) is turned on whenever a nominated group fails to meets its gas (or oil) production rate target, which is specified with keyword GCONPROD or GCONPRI as usual. If the nominated group does not have a target set, but has a guide rate defined in GCONPROD and is under 'FLD' control from a higher level group, then the compressor is turned on when the nominated group cannot make its share of the higher level group's production target.

Compressor switching sequence

It is possible for more than one compressor to respond the same group. Each compressor can be given a sequence number, and when the group fails to meet its production target the compressor with the lowest sequence number that responds to the group will be activated. If the group still cannot meet its target, but has other compressors that respond to it, these other compressors are turned on in increasing order of their sequence number. All compressors in the network that share the same sequence number are turned on simultaneously.

Production rule order

If there is a drilling queue (keyword QDRILL), by default ECLIPSE scans it for a suitable well to open in preference to switching on a compressor when a group cannot make its production target. However, by using the PRORDER keyword to define the order in which various actions are taken to increase group production, compressor switching can be prioritized with respect to the other possible actions. For example, if 'COMP' is placed before 'DRILL', ECLIPSE activates the automatic compressors before looking for a new well to open.

Multi-level compression

Multi-level compressors can be defined. Compression is increased one level at a time until the nominated group can meet its production target. The ALQ, and the gas consumption rate if there is any, increase linearly with the level number to their "full on" values. Any number of levels may be requested, but having a large number of levels slows the run down as each level must be tried in turn by re-balancing the network.

Turning compression off

Compressors can be turned off (or reset to zero compression) manually at any time by using the COMPOFF keyword. They are turned on automatically again as soon as they are needed. For example, the COMPOFF keyword can be entered whenever the target production rate is reduced, or when extra production capacity is added. (When using the Gas Field Operations model, the compressors are turned off automatically whenever the contract group's target rate decreases.)

Automatic chokes

Automatic chokes form an alternative means of controlling a group's production rate when using the Extended Network option, which is distinct from guide rate control or prioritization. The choke is placed in a designated branch of the network, and at each network balancing iteration ECLIPSE adjusts the pressure loss across it to keep a specified group producing at its target rate. The operation of the choke varies the pressure in its inlet node, which in turn affects the productivity of the subordinate wells while they are operating under THP control.

Defining the choke

ECLIPSE 100

A branch is defined to be a choke by answering 'YES' to Item 3 of the NODEPROP keyword for the **inlet node** of the choke. A fixed-pressure node cannot be defined as a choke inlet node, since its pressure cannot respond to the choke's pressure loss. Also, source nodes cannot be defined as choke inlet nodes. Source nodes that are well-groups can instead be designated to be 'manifold groups', whose operation is similar to choke inlet nodes in that ECLIPSE determines a common well THP that makes the group meet its target rate (see "Group production control in the network option" on page 683).

ECLIPSE 300

A branch is defined to be a choke in a **production** network by answering 'YES' to Item 3 of the NODEPROP keyword for the **inlet node** of the choke. For an **injection** network this should be the **outlet** node of the choke; in either case it should be the node at the end of the branch that is nearer the reservoir. A fixed-pressure node cannot be defined as a choke inlet node, since its pressure cannot respond to the choke's pressure loss.

The only pressure drop allowed across a branch acting as a choke is that caused by the action of the choke itself. The branch should therefore be declared in the BRANPROP keyword with a VFP table of 9999. Any other table number is flagged as an error.

ECLIPSE 100

The wells subordinate to the choke respond to the choke pressure drop whenever they are operating under THP control. Thus, for the automatic choke to work correctly, its subordinate wells must be made to ignore any rate targets set for them by guide rate group control. Accordingly, ECLIPSE 100 automatically makes them unavailable for guide rate group control (this is not necessary in ECLIPSE 300), equivalent to setting Item 2 in the WGRUPCON keyword to 'NO'. This is done for all producers in source groups subordinate to the choke, as long as there is no intermediate fixed-pressure node between the source group and the choke. The wells can still be given individual flow rate and BHP limits with keyword WCONPROD, which causes them to come off THP control if they are violated.

ECLIPSE 100

Note If a producer subordinate to a choke is subsequently turned into an injector which is subject to group injection control, you must make the well available for group control again by entering the WGRUPCON keyword with 'YES' in Item 2 at the time that the well is redefined as an injector.

Setting the rate target

The operation of the choke is governed by the production rate target of a particular group, which is specified in Item 5 of keyword NODEPROP. If the inlet node to the choke shares the same name as a group, as declared with the GRUPTREE keyword, then Item 5 will default to that group. Obviously the choke's inlet node in the network and the nominated group in the GRUPTREE hierarchy must have a common set of subordinate wells, in order for the choke to influence the group's production rate.

The group's production rate target is set in the same way as for guide rate group control, using keyword GCONPROD (or GCONENG for an energy rate target). Alternatively the group's target may be passed down, using guide rates set with the GCONPROD keyword, as a share of a superior group's target. Thus the group must either have its own production rate target or have a guide rate set to give it a share of a superior group's target.

Since guide rate group control will not operate below a choke, groups subordinate to the nominated production rate target group must not be given production guide rates.

ECLIPSE 300

Injection networks that use the Extended model may also have chokes, which are governed by the injection rate target of a particular group, for the phase that corresponds to the network's injection phase. The group's injection rate target is set in the same way as for guide rate group control, using keyword GCONINJE. Alternatively the group's target may be passed down, using guide rates set with the GCONINJE keyword, as a share of a superior group's target. Thus the group must either have its own injection rate target or have an injection guide rate set to give it a share of a superior group's target. Since guide rate group control will not operate below a choke, groups subordinate to the nominated injection rate target group must not be given injection guide rates for the same phase.

Calculating the choke's pressure drop

Having defined the choke and its corresponding group's production rate target, ECLIPSE proceeds as follows. At each network balancing iteration, it compares the group's flow with its target. If the flow is too large ECLIPSE increases the inlet pressure of the choke to reduce the flow. If the flow is below the target the pressure at the inlet will be decreased until either the target is met or the inlet pressure is the same as the outlet pressure (that is there is no pressure drop across the choke). In this latter case ECLIPSE then looks to the production rules to see if the flow may be increased by drilling a new well, for example (see keyword PRORDER). The choke's pressure drop can be written to the SUMMARY file using the SUMMARY section keyword GPRB and naming the choke's inlet node.

ECLIPSE 300

Chokes in injection networks operate in a similar manner. The pressure in the choke's outlet node (nearer the reservoir) is iterated until the injection rate of the subordinate wells matches the injection target of the corresponding group. If the wells do not have sufficient injectivity to meet the target, the outlet node's pressure is kept the same as the inlet node's pressure (so there is no pressure drop across the choke) and the injection rate will fall below the target. ECLIPSE then scans the drilling queue to see if there is a suitable injector to drill to increase the group's injectivity. The choke's pressure drop can be written to the SUMMARY file using the SUMMARY section keyword GPRBW or GPRBG, for a water or gas injection network respectively, and naming the choke's outlet node.

ECLIPSE 100

The group targets are met to within a tolerance specified in Item 4 of the NETBALAN keyword. But unlike sub-sea manifold groups, the calculation of the pressure drops at chokes is calculated as a part of the main network iteration and hence the maximum number of iterations to solve for the choke is the same as for the network as a whole, i.e. it is governed by Item3 of NETBALAN and **not** Item 5. Whilst it is clearly dependent upon the relative sizes of the tolerances in Items 2 and 4 of NETBALAN, in general substantially more iterations are needed if there are chokes in the network than if there were not. You should be prepared to increase Item 3 of NETBALAN accordingly.

Note Convergence problems may arise if a group is given a very small rate target, either manually or as a result of a relatively small guide rate. This is particularly true if the low flow required from the wells under that group results in some of them being unable to operate under THP control.

Well and group efficiency factors

When wells have efficiency factors set in keyword WEFAC, by default each well contributes its 'time-average' production rate to the network flows, that is its flow rate multiplied by its efficiency factor. The network branch pressure losses are thus calculated from the time-average rate sums of the subordinate wells. This is equivalent to the way in which the group production rates are summed, and is most appropriate when the wells go down individually for short 'random' periods.

But if you wish to model the 'peak' pressure losses in the network, when all the wells are flowing at the same time, then it is possible to specify that the network branch flows should reflect the full flow rates of each well. By setting Item 3 of keyword WEFAC, it is possible to specify for each well individually whether its production rate should or should not be multiplied by its efficiency factor when adding its contribution to the network flows. The **group** flow rates, however, are still the time-average sum of the well flows.

ECLIPSE 100

If all the wells in a group have their downtimes synchronized, so that the group either flows with its full rate or not at all, it is more appropriate to apply an efficiency factor to the group (with keyword GEFAC) rather than the individual wells. If there is a network node with the same name as this group, by default the group's efficiency factor is applied to this node and its flow rate is multiplied by the efficiency factor when adding the node's flow rate to the flow in its parent node. The pressure drop in a node's outlet branch is calculated from the node's flow rate. Thus the node's outlet branch and the branches upstream of the node reflect the full flow rates, while the branches downstream of its parent node reflect the time-average flows. (This is equivalent to the Standard Network model's treatment of group efficiency factors.) However, by setting item 3 of keyword GEFAC it is possible to specify that the group's full flow rates should be used for calculating all downstream pressure losses, by not assigning the group's efficiency factor to the corresponding network node. If there is no network node with the same name as the group, it is possible to apply an efficiency factor to another appropriate network node using keyword NEFAC.

This flexibility is only available when using the Extended Network model, since network node and branch flows are accounted separately from the group flows. In the Standard Network option the branch flows are always the same as the reported group flow rates.

Using the extended network model

Below, we summarize how the use of the Extended Network model differs from that of the Standard model. Other keywords that are not mentioned here (for example NETBALAN) are common to both models.

RUNSPEC section

To enable the extended version of the network model, the keyword NETWORK must be entered in the RUNSPEC section. It has two items of data: the maximum number of nodes in the network and the maximum number of branches in the network. So, for the network in Figure 45.4 we might use

```
NETWORK
 9 8 /
```

SUMMARY section

The same keywords apply to both the Standard and Extended Network models. When used with the Extended version, names supplied to these keywords must correspond to network nodes instead of GRUPTREE groups.

Construct the network with BRANPROP and NODEPROP instead of GRUPNET

The network structure is defined in the SCHEDULE section by its branches. Each branch is identified by its inlet node and outlet node. The branches must first be defined with the keyword BRANPROP. This also sets the branch properties, that is the VFP table number and ALQ value. An example of how the production network in Figure 45.4 may be constructed is:

```
BRANPROP
-- Inlet   Outlet     VFP     ALQ    ALQ=
-- node     node     table   value   dens?
  'GB2'   'N4'          3 /
  'GA2'   'N3'          2 /
  'GB1'   'N2'          1 /
  'GA1'   'N1'       9999 /
  'N4'    'N3'          4 /
  'N3'    'N2'          4 /
  'N2'    'N1'          5 /
  'N1'    'FIELD'       6 /
/
```

After the network structure has been defined with BRANPROP, the properties of the network nodes must be defined with keyword NODEPROP. Only the fixed pressure node(s), and other nodes with non-default properties need to be specified here. In the example in Figure 45.4, the fixed pressure node corresponds to the FIELD group and is thus named 'FIELD'. Also let us assume that the source node GA1 corresponds to an automatic choke inlet (or manifold group in ECLIPSE 100) and there is no gas lift. The NODEPROP keyword is then:

```
NODEPROP
-- Node      Fixed    Choke inlet or      Add      Choke control
-- name     pressure  manifold group?   lift gas?       group
   'FIELD'    200.0             /
   'GA1'       1*         'Y'   /
/
```

Define any gas consumption or import with either GCONSUMP or NCONSUMP

ECLIPSE 100

Any gas consumption or import should be declared in both the network and the group hierarchy. This can be done with either keyword GCONSUMP or NCONSUMP. GCONSUMP declares consumption/import at a group, and allows you to specify the network node where this should be applied as well. NCONSUMP declares consumption at a network node, and allows you to specify the group where this should be applied as well. For a constant consumption rate, the two keywords are equivalent and it does not matter which you use. However, there are some differences between the two keywords:

- NCONSUMP allows only a constant consumption rate, while GCONSUMP also allows a consumption rate that is a fraction of the group's production, and also a gas import rate.
- The two keywords have different defaults for assigning consumption to the other structure. If consumption is declared at a source group in GCONSUMP, by default it will also be assigned to the corresponding source node of the network. However, if consumption is declared at a source node in NCONSUMP, by default it will **not** be assigned to the corresponding group.
- NCONSUMP may only be used with the Extended Network model. GCONSUMP may be used with either the Extended or Standard models.

Remove any fuel gas from the network with NGASREM

ECLIPSE 300

Any fuel gas consumed at a group (see keyword GRUPFUEL) by default still remains present in the network flow and contributes to the downstream branch pressure drops. But it is possible to remove a corresponding amount of gas from the network at a particular node by using the NGASREM keyword. The gas removal maybe defined as a specified rate or a fraction of the gas flow through the node.

ECLIPSE 100

Define any automatic compressors with NETCOMPA

Compressors or pumps can be modeled 'manually' by re-entering the BRANPROP keyword with a different ALQ value or VFP table number. However, more versatile automatic compressors can be defined using the NETCOMPA keyword. (Do not use GNETPUMP for these compressors.) They will be turned on automatically when a nominated group fails to meet its gas or oil production target set in GCONPROD or GCONPRI. If other actions to increase the group's production rate may also be performed (for example, opening a new well from the drilling queue), the order in which the possible actions are performed can be defined with the PRORDER keyword.

The following example defines two compressors responding to the gas production rate of groups 'PL-A' and 'PL-B' respectively. The second one has three compression levels, and consumes gas from the network and from group 'G-B3' in the GRUPTREE hierarchy.

```
NETCOMPA
--
 Inlet Outlet  Target  Phase  VFP  ALQ  Cons  Cons  Type   Num   ALQ
   Seq
--
 node   node   group       tab    rate  group        lev  lev 1  num
   'N4'   'N3'  'PL-A'  'GAS'   1*   50.0   1*   1*   'TEMP'    /
   'N6'   'N5'  'PL-B'  'GAS'   1*  100.0 1000 'G-
B3' 'MULT'  3   40.0   /
/
```

Automatic compressors can be turned off manually at any time by using the COMPOFF keyword (except for those defined in NETCOMPA to stay on permanently). They are turned on automatically again as soon as they are needed.

Defining a choke

In Figure 45.4 suppose we wish to control the flow of the FIELD group using a choke. This is done by introducing a new node, NCHOKE, to represent the choke inlet, between the nodes FIELD and N1. The single entry in BRANPROP for the branch N1 - FIELD is now replaced with two records. The choke itself is represented by the branch NCHOKE - FIELD, and this must have a VFP table of 9999 so that its only pressure loss is that across the choke. The branch N1 - NCHOKE accounts for the pressure loss along the rest of the pipe:

```
BRANPROP
-- Inlet   Outlet     VFP     ALQ
-- node     node     table   value
  .  .
  .  .
  'N1'     'NCHOKE'  6 /
  'NCHOKE' 'FIELD'   9999/
/
```

The node NCHOKE is declared to be a choke inlet in the NODEPROP keyword, and the choke is told to respond to the rate target of the group FIELD.

```
NODEPROP
-- Node      Fixed    Choke inlet or      Add      Choke control
-- name     pressure  manifold group?   lift gas?      group
  'NCHOKE'     1*          'YES'           'NO'        'FIELD'/
/
```

The FIELD's rate target is set in the usual way as for guide rate group control.

The network model in ECLIPSE 300

There are some important differences between the implementations of the network model in ECLIPSE 100 and ECLIPSE 300. However, the ECLIPSE 300 model can emulate much of the ECLIPSE 100 model's capability and shares a common set of keywords.

Standard and Extended network models

ECLIPSE 300 can run either the Standard model or the Extended model. When constructing a network model from scratch, you are recommended to use the Extended Network model as this is more versatile than the Standard model. The Standard model keywords, however, are available to assist the porting of ECLIPSE 100 datasets.

Unlike ECLIPSE 100, ECLIPSE 300 allows you to use the Extended model for injection networks as well as production networks. The two types of model cannot be mixed, so if you use the Extended model for the production network then you must also use it for any injection networks. This requirement differs from ECLIPSE 100, where the injection networks have to use the Standard model even if the Extended model is used for the production network. Thus the data file for runs containing both production and injection networks cannot be ported directly between the simulators if the production network uses the Extended model.

When using the Extended network model for both production and injection networks, node names must be unique across all networks; an injection network node cannot have the same name as a node in the production network. However, a source node in the production network may represent the same group as a sink node in the injection network, if the group contains both producers and injectors. The corresponding nodes in the two networks must have different names; they cannot both have the same name as the well group (although one node may have the same name). Accordingly, ECLIPSE 300 allows source nodes (corresponding to well groups) to have a different name from their corresponding group.

Currently there are no automatic pump/compressor facilities in ECLIPSE 300.

Balancing the network

The network balancing calculation solves the branch flows and pressure losses across the network, and balances the flows into the network from the wells with the pressures at the source nodes, which are translated into THP limits for the wells. ECLIPSE 300 balances the combined network and reservoir system in each of the first `NUPCOL` Newton iterations at every time step. (The value of `NUPCOL` may be set using keyword `NUPCOL` or keyword `GCONTOL`.) During any subsequent iterations that are required to converge the time step, the nodal pressures and well THP limits remain unchanged. Thus the balancing error at the end of the time step reflects only the changes in reservoir conditions that have taken place after the `NUPCOL`'th Newton iteration. ECLIPSE 100 allows you to reduce the frequency of the balancing iterations or to balance the network only during the first iteration of the time step; either of these options would increase the balancing error and so they are not implemented in ECLIPSE 300.

The default convergence criterion (item 2 of keyword `NETBALAN`) is substantially tighter in ECLIPSE 300 because the balancing calculation is more advanced. The branch flows and pressure losses are solved in an inner iteration loop nested inside the iterations that balance the network with the reservoir. The inner loop is converged to a tolerance that is a factor of 10 tighter than the tolerance for the network/reservoir coupling iterations.

The total number of network/reservoir coupling iterations performed over each time step is reported in the time step summary table in the column headed `NIT`.

Another difference with ECLIPSE 100 is that ECLIPSE 300 automatically activates the WAITBAL instruction, so that a new well from the drilling queue is drilled only after the network/reservoir iterations have converged. In ECLIPSE 100 it is necessary to activate the instruction with the WAITBAL keyword, otherwise wells may be drilled prematurely before the network is properly balanced.

Rate constraints and automatic chokes

ECLIPSE 300 has the option of meeting group rate targets by applying a pressure differential across an automatic choke, similarly to the Extended network model in ECLIPSE 100 (see "Automatic chokes" on page 695). However, there are some differences between the two simulators:

- ECLIPSE 100 uses a ‘sub-sea manifold’ model to handle rate constraints for well groups via the network (it iterates to find a common well THP that produces the required total rate). ECLIPSE 300, on the other hand, does not use this approach; it uses automatic chokes to handle rate constraints at all levels of the node or group hierarchy, including well groups. While this unifies the handling of rate constraints for all levels of the hierarchy, it may require some data changes to be made when porting an ECLIPSE 100 data set to ECLIPSE 300. If a well group acts as a sub-sea manifold in ECLIPSE 100, the pressure losses in its outlet pipeline can still be represented by a VFP table. When this becomes an automatic choke in ECLIPSE 300, its outlet branch must be given a VFP table number of 9999 so the only pressure differential along it is that of the choke. In order to include the pressure losses in the rest of the pipeline it will be necessary to create an intermediate network node to split the pipeline into two sections: one section to represent the choke with a VFP table number of 9999, and the other section to represent the rest of the pipeline with the well group’s original VFP table number.
- ECLIPSE 300 can apply automatic chokes to handle rate constraints in injection networks using the Extended network model. Extended injection networks are not available in ECLIPSE 100.
- When using the Standard Network model, ECLIPSE 100 can only handle production rate constraints via the network at well groups, by declaring them as sub-sea manifolds. In the Standard network model of ECLIPSE 300, production rate constraints may be handled through the network at any level of the group hierarchy by automatic chokes.
- Automatic chokes in ECLIPSE 300 can apply constraints on the flow of oil, water, liquid (oil + water) and gas directly. Other constraints (reservoir fluid volume rate, wet gas rate) are converted into constraints on these quantities using the latest available ratios, and so may not be met exactly.

Summary of keywords

RUNSPEC section

The keywords are:

Table 45.2 RUNSPEC section

Keyword	Description
NETWORK	Indicates that the Extended Network model is to be used, and sets array dimensioning data. The keyword should not be entered if the 'standard' network model is to be used for the production network.

SUMMARY section

The following keywords control output of data specific to the Network option.

When using the Extended Network Model, the list of names following these keywords must correspond to network nodes instead of GRUPTREE groups.

Table 45.3 Network SUMMARY output controls (ECLIPSE 100)

Group	Information
GPR	Group or node PRessure in the production network
GPRG	Group or node PRessure in the gas injection network
GPRW	Group or node PRessure in the water injection network
GPRB	Pressure drop along the group's or node's outlet branch in the production network
GPRBG	Pressure drop along the group's or node's inlet branch in the gas injection network
GPRBW	Pressure drop along the group's or node's inlet branch in the water injection network
GALQ	ALQ in the group's or node's outlet branch in the production network. No units are given, as there are several possible definitions for this quantity.
GOPRNB	Oil flow rate along the group's or node's outlet branch in the production network
GWPRNB	Water flow rate along the group's or node's outlet branch in the production network
GGPRNB	Gas flow rate along the group's or node's outlet branch in the production network
GWIRNB	Water flow rate along the group's or node's inlet branch in the water injection network
GGIRNB	Gas flow rate along the group's or node's inlet branch in the gas injection network

Table 45.4 Network SUMMARY output controls (ECLIPSE 300)

Group	Information
GPR	Group or node PRessure in the production network
GPRG	Group or node PRessure in the gas injection network
GPRW	Group or node PRessure in the water injection network
GPRB	Pressure drop along the group's or node's outlet branch in the production network
GPRBG	Pressure drop along the group's or node's inlet branch in the gas injection network
GPRBW	Pressure drop along the group's or node's inlet branch in the water injection network
GALQ	ALQ in the group's or node's outlet branch in the production network. No units are given, as there are several possible definitions for this quantity.
GOPRNB	Oil flow rate along the group's or node's outlet branch in the production network
GWPRNB	Water flow rate along the group's or node's outlet branch in the production network
GGPRNB	Gas flow rate along the group's or node's outlet branch in the production network
GLPRNB	Liquid flow rate along the group's or node's outlet branch in the production network
GWIRNB	Water flow rate along the group's or node's inlet branch in the water injection network
GGIRNB	Gas flow rate along the group's or node's inlet branch in the gas injection network

SCHEDULE section

The keywords are:

Table 45.5 SCHEDULE section

Simulator	Keyword	Description
	BRANPROP	Defines the branches in an Extended Network.
ECLIPSE 100 only	COMPOFF	Switches off automatic compressors defined by NETCOMPA in the Extended Network model or GASFCOMP in the Gas Field Operations model.
ECLIPSE 100 only	GNETDP	Dynamically adjusts the pressure of a fixed-pressure node with the aim of maintaining its production flow within specified limits. If the network is an Extended network, the automatic choke facility (item 3 of keyword NODEPROP) provides a better method of controlling the flow rate.
	GNETINJE	Defines the structure of a Standard model injection network.
ECLIPSE 100 only	GNETPUMP	Defines automatic compressors or pumps in the Standard model production network. If the network is an Extended Network, use keyword NETCOMPA instead
	GRUPNET	Defines the structure of a Standard model production network.
ECLIPSE 100 only	NCONSUMP	Specifies gas consumption rates at Extended Network nodes.
ECLIPSE 100 only	NEFAC	Specifies efficiency factors for Extended Network nodes.
	NETBALAN	Sets parameters for the network balancing calculation.
ECLIPSE 100 only	NETCOMPA	Defines automatic compressors in the Extended Network.

Table 45.5 SCHEDULE section

Simulator	Keyword	Description
ECLIPSE 300 only	NGASREM	Removes gas from a node in the Extended Network.
	NODEPROP	Defines node properties in the Extended Network.
	NWATREM	Removes water from a node in the Extended Network.
ECLIPSE 100 only	OPTIONS	Switch 33 sets the number of network balancing iterations in which a well can be revived after being closed because it cannot operate against its current THP. Switch 57 invokes an alternative procedure for handling automatic choke calculations when a low flow target has been set.
ECLIPSE 300 only	OPTIONS3	Switch 47 sets the number of network balancing iterations in which a well can be revived after being closed because it cannot operate against its current THP. Switch 51 controls the frequency of re-calculating the well potentials, which depend on the THP limits obtained from the network. Switch 63 prevents extrapolation of the VFP tables for the branch pressure drop.
ECLIPSE 100 only	WAITBAL	Prevents automatic drilling and other production rule actions being performed until the network is balanced.
ECLIPSE 100 only	WNETCTRL	Selects whether well THP limits or rate limits should be held constant after the network has been balanced.
	WNETDP	Applies an additional fixed pressure drop between the tubing head of a well and the corresponding network node of its group.

Nine-point Schemes

Chapter 46

Introduction

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

The default discretization in ECLIPSE employs a five-point discretization in two dimensions or a seven point discretization in three dimensions. However, when the grid is non-orthogonal, or the flow is not aligned with the principal directions of the grid, then the solution accuracy may be compromised by this discretization. ECLIPSE uses a nine-point scheme to address this issue.

There are two nine-point schemes in ECLIPSE that address grid non-orthogonality errors and grid orientation effects only. The two nine-point schemes should not be used together.

The NINEPOIN scheme reduces the effect of grid orientation effects for flows not aligned with the principal grid directions (such as diagonally across the grid), but does not account for non-orthogonality errors.

ECLIPSE 300

The Multi Point Flux Approximation (MPFA) scheme is a rigorous treatment of grid non-orthogonality and tensor permeability. This leads to a nine-point scheme when the grid is non-orthogonal or the permeability tensor is not aligned with the coordinate directions. The MPFA nine-point scheme is activated by the MPFA keyword. Refer to "Tensor Permeability" on page 1015 for more details.

The nine-point scheme based on diagonal NNC transmissibilities

The NINEPOIN keyword in the RUNSPEC section activates a nine-point scheme based on adding diagonal transmissibilities in the areal (XY) direction in order to reduce grid orientation effects when the flow is not aligned with the grid. Additional non-neighbor connections are added between diagonal neighbors. The flows between each cell and its eight neighboring cells (in the areal dimension) are each computed using a two-point approximation to the pressure gradient. The scheme is a modified version of that given in [Ref. 39]. The standard five-point transmissibilities are first obtained from data entered in the GRID and EDIT sections, and these are then modified to obtain the nine-point transmissibilities.

To derive the nine-point scheme, we start by splitting each transmissibility in the usual five-point scheme into two half-transmissibilities:

Figure 46.1 Splitting each five-point transmissibility into two half-transmissibilities

Considering just the transmissibility values ($T12$ $T24$ $T13$ $T34$), we wish to replace these by the equivalent system ($t12$ $t13$ $t23$ $t24$ $t34$ $t14$). This will introduce a diagonal coupling between cells 1 and 4 and cells 2 and 3.

Figure 46.2 Deriving the nine-point transmissibilities using diagonal coupling between cells

We wish the values of ($t12$ $t13$ $t23$ $t24$ $t34$ $t14$) to preserve the total x and y transmissibilities for the pattern, so that

$$T13 + T24 = t13 + t24 + t23 + t14$$

$$T12 + T34 = t12 + t34 + t23 + t14$$

We also wish to avoid tensor effects, so that a pressure gradient in the x-direction does not produce a flow in the y-direction. This implies

$$t14 = t23$$

A choice for $t14$ may now be made. This should not be so large that one of the other transmissibility values is forced to be negative. Following Shiralkar and Stephenson [Ref. 41] we use

$$t13 = t24 = \frac{1}{3}\left[\frac{(T13 + T24)(T12 + T34)}{(T13 + T24 + T12 + T34)}\right]$$

The total x and y transmissibility conditions above then define the values of the combinations $sx = t13 + t24$ and $sy = t12 + t34$, for example

$$sx = t13 + t24 = T13 + T24 - \frac{2}{3}\left[\frac{(T13 + T24)(T12 + T34)}{(T13 + T24 + T12 + T34)}\right]$$

Finally, the ratio of $t13$ and $t24$ is chosen to be in the same ratio as the original values T13 and T24,

$$t13 = \frac{sx\ T13}{(T13 + T24)}$$

$$t24 = \frac{sx\ T24}{(T13 + T24)}$$

A similar relation defines $t12$ and $t34$. This then defines all the new transmissibilities. This is repeated for each quadrant of active cells. The result is a set of nine-point transmissibilities that preserves the original total x- and y-direction flows but allows diagonal coupling between cells.

Within the code, the transmissibilities that fall outside the normal pattern are stored as non-neighbor connections. There is an extra cost associated with these terms, and the nine-point option is best used only for injection of highly adverse mobility fluids where the grid orientation effect is a problem.

The option is invoked by using the `NINEPOIN` keyword in the `RUNSPEC` section. The nine-point transmissibilities will then by default be calculated across the entire grid. In ECLIPSE 100, by using the `NINENUM` keyword in the `GRID` section, you can choose to apply the scheme only in nominated regions of the reservoir. Note that the `NINENUM` keyword is not compatible with the Local Grid Refinement option.

It should be noted that this option is specifically designed to account for grid orientation effects and is not designed to deal with discretization errors arising from grid non-orthogonality.

This nine-point scheme based on diagonal NNC transmissibilities is invoked by the `NINEPOIN` keyword in the `RUNSPEC` section, for example:

```
RUNSPEC
..
NINEPOIN
```

The run time will be increased due to the large number of additional NNCs created between diagonal neighbors.

The nine-point transmissibilities can be obtained from the ‘`TRANX`’, ‘`TRANY`’, ‘`TRANZ`’ and ‘`ALLNNC`’ switches in the `RPTGRID` keyword.

It is possible to apply the `NINEPOIN` scheme only in the XZ or YZ planes with the `NINEXZ` and `NINEYZ` keywords.

Non-Darcy Flow

Chapter 47

Introduction

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

The non-Darcy feature we consider here is the Forchheimer correction; which takes into account the inertia effects due to high velocity that may occur in high permeability regions, such as fractures. The Forchheimer parameter can be input by using either the VDFLOW or VDFLOWR keywords in the PROPS section. For a homogeneous reservoir the model writes (cgs units):

$$\frac{dP}{dx} = \left(\frac{\mu}{Kk_rA}\right)q + \beta\rho\left(\frac{q}{A}\right)^2 \qquad \text{[EQ 47.1]}$$

where:

q is the volumetric flow rate

K is the rock permeability

k_r is the relative permeability

A is the area through which flow occurs

μ is the fluid viscosity

ρ is the fluid density

β is the Forchheimer parameter

dP/dx is the pressure gradient normal to the area

The Forchheimer parameter β is the user input in Forchheimer unit: 1F = 1 atm.s^2.g^{-1}. An alternative unit for β can be obtained by noting that: 1F = 1 atm.s^2.g^{-1} = 1.01325.10^6 g.cm.s^{-2}.cm^{-2}.s^2.g^{-1} = 1.01325.10^6 cm^{-1}.

Typical values in cm^{-1} are rather large for engineering purposes, and ECLIPSE 300 uses the Forchheimer (F) in all unit sets. Dake [Ref. 6] shows a typical value of $\beta = 10^7 \text{cm}^{-1} = 9.86\,F$ at $K = 60mD$.

For other unit sets (FIELD, METRIC, LAB and PVT) two constants C_1 and C_2 are introduced as

$$\frac{dP}{dx} = \frac{1}{C_1}\left(\frac{\mu}{Kk_rA}\right)q + \frac{C_2}{C_1}\beta\rho\left(\frac{q}{A}\right)^2 \qquad \text{[EQ 47.2]}$$

When considering the flow between two grid blocks i and n we use transmissibility. For the sake of simplicity we use the block centered transmissibility (`OLDTRAN`) to demonstrate. Thus

$$q = C_1F_{ND}T_{in}\frac{k_r}{\mu}\Delta P_{in} \qquad \text{[EQ 47.3]}$$

where

$$T_{in} = \frac{A}{\frac{\Delta X_i}{2K_i} + \frac{\Delta X_n}{2K_n}} \qquad \text{[EQ 47.4]}$$

and where the **non-Darcy flow** factor is given by:

$$F_{ND} = \frac{1}{1 + C_2\rho\frac{T_{in}k_r}{A^2\mu}\frac{1}{2}(\beta_i\Delta X_i + \beta_n\Delta X_n)q} \qquad \text{[EQ 47.5]}$$

Note that the F_{ND} factor is dimensionless and gives a quantitative indication of the deviation of the flow behavior from Darcy's law for which $F_{ND} = 1$.

Unit conversion factors

Denoting by a star index other unit sets and taking advantage of some variables being expressed in the same units whatever the unit system (such as the viscosity and the β factor), the constants C_1 and C_2 are obtained as:

$$C_1 = \frac{\Delta P_{cgs}}{\Delta P^*}\frac{\Delta X^*}{\Delta X_{cgs}}\frac{K_{cgs}}{K^*}\frac{A_{cgs}}{A^*}\frac{q^*}{q_{cgs}} \qquad \text{[EQ 47.6]}$$

$$C_2 = \frac{\rho_{cgs}}{\rho^*}\frac{q_{cgs}}{q^*}\left(\frac{\Delta X_{cgs}}{\Delta X^*}\right)^2\frac{K_{cgs}}{K^*}\left(\frac{A^*}{A_{cgs}}\right)^2 \qquad \text{[EQ 47.7]}$$

Table 47.1 gives the numerical values of these unit conversion factors.

Table 47.1 Unit conversion factors C1 and C2

	Metric	Field	Lab	PVT-Metric
C_1	0.00852702	0.00112712	3.6	0.00864
C_2	1.1574074E-9	3.17277463E-8	2.77777778E-7	1.1574074E-9

Construction of flows

From the equation for F_{ND} we have two coupled equations to solve for each phase p:

$$F_{ND} = \frac{1}{1 + C_2 \rho_p \frac{T_{in}}{A^2} \frac{k_{rp}}{\mu_p} \frac{1}{2} (\beta_i \Delta X_i + \beta_n \Delta X_n) q} \qquad \text{[EQ 47.8]}$$

and

$$q = C_1 F_{ND} T_{in} \left(\frac{k_{rp}}{\mu_p} \right) \Delta P_{pin} . \qquad \text{[EQ 47.9]}$$

Substituting q into F_{ND} we obtain a quadratic equation for F_{ND}

$$B F_{ND}^2 + F_{ND} - 1 = 0 \qquad \text{[EQ 47.10]}$$

that can be solved to give (the negative root being ignored)

$$F_{ND} = \frac{-1 + \sqrt{1 + 4B}}{2B} \qquad \text{[EQ 47.11]}$$

where for simplicity we put

$$B = C_2 C_1 \rho_p \left(\frac{T_{ni}}{A} \right)^2 \left(\frac{k_{rp}}{\mu_p} \right)^2 \frac{1}{2} (\beta_i \Delta X_i + \beta_n \Delta X_n) \Delta P_{pni} \qquad \text{[EQ 47.12]}$$

Assumption and simplifications

As the Forchheimer correction is expected to be significant only in regions of high velocity, it is assumed that those regions have a constant rock permeability K and constant porosity ϕ. Under these assumptions the B-factor can be simplified to

$$B = C_2 C_1 \beta \rho_p \left(\frac{T_{ni} K}{A} \right) \left(\frac{k_{rp}}{\mu_p} \right)^2 \Delta P_{pni} \qquad \text{[EQ 47.13]}$$

Using the fact that in ECLIPSE the factor unit C_1 is included in the transmissibility ($T_{ni} = C_1 T_{ni}$), we obtain the final form for the B-factor used in ECLIPSE 300

$$B = C_2 \beta \rho_p \left(\frac{T_{ni} K}{A} \right) \left(\frac{k_{rp}}{\mu_p} \right)^2 \Delta P_{pni} \qquad \text{[EQ 47.14]}$$

The values of F_{ND} and $B/(\Delta P_{pni})$ for gas and oil may be examined using the mnemonics `FFOR(G,O)` and `BFOR(G,O)` in the `RPTSOL` or `RPTSCHED` keywords.

Note In the current version ECLIPSE 100 reports `FFORG` and `BFORG`, that is for the gas phase.

Forchheimer models

In the above discussion the Forchheimer parameter (β) is a constant supplied by the user using the keywords VDFLOW or VDFLOWR. This parameter may alternatively be calculated from one of two models implemented in ECLIPSE 300. The choice of model is made using the VELDEP keyword in the RUNSPEC section, with the model parameters supplied using the VDKRG and VDKRO keywords in the PROPS section.

For each phase p , the models are:

ECLIPSE 300 only

- Model 1:

$$\beta_p = \frac{a_p}{\phi^{b_p} S_p^{c_p} (Kk_{rvp})^{d_p}} \qquad \text{[EQ 47.15]}$$

ECLIPSE 300 only

- Model 2:

$$\beta_p = \beta_{dp} S_p^{c_p} (Kk_{rvp})^{d_p} \qquad \text{[EQ 47.16]}$$

where S_p is the normalized saturation of the pth phase, which is related to the true saturation $S_p^{(3P)}$ by:

$$S_p = \frac{S_p^{(3P)}}{1 - S_w} \qquad \text{[EQ 47.17]}$$

where S_w is the water saturation. The set of parameters (a_p, b_p, c_p, d_p) or (β_{dp}, c_p, d_p) are, **and must be**, experimentally determined. There appears to be no real physical basis for the porosity and saturation dependency in [EQ 47.15] and saturation dependency in [EQ 47.16] other than in the original work done on the Forchheimer effect. These dependencies may be ignored by setting the values of (b_p, c_p) in the VDKRO and/or VDKRG keywords in the PROPS section to zero.

The values of β for gas and oil may be examined using the mnemonics BTFOR(G,O) in the RPTSOL or RPTSCHED keywords.

Forchheimer parameter based well D-factors

If a Forchheimer parameter for the gas phase is calculated using one of the above described models, then a D-factor for each production well connection is also calculated.

Note A D-factor is **not** calculated if a constant Forchheimer parameter was supplied using the keywords VDFLOW or VDFLOWR.

Calculated D-factors override D-factors supplied using the keywords COMPDAT or WDFAC. If the WDFACCOR keyword is specified, however, then the value calculated from its correlation overrides the value calculated from [EQ 47.18]. Calculated D-factors for well connections may be written to the summary file using the mnemonic CDFAC.

The equation for the Forchheimer parameter dependent D-factor is as follows:

$$D = \alpha \cdot \beta_g \cdot \frac{K_e}{h} \cdot \frac{1}{r_w} \cdot \frac{\gamma_g}{\mu_g} \qquad \text{[EQ 47.18]}$$

where

α is a constant, the value of which depends on the units system (see Table 47.2)

β_g is the Forchheimer parameter for gas (in Forchheimer units, F)

K_e is the effective permeability of the connected grid block (generally the geometric mean of permeabilities involved)

h is the length of the connection

r_w is the wellbore radius

γ_g is the relative density of produced or injected gas at surface conditions with respect to air at standard temperature and pressure

μ_g is the gas viscosity

Table 47.2 Unit-dependent constants for the Forchheimer parameter-dependent skin-factor

	Metric	Field	Lab	PVT-Metric
Unit	day / sm^3	day / MScf	hr / scc	day / sm^3
Value	2.24460×10^{-10}	6.83352×10^{-8}	5.41375×10^{-11}	2.25533×10^{-10}
Pre-2006.1 value	2.70863×10^{-18}	7.67000×10^{-17}	6.50072×10^{-23}	2.70863x10-18

The values of the unit-dependent constant were changed at 2006.1. The pre-2006.1 values are still available for back-compatibility. To revert to the pre-2006.1 values, set item 96 of the `OPTIONS3` keyword to 2 or 3.

Note that [EQ 47.18] uses an approximation based on the assumption that the pressure equivalent radius, r_0, of the well is much greater than the wellbore radius. This may not be a good approximation with smaller grid sizes. [EQ 47.19], which does not use this approximation, can be used by setting item 96 of the `OPTIONS3` keyword to 1 or 3.

$$D = \alpha \cdot \beta_g \cdot \frac{K_e}{h} \cdot \left(\frac{1}{r_w} - \frac{1}{r_0}\right) \cdot \frac{\gamma_g}{\mu_g} \qquad \text{[EQ 47.19]}$$

Velocity-dependent relative permeabilities

At high gas velocities, in addition to the Forchheimer correction, it is also possible to take into account more complex effects by introducing a velocity dependence of the relative permeabilities [Ref. 118]. The model may make the oil and gas relative permeabilities functions of Capillary Number (CN) and/or the Forchheimer parameter, depending on the user-specified switches set using the VELDEP keyword in the RUNSPEC section. Properties of the velocity-dependent relative permeabilities for the oil and gas are set with the keywords VDKRO, VDKRG and VDKRGC.

These models are intended for gas condensate systems where the expected reduction in Productivity Index (PI) when a well bottom hole pressure drops below the fluid dew point pressure has not been observed in practise. Several theories have been suggested as to why the gas relative permeability in particular remains relatively high but the current consensus seems to imply that a combination of low InterFacial Tensions (IFT's) at reservoir conditions and high velocities as flow converges towards a producer are the main driving forces.

Normal engineering practise is to combine IFT and velocity through the so-called Capillary Number: several alternative definitions are possible and three are shown below. Of course, where flow velocities are high, turbulence effects as predicted by the Forchheimer model become significant and a model for this effect has been developed as part of this overall scheme. These models do not work with black oil runs since they are intended for gas condensate systems.

Capillary number model

The CN model has two effects on the gas and oil relative permeabilities, namely as the CN increases:

- It reduces the residual saturations
- It changes the relative permeability from the user-specified (immiscible) saturation curves towards an internally-generated miscible curve.

The CN is calculated from one of three alternate models. For phase p = (oil, gas)

- Capillary Number Model 1:

$$N_{cp}^{(1)} = \frac{v_g \mu_g}{\sigma} \qquad \text{[EQ 47.20]}$$

- Capillary Number Model 2:

$$N_{cp}^{(2)} = \frac{K k_{rvp} \Delta P_p}{\sigma L} \qquad \text{[EQ 47.21]}$$

- Capillary Number Model 3:

$$N_{cp}^{(3)} = (2\phi S_p K k_{rvp})^{1/2} \frac{\Delta P_p}{\sigma} \qquad \text{[EQ 47.22]}$$

where:

S_p is the normalized phase saturation (more details below)

ΔP_p is the pressure drop of the pth phase in the direction of flow

μ_g is the gas viscosity

v_g is the gas velocity in the direction of flow

σ is the gas-oil surface tension

k_{rvp} is the CN-modified relative permeability of the pth phase (more details below)

The CN-modified relative permeabilities k_{rvp} are calculated at the previous time-step. We will discuss this last point in more detail shortly.

Note Note that Capillary Number Model 1 for the oil and gas phases depends only on the gas velocity and gas viscosity. This apparent discrepancy has been verified by experiment.

Given a phase the CN calculated from [EQ 47.20], [EQ 47.21] or [EQ 47.22], and Normalized Capillary Number (NCN) are defined by:

$$N_{cnp} = \frac{N_{cbp}}{N_{cp}} \qquad \text{[EQ 47.23]}$$

where N_{cbp} is the phase Base Capillary Number (BCN). The phase BCN is a lower threshold value below which the CN has no effect on the phase relative permeabilities. As with all the parameters in this and the Forchheimer models, the phase BCN should be determined experimentally for the reservoir and fluid of interest. If this data is not available, it may be estimated using the following procedure.

Base capillary number estimation

Generally, the rock or base relative permeability curves used to describe the flow of gas and oil within the reservoir will have been generated at or near ambient conditions. Ideally, the phase CNs will have been measured from these laboratory analyses and are reported to you; in practise, this is rarely the case. At ambient conditions, the reservoir gas and condensate are as different in composition as it is possible to get; therefore, the gas-oil surface tension is likely to be its maximum. In addition, the low pressure gas viscosity is likely to be at its minimum.

Using PVT*i*, it should be possible to perform a Constant Volume Depletion (CVD) experiment on the reservoir fluid, starting from the dew point and ending at the likely abandonment pressure. The values of the gas-oil surface tension and gas viscosity can then be taken from this final stage pressure.

Away from the wells, fluid velocities rarely exceed 10 ft./day [~ 3.5×10^{-5} m/s]. This value for gas velocity along with the low pressure gas-oil surface tension and gas viscosity can be used in [EQ 47.20] to estimate the oil and gas BCN in the absence of experimental data.

The effect of CN on residual saturation

Experimental evidence suggests that as the CN rises above the BCN, the residual saturation is reduced. In the CN model implemented here, this effect is modeled through the use of the saturation scaling parameter, X_p, where:

$$X_p = 1 - \exp(-m_p N_{cnp}) \qquad \text{[EQ 47.24]}$$

where m_p is, **and must be**, an experimentally determined parameter. Therefore, given a phase residual saturation S_{rbp} determined from routine Special Core AnaLysis (SCAL), the effect of the CN dependency is to change the residual saturation to:

$$S_{rbp} \rightarrow X_p S_{rbp} \qquad \text{[EQ 47.25]}$$

Note if the m_p -parameter is set to zero, it has the effect of setting that phase residual saturation to zero regardless of the value of the phase CN.

The effect of CN on relative permeability curves

We can construct a miscible or straight-line relative permeability curve from:

$$k_{rmp} = \frac{(S_p - X_p S_{rbp})}{(1 - X_p S_{rbp})} \qquad \text{[EQ 47.26]}$$

where S_p is the normalized or HCPV saturation of the pth phase, which is related to the true saturation $S_p^{(3P)}$ by:

$$S_p = \frac{S_p^{(3P)}}{1 - S_w} \qquad \text{[EQ 47.27]}$$

where S_w is the water saturation.

The user-input base relative permeability curves, k_{rbp}, are modified because of the change in residual saturation through [EQ 47.27]. The CN-modified relative permeability curve for phase p is now calculated from:

$$k_{rvp} = N_{cnp}^{1/n_p} k_{rbp} + (1 - N_{cnp}^{1/n_p}) k_{rmp} \qquad \text{[EQ 47.28]}$$

where n_p is given by:

$$n_p = n_{1p} S_p^{n_{2p}} \qquad \text{[EQ 47.29]}$$

and (n_{1p}, n_{2p}) are, **and must be**, experimentally determined parameters.

Explicit calculation of capillary number dependency

From [EQ 47.20], [EQ 47.21] and [EQ 47.22], we see that the CN is a function of the phase relative permeability k_{rvp} : note [EQ 47.20] and [EQ 47.21] are effectively equivalent assuming the 2nd expression is only applied to the gas phase. However, from [EQ 47.24], [EQ 47.26] and [EQ 47.28], we see k_{rvp} is a function of CN.

It is possible to write this dependency as a non-linear equation that can be solved in a fully implicit scheme and it is planned to do this in a future release. For now, however, we calculate the phase CN and NCN at the end of the last time-step. This may have some stability implications for otherwise implicit completion cells (in Fully Implicit or AIM modes of operation). This is still an area of research.

Alternative model for capillary number effects

An alternative model for CN effects, based on the work of Whitson, Fevang and Saevareid [Ref. 99], is also available. This model is designed for near wellbore flows exhibiting condensate 'blockage' type behavior in a gas condensate reservoir. It may be activated via item 5 of the VELDEP keyword in the RUNSPEC section.

This model is more straightforward than the above in that:

- There is no effect on residual saturations
- There is no lower threshold or base capillary number value. (Below which the CN has no effect on the gas relative permeabilities.)
- It depends on only two parameters, both of which have default values.

In the implementation in ECLIPSE 300, the CN-modified gas relative permeability, k_{rg}, is an admixture of a straight-line miscible relative permeability, k_{rgM}, and the user-input immiscible, rock relative permeability, k_{rgI}; this admixture is controlled by a CN dependent transition function, f_I:

$$k_{rg} = f_I k_{rgI} + (1 - f_I) k_{rgM} . \qquad \text{[EQ 47.30]}$$

The calculation of the miscible relative permeability is discussed later. The transition function depends on the gas capillary number, N_{cg}, and is given by:

$$f_I = \frac{1}{(\alpha \cdot N_{cg})^n + 1} \qquad \text{[EQ 47.31]}$$

where

$$\alpha = \alpha^0 / \overline{k_{rg}} \text{ with } \overline{k_{rg}} = \frac{k_{rgM} + k_{rgI}}{2} \qquad \text{[EQ 47.32]}$$

and α^0 is a constant depending only on rock properties:

$$\alpha^0 = \frac{\alpha_c^{\,0}}{\sqrt{K \cdot \phi}} \qquad \text{[EQ 47.33]}$$

where K is the rock permeability and ϕ is the porosity. The gas capillary number is calculated from Model 1, [EQ 47.20], using a pore gas velocity:

$$v_{pg} = \frac{v_g}{\phi \cdot (1 - S_w)} \qquad \text{[EQ 47.34]}$$

This model depends on two parameters: the exponent n in [EQ 47.31]and the coefficient $\alpha_c^{\,0}$ in [EQ 47.33]. These parameters are defaulted to 0.65 and 1.0E4 respectively, with the keyword VDKRGC permitting you to override these values.

Calculation of the miscible gas relative permeability

In the well inflow generalized pseudo-pressure option, at each pseudo-pressure below the saturation pressure, the model iterates to find a gas saturation such that the ratio of the **immiscible** mobilities (that is, using immiscible relative permeability) of oil and gas is equal to the ratio of the total liquid moles (L) to total vapor moles (V). Hence

$$\frac{k_{roI}b_o}{\mu_o}\frac{\mu_g}{k_{rgI}b_g} = \frac{L}{V} \qquad \text{[EQ 47.35]}$$

The ratio of the immiscible relative permeabilities is calculated at this gas saturation,

$$R = k_{rgI}/k_{roI} \qquad \text{[EQ 47.36]}$$

Note that, in block-to-block flow calculations, the gas saturation is available so k_{rgI} and k_{roI} are calculated directly at that saturation in order to calculate the ratio R.

The miscible gas relative permeability is calculated from

$$k_{rgM} = k_{rg}^0\left\{\frac{1}{1+(1/R)}\right\} \qquad \text{[EQ 47.37]}$$

where k_{rg}^0 is the gas relative permeability at $S_g = 1 - S_{wco}$ (and S_{wco} is the connate water saturation).

Pre-2007.1 behavior

Pre-2007.1, the miscible gas relative permeability was calculated using a straight line curve of the form [EQ 47.26] but with no change in the residual saturation (that is, X_g has a value of unity). This behavior can be restored (instead of using [EQ 47.37]) by setting `OPTIONS3` item 99 to a non-zero value.

Calculation of the oil relative permeability

The calculation of the oil relative permeability depends on the value of item 5 in the `VELDEP` keyword.

- If `VELDEP` item 5 has a value of 1, then

  $$k_{ro} = f_{oI}k_{roI} + (1 - f_{oI})k_{roM}$$

  where the miscible oil relative permeability is of the form given by [EQ 47.26] but with no change in the residual saturation (that is, X_o has a value of unity), and the immiscible-miscible transition function, f_{oI}, is the same as that used in the original model for capillary number effects in oil flows (see [EQ 47.28]) so

  $$f_{oI} = N_{cnp}^{1/n_p}$$

  The calculation of f_{oI} requires n_{1o}, n_{2o} and N_{cbo} (and the selection of the capillary number model using `VELDEP` item 1). Hence it increases the number of parameters required by the alternative model from 2 to 5.

- If `VELDEP` item 5 has a value of 2, then

$$k_{ro} = k_{rg}/R$$

with R calculated using [EQ 47.36]. This is the preferred method for achieving the desirable property that the alternative model only depends on two parameters.

Pre-2007.1 behavior of the alternative model

Pre-2007.1, the alternative model did not use [EQ 47.35], but instead found a gas saturation such that the ratio of the **modified** mobilities of oil and gas is equal to the ratio of the total liquid moles (L) to total vapour moles (V). That is

$$\frac{k_{ro}b_o}{\mu_o}\frac{\mu_g}{k_{rg}b_g} = \frac{L}{V}$$

where b_p is the phase molar density and μ_p is the phase viscosity.

This pre-2007.1 behavior can be recovered by setting `OPTIONS3` item 99 to a non-zero value.

Notes

1 This alternative model for CN effects in gas flows cannot be used at the same time as the original CN model for **gas** flows. If `VELDEP` keyword item 5 is set to 1, then the alternative model can be used in conjunction with the original model for CN effects in **oil** flows. If `VELDEP` keyword item 5 is set to 2, then it cannot be used with the original model. The selection of models is controlled by the `VELDEP` keyword.

2 Both CN and Forchheimer effects may be including in the generalized pseudo pressure option that calculates a blocking factor due to near well condensate drop-out (see "Special inflow equations" on page 1195). This is controlled by the `PICOND` keyword.

Non-Newtonian fluid flows

Chapter 48

Introduction

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL

Simulation of non-Newtonian fluid flows requires a significant change to the traditional Darcy flow equation. Indeed the rheology of the fluid, which can be described by an apparent or effective viscosity, may be a non-linear function of the flow rate and may as well depend on the rock properties such as the permeability and the porosity. In order to remain as close to the Darcy form as we can, we introduce a multiplier B that modifies the mobility of the fluid depending the cell properties and local flow rate. The solution of the non-linear equation is obtained by local Newton iteration backed up by a secant method.

The multi-phase Darcy equation is therefore modified as follows:

$$Q = AK(k_r/\mu)B\Delta P \qquad \text{[EQ 48.1]}$$

where B is **positive** and may be a complex function involving grid properties, flow rate or pressure drop, and the rheology of the fluid (models are provided below):

$$B = B(d, K, \phi, Q, |\Delta P|, n, \tau, \delta, \beta) \qquad \text{[EQ 48.2]}$$

The rheology (n, τ, δ, β) is set to depend on the polymer concentration (used as a tracer) that allows having a distributed rheology in the reservoir.

As B is dependent on the flow rate, we need to solve the flow between two adjacent cells and the flow at the connections between the wells and the reservoir.

Flow between two cells in the reservoir

The flow between two adjacent cells i and j in the reservoir in a given direction is the solution of the following flow equation:

$$\Delta P_{ij} = \frac{\mu_{ij}}{k_{rij}}\left[\frac{d_i}{AK_i}\frac{Q}{B_i(\ldots, Q, \ldots)} + \frac{d_j}{AK_j}\frac{Q}{B_j(\ldots, Q, \ldots)}\right] \qquad \text{[EQ 48.3]}$$

where k_{rij}/μ_{ij} is the upstream mobility. This equation is solved numerically using a Newtonian algorithm backed up with a secant method. The solution of this equation leads to:

$$Q = \begin{cases} \frac{k_{rij}}{\mu_{ij}} T^* \Delta P_{ij} & \text{if } (B_i > 0) \text{ and } (B_j > 0) \\ 0 & \text{otherwise} \end{cases} \quad \text{[EQ 48.4]}$$

where the new transmissibility T^* is

$$T^* = \frac{1}{\frac{d_i}{AK_i} \frac{Q}{B_i(\ldots, Q, \ldots)} + \frac{d_j}{AK_j} \frac{Q}{B_j(\ldots, Q, \ldots)}} \quad \text{[EQ 48.5]}$$

Flow at the well connection

For the flow between the well and the reservoir grid block, we assume that there is no preferred direction for the non-Newtonian fluid. Under that simplification we keep the connection factor unchanged, so the well flow rate writes:

$$Q = T_w M B \Delta P \quad \text{[EQ 48.6]}$$

where

- T_w is the well connection factor
- M is the mobility
- B is the non-Newtonian modifier, function of the flow rate Q
- ΔP is the pressure drop (draw)

Equation [EQ 48.6] is also solved by local Newtonian iteration backed up with a secant method to find the flow rate and therefore the right modifier value.

Note The assumption that there is no preferred flow path at the connection between the well and the grid block containing the well may become questionable if the degree of anisotropy (ratio between the permeabilities in the plane containing the completion) becomes significant. Because of the shear thinning effect the injectivity can become direction dependent.

Unit conversion

Equations [EQ 48.1] and [EQ 48.6] are only valid when expressed in cgs units. In addition to the unit factor conversion used for Newtonian fluids, we need to introduce an other coefficient to take into account the conversion for the B factor. For other units systems used in ECLIPSE we therefore introduce a coefficient C_u such as:

$$C_u = \frac{B_{cgs}}{B^*} \quad \text{[EQ 48.7]}$$

where B_{cgs} is the non-Newtonian modifier expressed in cgs units and B^* is expressed in other units.

Non-Newtonian model: Herschel-Bulkley fluids

Herschel-Bulkley fluids are described by the following non-Newtonian modifier

$$B = \begin{cases} \dfrac{\left(1 - \dfrac{d\alpha\tau}{|\Delta P|}\right)}{\dfrac{1}{12}\left(9 + \dfrac{3}{n}\right)^n (72\delta K)^{(1-n)/2}} \left(\dfrac{|Q|}{A}\right)^{1-n} & \text{if } \left(\dfrac{d\alpha\tau}{|\Delta P|}\right) < 1 \\ 0 & \text{otherwise} \end{cases}$$

where:

Q is the volumetric flow rate

K is the rock permeability

n is the power index $0 < n \leq 1$

A is the area through which flow occurs

d is the distance between cells

τ is the yield stress

α is a weighing factor for the yield. $\alpha = \sqrt{\dfrac{\phi}{2}}\sqrt{\dfrac{\delta}{K}}$, with the porosity ϕ

δ is the turtuosity

ΔP is the pressure difference between cells

As mentioned earlier the rheological parameters n, τ, δ are function of the polymer concentration to allow spatial distribution of the fluid properties. The unit conversion C_u is now also dependent on n

$$C_u = \left(\frac{Q_{cgs}}{Q^*}\right)^{1-n}\left(\frac{A^*}{A_{cgs}}\right)^{1-n}\left(\frac{K_{cgs}}{K^*}\right)^{(1-n)/2} \qquad \text{[EQ 48.8]}$$

Table 48.1 Units of conversion, Herschel-Bulkley fluids

	Metric	Field	Lab
C_u (HB.)	$\left(\frac{\sqrt{10}}{86.4}\right)^{1-n}$	$\left(\frac{541.16844955}{8640}\right)^{1-n}$	$\left(\frac{\sqrt{10}}{6}\right)^{1-n}$

Usage

This model applies only to aqueous phase (water and polymer). To activate the model the following keywords are required:

Table 48.2 Usage

Keyword	Description
POLYMER	activate the polymer flood option (RUNSPEC)
NNEWTF	activate the Herschel-Bulkley model and provides some table sizes (RUNSPEC)
FHERCHBL	provides the look up tables for the rheological parameters as a function of the polymer concentration.

Optimization

Chapter 49

Introduction

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

This chapter describes the optimization facilities within ECLIPSE.

Several aspects of an engineering study involve the optimization of an ECLIPSE model. Increasingly, these optimization tasks are being incorporated into software programs, often using the simulator as a 'black box' function evaluation in an outer optimization loop. Optimization methods are readily divided into gradient and non-gradient methods. Methods that do not use gradients of the objective function with respect to model parameters, such as simulated annealing, are typically only applied to greatly simplified models, as they require a great many runs of the simulator. However, even methods that do use gradient information can be prohibitively expensive if the gradients are derived numerically, from multiple runs of the 'black box'.

ECLIPSE production profile gradients can be calculated much more efficiently from within the simulator itself (see "The Gradient Option" on page 463). The SimOpt interactive application successfully makes use of these ECLIPSE gradients in an external gradient-based optimization loop, as an aid in the history matching phase of a study.

Optimization capabilities have also been built into ECLIPSE 300, which use gradients of a general objective function rather than the production profiles. The gradients are calculated using the Adjoint technique, which has significant performance advantages when the number of parameters is large. Production constraints are handled using a Lagrangian formulation of the problem, which also provides an enormous speed up compared to treatments external to ECLIPSE.

The sections that follow give a technical description of the various aspects of the optimization workflow environment within ECLIPSE 300. This is followed by some guidelines on the use of the optimization feature, a tabulated list of optimization keywords, and subsequently some examples of their use in the ECLIPSE 300 Reservoir Optimization option.

Non-linear Gradient-Based Optimization in ECLIPSE

Optimization Structure

The central task of optimization is to maximize some objective function, J, by variation of a set of control parameters, $\underline{\lambda}$. An example optimization problem is that of reservoir optimization, where J can be simply the total oil produced and $\underline{\lambda}$ are the controlling well rates during a simulation run.

This optimization task is invariably iterative. For gradient-based optimization, this can be represented schematically as Figure 49.1:

Figure 49.1 Schematic representation of gradient based optimization

This iterative structure is based on obtaining a search direction, $\boldsymbol{s}$, from a set of gradients of the objective function with respect to the control parameters, $dJ/d\underline{\lambda}$. A line search is performed using this initial search direction to find the optimum step size at which point the line search is terminated and a new direction calculated. This loop is repeated until one (or more) termination criteria are met. These termination criteria include lower limits on the relative changes of J or of $\underline{\lambda}$.

This optimization loop structure has been implemented in ECLIPSE 300, and is controlled by keyword definitions given in the `OPTIMIZE` section of the ECLIPSE 300 data file.

Impact of Constraints

The presence of constraints within an optimization greatly increases the complexity of the problem [Ref. 96]. Constraints may vary from simple bounds on parameter values to full non-linear functional inequality relationships between parameters and ECLIPSE outputs, for example, the well and group production constraints.

The former are dealt with relatively easily, by a restriction of the step during a line search. However, the latter require modifications to the search direction.

Inequality constraints are most frequently found in the form $f(x, y) \leq K$, which may be re-written as:

$$f(x, y) - K = V \leq 0 \qquad \text{[EQ 49.1]}$$

A constraint of this form is said to become "active" when the inequality becomes an equality, that is when $V = 0$. Otherwise the constraint is said to be "inactive".

A simple example that illustrates the impact of constraints is shown in Figure 49.2:

Figure 49.2 An Example of constrained optimization.

This diagram shows the objective function contour map for a problem with two variables, λ_1 and λ_2, in which a nonlinear inequality constraint function $f(\lambda_1, \lambda_2) \leq K$ divides the parameter space into feasible (bottom left) and infeasible (top right) regions. From a starting position at **A** using a search direction indicated by the dashed line, a line search would have to terminate at **B** (to avoid violating the constraint). From **B**, an unconstrained search direction would point towards **C**, which violates the constraint. The direction we require is towards the constrained maximum at **D**.

The Lagrangian formulation is a well established mechanism for combining an objective function with equality constraints. For the example above, we form the following Lagrangian function: [Ref. 96]

$$L(\underline{\lambda}) = J(\underline{\lambda}) - \Phi V(\underline{\lambda}) \qquad \text{[EQ 49.2]}$$

For suitable values of the Lagrange multiplier, Φ, unconstrained optimization of the Lagrangian simultaneously achieves constrained optimization of the original objective function.

The Lagrangian formulation relies on being able to determine which inequality constraints are active, that is which are currently an equality constraint. As the above illustration indicated, a specialized line search algorithm can be used for just such a purpose. However, one potential problem of this approach is that it may dominate the optimization convergence - at worst we may end up following the Simplex algorithm of linear programming, with one iteration per constraint.

"Adjoint Gradient Calculations in ECLIPSE" on page 731 shows how we use the Lagrangian formulation to construct the optimization gradients using the Adjoint method.

Adjoint Gradient Calculations in ECLIPSE

Basic formulation

After a simulation run, the optimization workflow starts with the calculation of the gradients of the objective function with respect to the chosen parameters. These gradients must take into account the equality constraints imposed by the fluid flow calculations solved in ECLIPSE.

At each simulator time-step, ECLIPSE forces the fully implicit residual in each simulator grid block to be zero (see"Formulation of the Equations" on page 341).

Caution The adjoint gradient calculations require a fully implicit formulation of the equations. If you try to use the Reservoir Optimization feature without this type of formulation, the run will be aborted.

This cell residual constraint equation may be written as

$$\boldsymbol{R}(\boldsymbol{X}^{n+1},\boldsymbol{X}^{n}) = \frac{\boldsymbol{M}(\boldsymbol{X}^{n+1})-\boldsymbol{M}(\boldsymbol{X}^{n})}{dt} + \boldsymbol{F}(\boldsymbol{X}^{n+1}) + \boldsymbol{Q}(\boldsymbol{X}^{n+1}) = \boldsymbol{0} \qquad \text{[EQ 49.3]}$$

where the superscripts $n+1$ and n denote the current and previous time-step respectively, and

$\boldsymbol{X}^{n}$ are the solution variables (pressure, molar densities, etc.) at time n

$\boldsymbol{M}(\boldsymbol{X}^{n})$ is the fluid-in-place in each simulator grid block at time n,

$\boldsymbol{F}(\boldsymbol{X}^{n})$ is the net flow rate into neighboring grid blocks at time n, and

$\boldsymbol{Q}(\boldsymbol{X}^{n})$ is the net flow rate into wells at time n.

Note that the objective function is a function of both the control parameters $\underline{\lambda}$ and the simulation primary variables $\boldsymbol{X}$; i.e. $J = J(\boldsymbol{X}, \underline{\lambda})$.

A Lagrangian formulation of the objective function is used in order to comply with the constraints imposed by the cell residuals,

$$L(\boldsymbol{X}, \underline{\lambda}) = J(\boldsymbol{X}, \underline{\lambda}) - \underline{\Phi} \cdot \boldsymbol{R}(\boldsymbol{X}, \underline{\lambda}) \qquad \text{[EQ 49.4]}$$

Note that when all the residuals are satisfied ($\boldsymbol{R} = \boldsymbol{0}$) the Lagrangian is simply the objective function. The vector of Lagrange multipliers on the residuals, $\underline{\Phi}$, is known as the "adjoint vector".

We are interested in forming the total parameter derivatives of this Lagrangian:

$$\frac{dL}{d\underline{\lambda}} = \frac{\partial L}{\partial \underline{\lambda}} + \frac{\partial L}{\partial \boldsymbol{X}} \cdot \frac{\partial \boldsymbol{X}}{\partial \underline{\lambda}} \qquad \text{[EQ 49.5]}$$

for which we require the partial Lagrangian derivatives:

$$\frac{\partial L}{\partial \boldsymbol{X}} = \frac{\partial J}{\partial \boldsymbol{X}} - \left(\underline{\Phi} \cdot \frac{\partial \boldsymbol{R}}{\partial \boldsymbol{X}}\right) \qquad \text{[EQ 49.6]}$$

and

$$\frac{\partial L}{\partial \underline{\lambda}} = \frac{\partial J}{\partial \underline{\lambda}} - \left(\underline{\Phi} \cdot \frac{\partial \boldsymbol{R}}{\partial \underline{\lambda}}\right) \qquad \text{[EQ 49.7]}$$

as well as the solution derivatives $\frac{\partial \boldsymbol{X}}{\partial \underline{\lambda}}$.

Since $\boldsymbol{R} = \boldsymbol{0}$, we are free to choose our vector of Lagrange multipliers in any way we choose. As there is an implicit dependence of the solution variables on the control parameters, $\boldsymbol{X} = \boldsymbol{X}(\underline{\lambda})$, the natural choice of $\underline{\Phi}$ is that which sets [EQ 49.6] to zero. This removes the need to calculate the solution derivatives. Hence we solve the adjoint set of linear equations:

$$\underline{\Phi} \cdot \frac{\partial \boldsymbol{R}}{\partial \boldsymbol{X}} = \frac{\partial J}{\partial \boldsymbol{X}} \qquad \text{[EQ 49.8]}$$

for the adjoint vector $\underline{\Phi}$. The dimension of the adjoint vector for each time period is that of the normal fluid flow Jacobian (number of grid cells times number of simulation variables). As an example, scoping [EQ 49.8] into four distinct time periods, it may be written as:

$$\left[\underline{\Phi}^1 \ \underline{\Phi}^2 \ \underline{\Phi}^3 \ \underline{\Phi}^4\right] \begin{bmatrix} \boldsymbol{A}^{11} & \boldsymbol{0} & \boldsymbol{0} & \boldsymbol{0} \\ \boldsymbol{A}^{21} & \boldsymbol{A}^{22} & \boldsymbol{0} & \boldsymbol{0} \\ \boldsymbol{0} & \boldsymbol{A}^{32} & \boldsymbol{A}^{33} & \boldsymbol{0} \\ \boldsymbol{0} & \boldsymbol{0} & \boldsymbol{A}^{43} & \boldsymbol{A}^{44} \end{bmatrix} = \left[\frac{\partial J}{\partial \boldsymbol{X}^1} \ \frac{\partial J}{\partial \boldsymbol{X}^2} \ \frac{\partial J}{\partial \boldsymbol{X}^3} \ \frac{\partial J}{\partial \boldsymbol{X}^4}\right] \qquad \text{[EQ 49.9]}$$

where the $\boldsymbol{A}$ matrices are derivatives of the fluid flow residuals with respect to solution variables, given by

$$\boldsymbol{A}^{mn} = \frac{\partial \boldsymbol{R}^m}{\partial \boldsymbol{X}^n} \qquad \text{[EQ 49.10]}$$

where m and n are time indices. By multiplying out [EQ 49.9] term by term the general equation, which must be solved at any given time, index n is

$$\underline{\Phi}^n \boldsymbol{A}^{nn} = \frac{\partial J}{\partial \boldsymbol{X}^n} - \underline{\Phi}^{n+1} \boldsymbol{A}^{n+1,n} \qquad \text{[EQ 49.11]}$$

This involves the value of the adjoint vector at time period, $n+1$. Since the only independent equation is the last one (for which we can solve for $\underline{\Phi}^4$) then the adjoint equation [EQ 49.11] must be solved backwards in time.

With the adjoint vector, $\underline{\Phi}$, chosen as the solution of this system, the Lagrangian gradients we require are:

$$\frac{dL}{d\underline{\lambda}} = \frac{\partial J}{\partial \underline{\lambda}} - \left(\underline{\Phi} \cdot \frac{\partial \boldsymbol{R}}{\partial \underline{\lambda}}\right) \qquad \text{[EQ 49.12]}$$

Note The effort of calculating these gradients is dominated by the solution for the adjoint vector, $\underline{\Phi}$, which is independent of the type or number of parameters. This makes the adjoint method very efficient for large number of parameters compared to other gradient calculation techniques.

Introducing Additional Constraints

In the reservoir simulator the fluid flow residual equations ($\boldsymbol{R} = 0$) are constraints, which must always be observed as the objective function is optimized. Optionally, you may impose additional constraints on the system, for example limiting the production of a certain phase from a group of wells.

The Lagrangian approach allows us to readily incorporate these additional equality constraint equations, $\boldsymbol{V} = \boldsymbol{0}$, by the introduction of further Lagrange multipliers, $\underline{\Psi}$:

$$L(\boldsymbol{X}, \underline{\lambda}) = J(\boldsymbol{X}, \underline{\lambda}) - \underline{\Phi} \cdot \boldsymbol{R}(\boldsymbol{X}, \underline{\lambda}) - \underline{\Psi} \cdot \boldsymbol{V}(\boldsymbol{X}, \underline{\lambda}) \qquad \text{[EQ 49.13]}$$

The Lagrangian derivatives given in equations[EQ 49.5] to [EQ 49.7] above may be extended to include derivatives of the additional constraint functions. This allows us to form an extended adjoint system of equations, from which we obtain both $\underline{\Phi}$ and $\underline{\Psi}$. The Lagrangian gradient equations, [EQ 49.6] and [EQ 49.7], then become:

$$\frac{\partial L}{\partial \boldsymbol{X}} = \frac{\partial J}{\partial \boldsymbol{X}} - \left(\underline{\Phi} \cdot \frac{\partial \boldsymbol{R}}{\partial \boldsymbol{X}}\right) - \left(\underline{\Psi} \cdot \frac{\partial \boldsymbol{V}}{\partial \boldsymbol{X}}\right) \qquad \text{[EQ 49.14]}$$

$$\frac{\partial L}{\partial \underline{\lambda}} = \frac{\partial J}{\partial \underline{\lambda}} - \left(\underline{\Phi} \cdot \frac{\partial \boldsymbol{R}}{\partial \underline{\lambda}}\right) - \left(\underline{\Psi} \cdot \frac{\partial \boldsymbol{V}}{\partial \underline{\lambda}}\right) \qquad \text{[EQ 49.15]}$$

Notice that we have retained the special identity of the cell residual equations, as distinct from all other constraints. This allows us to solve for the Lagrange multipliers using modifications to the existing simulation fluid flow matrix $\boldsymbol{A}$, rather than having to construct an augmented system.

In order to solve for $\underline{\Psi}$, we arbitrarily set $\frac{\partial L}{\partial \lambda_i} = 0$ in equation [EQ 49.15] for as many parameters, index i, as there are active constraints. Consequently, equation [EQ 49.15] only applies to those parameters not used in determining $\underline{\Psi}$. This effectively divides the parameter set into ‘free’ and ‘dependent’ subsets: $\underline{\lambda} = (\underline{\lambda}_f \underline{\lambda}_d)$, where the free parameters are those for which we have Lagrangian gradients, and the dependent parameters are those which ‘depend’ on the free parameters using the constraint equations. This has important implications for the optimization workflow, as discussed in "Optimizer Search Direction" on page 734.

Optimizer Search Direction

Given the Lagrangian gradients, $dL/d\underline{\lambda}$, the optimization workflow then proceeds by calculating a search direction, $\underline{s}$.

However, in the presence of equality constraints other than the cell residual equations, we do not have Lagrangian gradients for dependent parameters, $\underline{\lambda}_d$. The usual course of action is to determine the 'free' search direction, $\underline{s}_f$, update the free parameter values, $\underline{\lambda}_f$, and then invert the constraint equations to update $\underline{\lambda}_d$. Unfortunately, since the constraint equations are most generally non-linear, this inversion is iterative, and expensive, since each function evaluation requires a simulation run.

Instead, changes in the dependent parameters are determined from changes in the free parameters using a linearization of the perturbed constraint equations:

$$\frac{\partial \boldsymbol{R}}{\partial \boldsymbol{X}} \cdot \Delta \boldsymbol{X} + \frac{\partial \boldsymbol{R}}{\partial \underline{\lambda}_d} \cdot \Delta \underline{\lambda}_d + \frac{\partial \boldsymbol{R}}{\partial \underline{\lambda}_f} \cdot \Delta \underline{\lambda}_f = 0 \qquad \text{[EQ 49.16]}$$

$$\frac{\partial \boldsymbol{V}}{\partial \boldsymbol{X}} \cdot \Delta \boldsymbol{X} + \frac{\partial \boldsymbol{V}}{\partial \underline{\lambda}_d} \cdot \Delta \underline{\lambda}_d + \frac{\partial \boldsymbol{V}}{\partial \underline{\lambda}_f} \cdot \Delta \underline{\lambda}_f = 0 \qquad \text{[EQ 49.17]}$$

With $\Delta \underline{\lambda}_f = \boldsymbol{s}_f$, we solve this system for $\boldsymbol{s}_d = \Delta \underline{\lambda}_d$, thus obtaining the full search direction. In solving the above, our boundary conditions are that $\Delta \boldsymbol{X} = 0$ at time zero, and so these equations must be solved forwards in time.

The modified workflow for determining the optimization search direction now consists of two distinct parts.

- First the search direction for the free parameters is determined in the backwards solve.
- Then the free parameter search direction is used to determine the search direction for the set of dependent parameters in a forward solve of the system.

Note In this release, the calculation of the search direction from the gradients is performed using either a steepest descent or a conjugate gradient algorithm. The conjugate gradient algorithm requires slightly more computational work and storage. Due to the presence of the constraints, our experience is that there is not much difference in the convergence rate of the two methods, although the conjugate gradient algorithm does appear to give a more stable convergence path.

Line Search

Given a search direction, $\underline{s}$, the standard line search algorithm is to determine some step size multiplier, α, which maximizes the objective function value. The control parameters are updated using the algorithm $\underline{\lambda} = \underline{\lambda} + \alpha \underline{s}$. Figure 49.3 illustrates the problem.

Figure 49.3 Line search

With no preferred scale, a line search may take many iterations to find the optimum position. However, along the way, we may encounter, or jump over, a new constraint. An indication of the position of the nearest constraint to the starting position of a line search can be obtained by expanding the constraint inequalities to give:

$$V_i(\boldsymbol{X} + \alpha_i \Delta \boldsymbol{X} \ , \ \underline{\lambda}_f + \alpha_i \Delta \underline{\lambda}_f) = V_i(\boldsymbol{X}, \underline{\lambda}) + \alpha_i \left(\frac{\partial V_i}{\partial \boldsymbol{X}} \cdot \Delta \boldsymbol{X} \right) + \alpha_i \left(\frac{\partial V_i}{\partial \underline{\lambda}_f} \cdot \Delta \underline{\lambda}_f \right) \leq 0 \qquad \text{[EQ 49.18]}$$

Clearly, the closest constraint (that which would be first violated if an unrestricted amount of the search direction were used) is that for which the value of α is smallest when the inequality becomes an equality constraint. This may be obtained by rearranging [EQ 49.18], from which:

$$\alpha_i = -V_i \Big/ \left(\frac{\partial V_i}{\partial \underline{\lambda}_f} \cdot \Delta \underline{\lambda}_f + \frac{\partial V_i}{\partial \boldsymbol{X}} \cdot \Delta \boldsymbol{X} \right) \qquad \text{[EQ 49.19]}$$

This sets a useful guide for the line search algorithm in the presence of constraints, and in practice can save many iterations of the line search.

As a further aid to convergence of the overall optimization loop, the tolerance used in determining the proximity of a constraint, ε, can be specified to potentially ‘catch’ several new constraints at once.

Reservoir Optimization in ECLIPSE

The ECLIPSE 300 Reservoir Optimization option allows you to optimize production by variation of well-based parameters. The option is triggered by the RESOPT keyword in the RUNSPEC section. This activates the processing of the OPTIMIZE section of a data set, in which you can specify particular forms of the objective function, parameters and constraints appropriate to the reservoir optimization task. These are described below.

The Objective Function

The reservoir optimization objective function is specified using the OPTFUNC keyword. This keyword permits specification of a very general function of the form:

$$J = \sum_i C_i \sum_{t_1(i)}^{t_2(i)} d_i(t) f_i(t) \Delta t \qquad \text{[EQ 49.20]}$$

where $f_i(t)$ denotes a produced or injected fluid rate (for example, field oil production rate), t_1 and t_2 are restart indexes defining the time range of interest, C_i is a function-dependent weighting coefficient (whereby for example water production could be penalized), and d_i is a discount factor, given by:

$$d_i(t) = (1 + r_i)^{-N(t)} \qquad \text{[EQ 49.21]}$$

for annual discount rate r_i and simulation time $N(t)$ in fractional years. The time limits appearing on the second summation allow functions to be included in the total objective function at different times with different weights, and different discount factors.

Control Parameters

Parameters are specified using the OPTPARS keyword. In this release, the available parameters are:

- Well bottom hole pressures
- Well oil and gas production target rates
- Well oil, water and gas injection target rates
- Multi-segmented well segment cross sectional area

As an example, consider the bottom hole pressure at which a well is operated as a parameter. In general the value of this parameter varies over the course of a simulation, resulting in many control parameters for this well, one for each period of time. A general parameter is therefore denoted λ_p^t, where p identifies a specific control parameter (for example, bottom hole pressure for a particular well), and t scopes the parameter in time.

Parameters are created using the time axis defined by restart files. Thus, to vary well rates quarterly, restart files must be requested at three month intervals in the data set. For m user specified parameters, and with the number of restart files defined as n, the complete vector of control parameters for the problem is thus:

$$\underline{\lambda} = \left[\left[\lambda^1{}_1\ \lambda^2{}_1\ \ldots\ \lambda^n{}_1\right]\ \left[\lambda^1{}_2\ \lambda^2{}_2\ \ldots\ \lambda^n{}_2\right]\ \cdots\ \left[\lambda^1{}_m\ \lambda^2{}_m\ \ldots\ \lambda^n{}_m\right]\right] \qquad \text{[EQ 49.22]}$$

This vector is reported in the PRINT file during an optimization run.

Note In reservoir optimization, the total number of control parameters can be very high. For example, in a large study with perhaps 1000 wells under optimizer control, and with settings to be changed four times a year over a twenty-five year period, there are a total number of 10^5 control parameters. Thus an efficient gradient calculation scheme, such as the adjoint method, is critically important.

Production Constraints

Production inequality constraints are specified using the OPTCONS keyword. The Reservoir Optimization feature allows you to specify constraints in the general form:

$$f(\boldsymbol{X}, \underline{\lambda}) \leq K \qquad \text{[EQ 49.23]}$$

where $\boldsymbol{X}$ are the reservoir solution variables, $\underline{\lambda}$ is the vector of control parameters, K is a constant, and f is a fluid production or injection rate, from either a group or a well. You may also specify the time period for which the constraint is to apply, defaulted to the entire simulation period.

File Output

The Reservoir Optimization option writes output to the ECLIPSE PRINT file. For each iteration of the optimization procedure, the following are reported:

- Objective function value
- Parameter values
- Constraint values
- A list of constraints which are currently active
- Optimizer search direction
- Line search details.

As an option the results of each optimization iteration may be output to separate summary and restart files for visualization by ECLIPSE Office. This is achieved by using the RPTOPT keyword mnemonic SEPARATE.

Optimizer Performance

Parameter Selection

It is strongly recommended that, from the available selection of optimization parameters, you try the bottom hole pressure parameters first.

The reason for this is that production rates are insensitive to well target rates whenever a well falls onto bottom hole pressure control. With bottom hole pressures as parameters, the production rates are only insensitive when a well is shut, which happens less frequently, and thus optimizations performed with such parameters are usually more successful.

Adjoint Gradient Accuracy

In general the most accurate gradients are obtained when the adjoint functions are integrated over small time steps. The adjoint solution is generated using restart files, so it is best to request frequent restarts.

In principle, the obvious time steps to use are those used in the forward simulation. Unfortunately, the variation of time steps between optimizer iterations makes this unsuitable for our purposes, unless the parameters are defined on a separate time axis. Thus, for example, the adjoint functions could be integrated at the time step level, whilst the parameters are defined at some coarser scale. The presence of the constraint functions precludes this as a practical technique; since with fewer parameters than constraint functions, the adjoint system is under determined, making gradient calculations impossible.

Parameter Bounds

It is quite important to set realistic parameter bounds for the optimization, using the parameter definition under the `OPTPARS` keyword. This prevents unphysical (and unacceptable) values from occurring, and also prevents the simulator from going into tricky convergence regions.

Termination Criteria

It is important to set the optimizer tolerances appropriately using the `OPTTUNE` keyword. These have been set with broadly reasonable defaults but, if they are changed, then care should be taken not to set them too tight - for example, by allowing termination only when the objective function is changing by too small a fraction.

Guidelines on using the reservoir optimization feature

This section gives you some guidelines on how to use the reservoir optimization feature. It is divided into the following sections:

- "Screening your model for suitability" on page 739
- "Deciding the optimization problem you want to solve" on page 740
- "Modifying your model for reservoir optimization" on page 740
- "Checking your modified model" on page 741
- "Tracking the progress and results of the optimizer" on page 742

Screening your model for suitability

Note The starting point for the use of the Reservoir Optimization feature is that you should have an ECLIPSE 300 model that runs without any problems.

The first thing you should do is check your model for its compatibility with the requirements of the Reservoir Optimization feature.

Features that the model must include

1. It should be fully implicit.
2. It should have reasonably frequent report step output; monthly report output is a good rule-of-thumb.
   - The adjoint method in the Reservoir Optimization feature uses report steps, as opposed to time steps, to calculate the required gradients, so the gradient calculation will lose accuracy if the report steps are too far apart.

Features that the model should not include

1. The use of the parallel option (`PARALLEL` keyword).
2. The use of fast restarts (`LOAD` keyword).
3. The use of local grid refinements (`CARFIN`, `EXTFIN` or `RADFIN` keywords).
4. The use of grid coarsening (`COARSEN` keyword).
5. The use of the sector modelling option (`SECTOR` keyword).
6. The use of the thermal option (`THERMAL` keyword).
7. AIM, IMPES or IMPSAT formulations.
8. Any well controls which will compete against the optimizer, for example `ACTION` keywords, `WECON`.

If necessary (and possible) you need to decide if it is possible for you to modify your model to meet these requirements. If you change your model, make sure that it can still be run successfully in ECLIPSE 300 and that these changes do not affect the results of the simulation dramatically.

If your model is large, then it may be useful to construct a coarse version so that you can determine if the Reservoir Optimization feature gives useful results, before attempting the use the feature on the detailed model.

Deciding the optimization problem you want to solve

You need to decide what objective you are trying to achieve, for example, maximizing discounted oil production total while minimizing water production total.

You also need to decide what constraints (if any) you need to impose while trying to achieve this objective, for example, the water cut must never rise above 10% in certain wells and groups.

Finally you must decide which well control parameters you want to use to try to achieve your constrained objective.

1. Well target production/injection rates are only good for optimization purposes as long as the well stays on target production. Once the well falls off target rate control there will be no response to changing the target. In this situation, WBHP is a good control parameter as there will always be a response while the well is open.
2. For smart well optimization, the use of valve parameters controlling the cross-sectional area of a valve segment (`WSEGVALV`) is encouraged.

Now that you have decided your optimization problem, there are a couple of issues for you to consider related to the time it takes for your model to run.

3. Typically the calculation of adjoint gradients (sensitivities) will take a time which is less than, but of the order of, the time of the normal simulation.
4. Each simulation performed with the control parameters obtained by a model update from the adjoint gradients will take a similar time, and as this is a gradient-based optimization, the use of a line-search algorithm means that there may be several simulations before an optimum is achieved using a given search direction.

The use of constraints and also simple bounds on parameters help to constrain the line-search step size and reduce the number of simulation runs required. Clearly however the overall time to run the optimizer can become a concern. You can control the overall run time as follows:

5. Control on the maximum number of optimization iterations (gradient evaluations) and simulations is provided using the `OPTDIMS` keyword.
6. If the optimization problem (that is the objective function, parameters and constraints) is restricted to a particular sector of the model, then you should consider using a model of just that sector to reduce the time required for each simulation.
7. If your model has a long history period, and the optimization problem (that is the objective function, parameters and constraints) are only specified in the prediction part of the schedule, then you should consider using a restart file to reduce the time required for each simulation.

Modifying your model for reservoir optimization

To specify your Reservoir Optimization problem in the model, you need to:

1. Add the `RESOPT` keyword to the `RUNSPEC` section to request the Reservoir Optimization
2. Add the `OPTIMIZE` section keyword before your `END` keyword
3. Add the `OPTFUNC` keyword to the `OPTIMIZE` section to specify your objective function

4 Add the OPTCONS keyword to the OPTIMIZE section to specify your production/injection constraints.

- You should ensure that none of these constraints are violated at the start of the optimization because the optimizer must start from a feasible position honoring all the constraints.

  ECLIPSE 300 will check for this when processing the OPTCONS keyword, but it is a good idea to look at the values of constraint quantities in the forward simulation before adding the OPTCONS keyword.

  For example, if the maximum FWPR in the forward simulation is 2000sm3/day, then setting an optimization constraint of 1000sm3/day is invalid.The starting point for simulation will violate constraints and there is no mechanism to return to a feasible parameter space honoring the constraint.

5 Add the OPTPARS keyword to the OPTIMIZE section to specify your control parameters.

- You should try to make sure that your initial parameter values lie within the upper and lower limits set by this keyword.

  There is an initial feasibility check in ECLIPSE 300 to ensure that the parameter values in the forward simulation are within these limits, though it is a good idea to look at the range of parameter values in the forward simulation before specifying the limits.

6 Add the OPTDIMS keyword to the OPTIMIZE section to specify how many optimization iterations and how many simulations are allowed in the Reservoir Optimization feature.

Hint Initially these limits should both be set to one so that you can perform a simple sensitivity analysis for parameter screening.

---

7 Add the RPTOPT keyword to the OPTIMIZE section and request SEPARATE output.

- This uses more hard-disk space than non-separate output but does allow tracking of the progress of the parameters, constraints and objective function components fully. A separate summary file will be created for each optimization iteration, and these production profiles may be visualized independently in ECLIPSE Office.

8 As you would in a normal simulation, add summary and reporting output requests for any simulation results you want to track.

- If a production constraint has been added in the OPTIMIZE section it is good practice to add the mnemonic for this constraint to the SUMMARY section so that you can track whether the constraint is violated in each simulation.

Note The method used for calculating the adjoint gradients relies on restart files (containing the well and grid block solutions) being generated at each report time during the simulation. Any RPTRST keyword mnemonics that turn off the required reporting will be ignored.

Checking your modified model

Now that you have specified the Reservoir Optimization problem in the model, and you have indicated that you just want one gradient calculation performed and no optimization (using the OPTDIMS keyword), you should run the model in ECLIPSE 300 and examine the content of the PRT file.

1 Look for general errors and warnings, in ECLIPSE 300 these always start with the strings "`@--Error`" and "`@--Warning`". There should be no errors introduced if the steps above have all been followed.

- This should tell you if any constraints are violated at your starting point. If this is the case, you will get the message:

```
@--Error   at   150.0000 Days  4 May 1988
@ Optimization initial point is infeasible
```

The optimizer must stay within the feasible parameter space at all times (except when backtracking from a constraint). If you are violating any constraints, then you either have to change your starting point to make it feasible or you have to change your constraint.

- This will also tell you if you initial parameter values are outside their lower and upper bounds. If this is the case, you will get a message similar to:

```
@--Error   at   150.0000 Days  4 May 1988
@ Optimization Parameter     5 is out of range
@ Parameter Name  : WBHP
@ Parameter Domain: P1
@ Restart Index   :    5
@ Current parameter value is   250.000
@ Lower limit =0.100000E+03, Upper Limit =0.200000E+03
@ Change parameter range using the OPTPARS keyword
```

If you have initial parameter values that are outside their lower and upper bounds, then you must either change your initial parameter values, or you must change your parameter bounds.

2 Look at the sensitivity of your objective to the parameter values. This is indicated by the magnitude of the search direction reported for the parameters. These are reported in a table immediately after the text:

```
=Optimization Parameter Update=====================================
```

- You need to decide whether there is a reasonable response of the objective function to the parameters you specified. (The definition of a reasonable response is largely problem dependent, the expected parameter changes should not all be too close to zero.) If there is not a reasonable response, then you should re-consider your parameter choice as the optimizer will not be able to make much progress with ineffective parameters.

If you need to change the parameters or constraints for any of the reasons given in this section, then you will need to run the Reservoir Optimization again to check the new optimization problem you have specified.

Tracking the progress and results of the optimizer

If you have a feasible starting point, and you are happy with your choice of parameters, then you should change the limits on the number of iterations and simulations given in the `OPTDIMS` keyword. The new values you choose will be largely dependent on the time your model takes to simulate.

Hint A reasonable choice for the first optimization attempt is a limit of ten on each number.

Then you run the simulation model with the Reservoir Optimization feature. Essentially, this performs a sequence of simulations for you, calculating new gradient directions and performing line-searches when deemed necessary by the optimizer.

You can check the progress of the optimizer in the PRT file. At the end of each simulation, the following information in output:

```
=====================================================
Optimization Iteration:   1 Simulation No:  1
Current    Objective Function value:     2702118.0
Prev Valid Objective Function value:     1986265.8
Best Valid Objective Function value:     2702118.0  (at Itn: 1 Sim:
 1)
Tolerance in Objective Function    :     0.1000E-05
=====================================================
```

This output tells you how many simulations and gradient calculations have been performed so far. It also tells you the current and previous objective function values, the best valid objective function (that is, the largest value for which no constraints have been violated) and the iteration and simulation at which it occurred.

During line-searches, it is possible for the optimizer to step too far in the search direction and hence violate a constraint. If this happens, you will see text such as:

```
Constraints have been overshot using Alpha =  512.00000
Backtracking Alpha to honour constraints.
Proceeding with new Line Search Alpha value =  259.17128
```

The status of any active constraints are reported in a table under the text:

```
|==Optimization: Constraints=========================================|
```

This table lists the constraints that we are sufficiently close to so that we can consider we are actually on the constraint (that is, have an activity ratio with an absolute value of less than unity).

Active Constraint		Report Index	Current Value	Limiting Value	Activity Ratio
FWPR	FIELD	14	0.1443E+03	0.1500E+03	-0.763
FWPR	FIELD	15	0.1501E+03	0.1500E+03	+0.019
FWPR	FIELD	16	0.1555E+03	0.1500E+03	+0.737

At each simulation, the parameter values (and the update in these values from the previous simulation) are output in a table after the text:

```
=Optimization Parameter Update=====================================
```

At the end of the optimization, the parameter values that were used in the simulation that gave the best objective function and did not violate the constraints are given in a table after the text:

```
=Optimization Parameters For Best Valid Iteration=====
```

The reason that the optimizer terminated is reported at the end of the PRT file:

```
==================================================
Optimization Terminating:
```

This will be one of the following:

1 Objective function has converged

- This means that the relative change in the objective function over the last two simulations is less than the tolerance specified in the OPTTUNE keyword

2 Objective function has zero gradient

- This means that the average gradient of the objective function with respect to the control parameters is less than the tolerance specified in the OPTTUNE keyword

3 Control parameters have converged

- This means that the average relative parameter change over the control parameters is less than the tolerance specified in the OPTTUNE keyword

4 No control parameters are active

5 Maximum number of optimization iterations reached

- This means that we have reached the limit on the number of optimization iterations specified in the OPTDIMS keyword

6 Maximum number of optimization simulation reached

- This means that we have reached the limit on the number of optimization iterations specified in the OPTDIMS keyword

If you ran the optimizer using the SEPARATE mnemonic in the RPTOPT keyword, then the Reservoir Optimization produces separate results files for each iteration in the optimization. These results files have names of the form BASE_01.xxx, BASE_02.xxx, etc., where BASE.DATA is the base data-set, and _01,_02 etc. label the optimization iteration. You can track the progress of the objective function, constraints and parameters at each optimization iteration by plotting their equivalent summary vectors (if you have requested these) in Results Viewer in ECLIPSE Office. If you intend to view the results in this way, then you should also specify the CLRFILES mnemonic in the RPTOPT keyword so that the results files from any previous optimizations are removed before starting this optimization, otherwise the presence of these old results files can cause confusion.

[Ref. 117] gives some examples of reservoir optimization studies.

Summary of keywords (ECLIPSE 300 only)

RUNSPEC section

The keywords are:

Table 49.1 RUNSPEC section

Keyword	Description
RESOPT	Activate the Reservoir Optimization License.

SCHEDULE section

The keywords are:

Table 49.2 SCHEDULE section

Keyword	Description
RPTRST	Set the restart output frequency using BASIC and FREQ.

OPTIMIZE section

The keywords are:

Table 49.3 OPTIMIZE section

Keyword	Description
OPTCONS	Specify the optimization constraint functions
OPTDIMS	Specify the key optimization dimensions
OPTFUNC	Specify the optimization objective function
OPTLOAD	Hot-start a reservoir optimization simulation
OPTOPTS	Specify the optimization options (for example which optimizer to use)
OPTPARS	Specify the optimization control parameters
OPTTUNE	Define tuning for the optimization (for example, tolerances)
RPTOPT	Request the style of optimization output

Miscellaneous

The keywords are:

Table 49.4 Miscellaneous keywords

Keyword	Description
OPTIONS3	Item 77 - compute and output constraint function gradients

Example problem (ECLIPSE 300 only)

The following is the OPTIMIZE section of a Reservoir Optimization example data set.

The Reservoir Optimization license has been activated by the presence of the RESOPT keyword in the RUNSPEC section. Restart files have been requested every calendar month.

```
------------------------
OPTIMIZE
------------------------

OPTDIMS
-- Maximum no.    Maximum no.
-- Iterations      Simulations
     30             100 /

OPTTUNE
-- Constraint Tolerance
 0.05 /

OPTFUNC
--   O.F.        O.F.
-- Component    Domain
   'FOPT'      'FIELD' /
/

OPTPARS
-- Parameter   Parameter   Lower  Upper
-- Mnemonic     Domain     Limit  Limit
   'WBHP'       'P1'       100.0 350.0 /
   'WBHP'       'P2'       100.0 350.0 /
/

OPTCONS
-- Constraint   Constraint   Constraint  Constraining
--   Mnemonic     Domain         Type         Value
     'FOPR'      'FIELD'        '<'         3015 /
     'FGPR'      'FIELD'        '<'       665000 /
/
```

In this example:

1 The objective is to maximize the field oil production total (specified using the OPTFUNC keyword).

2 This is subject to limits of 3015 SM3/DAY on the field oil production rate and 665000 SM3/DAY on the field gas production rate (specified using the OPTCONS keyword).

These constraints are considered to be met if the simulated value lies within 1 part in 20 of the limiting value (specified using the OPTTUNE keyword).

3 The optimizer tries to achieve this objective by varying the bottom hole pressure in wells P1 and P2 between 100 and 300 BARSA at each reporting interval over the whole time period of the simulation (specified using the OPTPARS keyword).

4 The optimizer is allowed a maximum of 30 gradient simulations or 100 forward simulations (whichever happens first) to try to achieve this objective (specified using the `OPTDIMS` keyword).

Parallel option

Chapter 50

Introduction

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL

The Parallel option allows the simulation of a single data set to be distributed across a number of processors. This allows large simulations to be carried out in a shorter time than would normally be the case with the standard simulators. Both ECLIPSE 100 and ECLIPSE 300 are optimized to provide the shortest time to solution, and consequently tailor the number of domains and hence the linear solver to the number of available processors. This has a number of consequences. Firstly, the results obtained on different numbers of processors will agree to engineering accuracy, not to machine accuracy. Secondly, the scalability is poorer than one might expect, as the linear solver becomes less efficient as we go to larger numbers of processors.

You can change the number of processors at restarts depending on the resources available, using flexible restarts.

The Parallel option is designed to be easy to use; you only need one additional keyword to run a data set in parallel mode. You may then use additional keywords to tune the data set for optimal performance.

Activating a parallel run

A parallel run of ECLIPSE 100 or ECLIPSE 300 is requested by the PARALLEL keyword in the RUNSPEC section. Suppose the following is used:

```
PARALLEL
16 /
```

When the ECLIPSE run is submitted, 16 copies of the executable will be spawned. Each of the 16 tasks is then running a copy of the simulator, dedicated to its section of the reservoir. A message passing system is used to pass boundary flow and well information between the tasks and keep them synchronized.

One of the tasks performs file input and output; it is in other respects the same as the remaining tasks, dealing with its own section of the reservoir.

In most respects, running ECLIPSE 100 and ECLIPSE 300 in parallel is the same as running in the standard serial mode. However, there will always be some features or combinations of features which are not available with the parallel option.

The Parallel option is intended for use on a “parallel” computer, such as an IBM SP, SGI origin 2000 or SUN E6000, which have dedicated communications hardware. Parallel runs must have dedicated resources, which means that the number of ECLIPSE processes (the number of domains + the number of serial runs) on any machine should not exceed the number of CPUs on that machine. If the machine is overloaded the performance degradation may be severe as most message passing systems assume a dedicated resource, and processes that are waiting for messages from other processes will often consume CPU time at the expense of those that are trying to do calculations.

Partitioning

Introduction

Parallel versions ECLIPSE 100 and ECLIPSE 300 are fundamentally different in two respects. Firstly, the well model calculations of ECLIPSE 100 are not parallelized. This has a minor impact on the scalability ECLIPSE 100. The second and more important difference is in the approach used in the parallelization of the linear solver. ECLIPSE 100 uses a modified nested factorization approach, which maintains full coupling across the grid at each linear iteration. ECLIPSE 300 uses a domain decomposition approach.

ECLIPSE 100 restricts you to a 1-dimensional domain decomposition, but maintains full coupling of the solution across the reservoir at each linear iteration, at the expense of increasing the work required per linear iteration by 1.3. We would however not expect the number of linear iterations to increase significantly as we increase the number of domains. ECLIPSE 100 parallel runs are restricted to three dimensional datasets.

ECLIPSE 300 allows a 2-dimensional decomposition and requires no extra work per linear iteration, at the cost of breaking the coupling across the reservoir at each linear iteration. We would therefore expect the number of linear iterations to increase with the number of domains. (Note that each domain is coupled to its neighbor only at each linear iteration, but the reservoir is fully coupled at the level of each Newton iteration.)

In both ECLIPSE simulators a balance has to be struck between increasing the amount of work per linear iteration and increasing the number of linear iterations.

ECLIPSE 100

ECLIPSE 100 partitions the reservoir either in the x or the y direction, whichever is the outer solver direction. It automatically divides the reservoir into domains with approximately equal numbers of active cells.

The number of planes allocated to each domain can be modified using the `DOMAINS` keyword in the `GRID` section. Since each domain is processed by an individual processor, it is advantageous to have broadly similar numbers of active cells within each domain. The partitions of the reservoir and the solver directions can be output to the Print file by including the `DOMAINS` mnemonic in `RPTGRID`, or graphically using the domains output to the `INIT` file.

At present the message passing option is based on the MPI (Message Passing Interface) system that is available on a wide range of computer systems, including networks of workstations (which may include machines from different vendors).

Options to improve performance

A number of options that may affect the performance of the code can be specified using the `PARAOPTS` keyword.

1 The use of a ‘tree’ or ‘hypercube’ algorithm for global reductions such as sums and maxima. In several places within the parallel code quantities have to be summed across domains/processors. This can either be done by sending messages from each processor to one processor (in practice processor 1) where they can be added and the result broadcast back to the other processors. Under some circumstances this can lead to a communications bottleneck at processor 1.

2 The use of the hypercube algorithm in which processor 2 sends its data to processor 1, and processor 4 sends its data to processor 3 which performs a sum or maximum operation on the available data and then forwards the results to processor 1. The disadvantage is that the algorithm has several levels to it, but it does reduce the number of message into a single processor. It is most likely to be useful in systems that have parallel communications hardware, such as an SP2.

3 The `DOMAINS` keyword can be used to control the size of domains.

Efficiency and performance estimation

The theoretical efficiency of the parallel option needs to be broken down into three components:

1 The modified linear solver algorithm may require more iterations than the purely serial solver (that is the algorithm may be less powerful). This effect will vary from model to model; however, experience to date suggests that the degradation in performance is small in most cases.

2 The modified algorithm requires more work even when the number of linear iterations remains the same. The extra work is required to allow the majority of the code to be parallelized. This increase is typically between 15% and 30% of the original run, for numbers of processors in the range 4 to 32. Two processors is a special case with typically no overhead, as no extra filling terms are required.

3 The critical issue for a parallel algorithm is the fraction of the code that remains serial. It is critical because this is the fraction that cannot be reduced by increasing the number of processors; it is the limit on the possible speed-up of the run.

The work that does not run in parallel consists of the calculations not associated with the grid block solution (for example wells and I/O). In addition, the section of the linear solver required to communicate information between domains is not wholly parallel, in the sense that some processors must remain idle.

The proportion of the total CPU time spent in the serial sections of the code can be investigated using the ECLIPSE timing facility. If the mnemonic '`CPU`' is specified in keyword `RPTSCHED`, ECLIPSE produces a timing report. This report is slightly different in the parallel case. Four extra mnemonics have been added:

Below `GINV`:

`GAM` is the tree elimination for the gamma calculation.

Below `SRCH`:

`FORW` is the parallel forward sweep through each domain

`TREE` is the tree elimination to communicate between domains

`BACK` is the parallel backward sweep through each domain.

An example is shown below:

```
=========================================================
TIMING REPORTS AT CPU=   955 SECS - ELAPSED=   989 SECS
=========================================================
ZZSS    1  0  0
               RR      1  0  0
SR      1  0  0
TS     20100100
              NX     52 99 99
                             WC    231  0  0
                             LMRS  231  1  1
                                           LMRI  231  0  0
                                           LMRO  231  0  0
                             CONV  231  0  0
                             LMJC  179  2  2
                                           LMJI  179  0  0
                                           LMJO  179  0  0
                             LXX   179 78 78
                                           WGIN  179  0  0
                                           GINV  179 19 18
                                                          GAM   179  2  2
                                         WRES  179  0  0
                                         ITER  179 59 59
                                                       NORM 1720  3  3
                                                       SRCH 1541 41 41
                                                                    FORW 1541 15 15
                                                                    TREE 1541  2  2
                                                                    BACK 1541 16 16
                                                       WADX 1541  0  0
                                                       ORTH 1541 12 12
                                                       UPDA 1541  3  4
                                         WSOL  179  0  0
                             XUIT  179  1  1
                             LCMI  179 16 16
XW     20  0  0
   31 READING END
```

ECLIPSE 100 optionally allows you to run the parallel algorithm on a single processor by specifying ‘S’ after the number of processors in the parallel code.

```
PARALLEL
16 'S' /
```

If a model is run on a single processor, the theoretical speed-up can be estimated from the fractions under the GAM and TREE mnemonics, provided that the well solution and I/O sections of the code only consume a negligible fraction of the CPU time. Note that the GAM and TREE sections of code are not genuine serial sections but include the serial component. The following application of Amdahl’s law can be thought of as a conservative measure of the theoretical speed-up.

Serial fraction: $S = GAM + TREE$ [EQ 50.1]

Theoretical parallel CPU time: $\left(\frac{1-S}{NP} + \frac{S}{NP-1}\log_2(NP)\right) CPU_{serial}$ [EQ 50.2]

where

NP is the number of processors (usually = number of domains)

The actual performance will be less than this theoretical value for a number of reasons, including:

1 Load balancing, where the amount of work on each processor is not exactly equal.

2 Overheads associated with the implementation on the parallel computer.

In the distributed memory case these include the latency and band-width limitations of the message passing system. In the shared memory case these include the start-up cost of the parallel loops and the bottle-neck of all processes accessing the memory at the same time.

ECLIPSE 300

In a parallel run the reservoir is partitioned areally into domains, each of which is assigned a processor. The default is to divide the reservoir into one-dimensional strips, along the reservoir's largest N_x or N_y direction. However, for large numbers of domains, a two-dimensional decomposition is desirable, to avoid thin strip domains. For example, with 16 processors, a 4-by-4 layout of domains can be realized by

```
NPROCX
4 /
NPROCY
4 /
```

If the basic simulation grid is 120 by 120 by 10, this will imply that each processor treats a domain of dimension 30 by 30 by 10.

For reservoirs with a significant number of inactive cells, the default partitioning is not sufficient to load-balance the domains; that is ensure all domains have roughly the same number of active cells and therefore work. It is possible to control further the way in which cells are assigned to domains. By default, the x- or y- dimension of the grid is equally divided. However, this may be over-ridden using the PSPLITX and PSPLITY keywords in the RUNSPEC section. Suppose, in the case above, we also specified

```
PSPLITX
60 120 /
40 80 120 /
40 60 80 120 /
/
```

The division of cells in the x-direction for the 4-processor case would then be as shown in Table 50.1. Note that this keyword also contains partitions for the 2 and 3 processor cases.

Table 50.1 The division of cells between processors using the PSPLITX keyword.

Process	Cells
1	1 to 40

Table 50.1 The division of cells between processors using the PSPLITX keyword.

Process	Cells
2	41 to 60
3	61 to 80
4	81 to 120

The keyword PSPLITY is analogous. The simulator will output good estimates of these keywords by using the PART mnemonic in the RPTSOL keyword. By performing an initial NOSIM run, then pasting the suggested PSPLITX and PSPLITY output in the RUNSPEC section, good load balancing can be achieved for all combinations of processors. The percentage active cell load balancing of the current run is reported to the PRT file. This is defined $\text{Active cell load balance \%} = \frac{\text{total active cells in reservoir}}{max(\text{active cells per partition}) \times \text{number of procs}} \times 100$

This figure represents the maximum parallel speed-up that can be achieved for a given run. For example, if the active cell load balance is 75% for a 4-processor job, then the maximum parallel speed-up that can be achieved is 3 (ignoring any super-linear speed-up that may be achieved by cache effects). The actual parallel speed-up achieved may be much lower than this. Consequently it is advisable to monitor this figure, especially if large parts of the reservoir are inactive. ECLIPSE 300 gives suggestions for better partitions if the PART mnemonic is put in the RPTSOL keyword in the SOLUTION section. The PRINT file contains suggestions on how the current partition may be improved. In addition the Debug file contains a list of all the possible combinations of PDIVX and PDIVY. This information can only be obtained once the GRID section is processed. A two-stage procedure is required to utilize the partition suggestions from ECLIPSE 300.

1. Run the simulator with default partitions with the PART mnemonic in the RPTSOL keyword (no SCHEDULE section is necessary).
2. If the active cell load balance is low, paste the relevant PDIVX and/or PDIVY output from the Debug file of the first run into the RUNSPEC section, and re-run.

To examine the partition of a particular run, the PART mnemonic can be placed in the OUTSOL keyword, and then plotted from the RESTART file using any standard package (for example, GRAF, FloViz). This is useful to determine if any wells cross partition boundaries.

Note the WARP linear solver splits each parallel partition into a number of sub-grids, which are connected through NNCs. Please refer to "The ECLIPSE 300 WARP linear solver" on page 949 for a detailed description.

Running LGRs in parallel (ECLIPSE 100)

Parallel domain decomposition runs can include local grids. These are defined in the normal way. By default, in ECLIPSE 100 each local grid is treated as a complete entity and is allocated to one of the slave processes. This will usually be the most efficient approach if there is a large number of local grids; if we have only a few LGRs, however, there will be a load imbalance. Each slave process can handle any number of local grids. Each LGR can be allocated manually to a specific process using the `IHOST` keyword. Alternatively, ECLIPSE 100 can allocate the LGRs automatically. While doing this ECLIPSE 100 attempts to balance the number of active refined cells.

In ECLIPSE 300 the refined cells inherit their host processor from the global grid. Consequently, LGRs are split across processors. If there is a large number of LGRs it can be difficult to achieve a good load balance.

Performance bottlenecks

There are several possible reasons for performance bottlenecks.

1. A poor active cell load balance. This can only be addressed by re-partitioning the grid. Some user intervention is required to improve the partitioning of any grid. The partition can be graphically examined from the RESTART file by putting the PART mnemonic in the OUTSOL keyword, and suggestions for better partitions can be acquired by putting the PART mnemonic in RPTSOL.

2. If completions of a well are on a different processor to the well head (specified, for example, by the COMPDAT and WELSPECS keywords) then off-processor completions will result, and the 10th argument of WELLDIMS needs to be set to the maximum allowed. Do not set this argument to too high a number. Off-processor completions will result in a significant communication overhead, and will consequently degrade parallel performance. If possible, these should be avoided, either by moving the location of the well head or re-partitioning. If all the horizontally-completed wells lie in the I or J direction, then re-partitioning may be a straightforward operation: just partition in the I or J direction to avoid the off-processor completions. However, off-processor completions are very often unavoidable if horizontally completed wells are present.

3. The parallel linear solver is a variant of nested factorization. The significant feature is that the serial and parallel algorithms are different. This results in some slight differences in results between the serial and parallel runs. More importantly, the parallel algorithm often requires more iterations to converge, and so more work is required by the parallel run which inevitably may result in some degradation of parallel performance. This can be monitored using the NLINEARS mnemonic in the SUMMARY section.

4. Disk output is a solely sequential activity, and so all processes within a parallel job will wait while one process writes to disk. Excessive output will significantly degrade overall parallel performance.

5. Numerical aquifers connect a group of cells, representing the aquifer, to the reservoir. If the group of cells (defined using AQUNUM keyword) or the connections from the aquifer to the reservoir (defined using AQUCON keyword) cross partition boundaries, then extra communication will result. This will degrade parallel performance. If such connections can be easily avoided it is advisable to do so.

6. LGRs are allowed with the parallel option. In ECLIPSE 300 the partition of the LGRs is “inherited” from the global grid. Any LGRs that lie across partition boundaries are automatically split. If many nested LGRs are present, then achieving a good active load balance can be problematic, given the constraints of the global partition. In addition, LGRs that are split across domain boundaries can have an adverse effect on the linear solver performance.

   In ECLIPSE 100 LGRs are processed as on one processor and the data swapped between processors. The default distribution of LGRs tries to place approximately equal numbers of refined cells on each processor. However, mixing LGRs in place and locally time stepped LGRs can destroy this load balancing as these are processed in completely different ways. Also, locally time stepped LGRs do not all take the same number of linear or non-linear iterations, or local time-steps. Hence it is possible that the smallest LGR in terms of the number of refined active cells may be the slowest to perform a given global time-step. In addition, the most difficult local grid to solve may change during the run. All this can destroy any attempt to load-balance the LGR calculations.

7 It is possible to connect any two cells of the grid through the NNC keyword. If the cells lie on different partitions, a noticeable communication overhead in conjunction with degraded linear solver performance may result.

Memory scaling

ECLIPSE 100 and ECLIPSE 300 allocate memory as needed using dynamic memory management. Therefore, the `MEMORY` keyword is not needed anymore.

The memory resources will reduce as the number of processors (as specified in the `PARALLEL` keyword) increases. This is a consequence of the individual domains becoming smaller as the number of processors increases. Thus less storage per domain is needed. How the memory requirements scale as the number of processors used for a particular job increases is very case-dependent, and is not linear. All the well and tabular data is stored simultaneously by all processes. Consequently, severely over-dimensioning the maximum number of wells or completions, for example, will waste memory on each process with a parallel job. This could become expensive as the number of processors used in a parallel job is increased.

Summary of keywords

RUNSPEC section

The keywords are:

Table 50.2 RUNSPEC section

Simulator	Keyword	Description
ECLIPSE 300 only	NPROCX	Defines the number of processors to be used in the x-direction.
ECLIPSE 300 only	NPROCY	Defines the number of processors to be used in the y-direction.
	PARALLEL	Initializes the Parallel option.
ECLIPSE 300 only	PDIVX	Defines the partitioning in the x-direction.
ECLIPSE 300 only	PDIVY	Defines the partitioning in the y-direction.
ECLIPSE 300 only	PSPLITX	Specifies multiple domain decomposition in the x-direction.
ECLIPSE 300 only	PSPLITY	Specifies multiple domain decomposition in the y-direction.

GRID section

The keywords are:

Table 50.3 GRID section

Simulator	Keyword	Description
ECLIPSE 100 only	DOMAINS	Defines the partitioning of the grid.
ECLIPSE 100 only	IHOST	Allocates LGRs to specific processes.
ECLIPSE 100 only	PARAOPTS	Permits some fine tuning of the simulator.
ECLIPSE 100 only	RPTGRID	Controls output from the GRID section. The DOMAINS mnemonic outputs the domain split.

SOLUTION section

The keywords are:

Table 50.4 SOLUTION section

Simulator	Keyword	Description
ECLIPSE 300 only	RPTRST	Controls solution output for GRAF and flexible restarts. Mnemonic PART outputs domain partitions.
ECLIPSE 300 only	OUTSOL	Controls solution output for GRAF and flexible restarts. Mnemonic PART outputs domain partitions.
ECLIPSE 300 only	RPTSOL	Controls output from the SOLUTION section. Mnemonic PART outputs domain partitions.

Petro-elastic Model

Chapter 51

Introduction

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL

The Petro-elastic Model option provides the ability to generate simulated seismic-derived quantities (such as acoustic impedance) in each active gridcell.

Note You must have an `eclipse4d` license in order to access this feature.

The acoustic response of the reservoir is a combination of the acoustic responses of the different rocks and fluids that constitute the reservoir.

In the Petro-elastic model, conventional simulation properties (such as pressure, fluid density, fluid saturations and effective porosity) are used to calculate the effective acoustic response of the fluids. This is then combined with the acoustic response of the rocks (both frame and minerals) to give the overall acoustic response of the reservoir.

The acoustic response is calculated and output for each gridcell in the simulation model.

Assumptions

Gassmann fluid substitution equation

The Petro-elastic model uses the Gassmann fluid substitution equation.

The use of this equation relies on the following assumptions:

1. The seismic frequencies used are sufficiently low (less than 100 Hz) such that the induced pore pressures are equilibrated throughout the pore space (that is there is sufficient time for the pore volume fluid to flow and eliminate wave-induced pore pressure gradients).
2. The rock is isotropic.
3. There is a homogeneous mineral modulus.
4. The fluid bearing rock is completely saturated.

Fluid viscosity

The fluids are assumed to be not “too” viscous so that the shear stiffness of the reservoir is entirely due to the shear stiffness of the dry rock frame.

The Petro-elastic Model

Overview

Most of the theory of this model (that is the assumptions and equations) can be found in [Ref. 90].

The Petro-elastic model can be decomposed into four levels. These are:

- "Level 1: Impedances and velocities" on page 765
- "Level 2: Effective bulk/shear modulus and density" on page 766
- "Level 3: Rock properties and fluid properties" on page 767
- "Level 4: Individual fluid phase properties" on page 769

Level 1 is the output and it requires results from the calculations in Level 2. In turn, Level 2 requires the results from the calculations in Level 3 etc.

Level 1: Impedances and velocities

The main outputs of the Petro-elastic model are:

1 I_P - the impedance to pressure waves for a particular simulation gridcell

2 I_S - the impedance to shear waves for a particular simulation gridcell

3 ν - the Poisson's ratio for a particular simulation gridcell

These are defined as

$$I_P = \rho V_P \qquad \text{[EQ 51.1]}$$

$$I_S = \rho V_S \qquad \text{[EQ 51.2]}$$

$$\nu = \frac{\frac{1}{2}V_P^2 - V_S^2}{V_P^2 - V_S^2} \qquad \text{[EQ 51.3]}$$

where

- ρ is the effective density of all the material (fluids, minerals, frame) in the gridcell
- V_P is the effective pressure wave velocity in the simulation gridcell
- V_S is the effective shear wave velocity in the simulation gridcell

These effective velocities are calculated by

$$V_P = \sqrt{\frac{K + (4G/3)}{\rho}} \qquad \text{[EQ 51.4]}$$

$$V_S = \sqrt{\frac{G}{\rho}} \qquad \text{[EQ 51.5]}$$

where

- K is the effective bulk modulus of the simulation gridcell
- G is the effective shear modulus of the simulation gridcell

Level 2: Effective bulk/shear modulus and density

Effective bulk modulus

The effective bulk modulus of a simulation gridcell is a combination of the effects of the fluids, the frame and the minerals (for example, sand, shale, silt) in the gridcell. The combination is performed using the Gassmann fluid substitution equation. This relies on the assumptions listed in "Assumptions" on page 764.

The following form of Gassmann's equation is used in the Petro-elastic model:

$$K = K_{frame} + \frac{\left(1 - \frac{K_{frame}}{K_{minerals}}\right)^2}{\frac{\phi_{eff}}{K_{fluid}} + \frac{(1-\phi_{eff})}{K_{minerals}} - \frac{K_{frame}}{(K_{minerals})^2}} \qquad \text{[EQ 51.6]}$$

where

- K_{frame} is the effective bulk modulus of the dry rock frame for this gridcell
- $K_{minerals}$ is the effective bulk modulus of the minerals in this gridcell
- K_{fluid} is the effective bulk modulus of the pore fluid in this gridcell
- ϕ_{eff} is the effective porosity of this gridcell

Effective shear modulus

With the assumption that the fluids in the gridcell are not too viscous, then the overall shear modulus is just the shear modulus of the dry rock frame, that is

$$G = G_{frame} \qquad \text{[EQ 51.7]}$$

where

- G_{frame} is the shear modulus of the dry rock frame for this gridcell

Effective density

The effective density in a gridcell is given by the effective porosity weighted average of the fluid density and the mineral density in that gridcell, that is

$$\rho = \phi_{eff} \cdot \rho_{fluid} + (1 - \phi_{eff}) \cdot \rho_{minerals} \qquad \text{[EQ 51.8]}$$

where

- $\rho_{minerals}$ is the effective density of the minerals in this gridcell
- ρ_{fluid} is the effective density of the fluids in this gridcell

The effective density of the minerals is supplied directly as an input for each petro-elastic region as part of the Petro-elastic model definition in the simulation model.

Level 3: Rock properties and fluid properties

Effective porosity

The calculation of the effective porosity for a gridcell is based on the porosity at the reference pressure, ϕ_0 ,in the gridcell and the effects of rock compressibility or rock compaction (depending on which type of pressure effect is present in the model).

Rock compressibility

If the simulation model contains the ROCK keyword, then rock compressibility is used in the calculation of the effective porosity.

$$\phi_{eff} = \phi_0\left(1 + X + \frac{X^2}{2}\right) \qquad \text{[EQ 51.9]}$$

where

$$X = C(P - P_{ref}) \qquad \text{[EQ 51.10]}$$

and

- C is the rock compressibility for the gridcell
- P is the pressure in the gridcell
- P_{ref} is the rock compressibility reference pressure for the gridcell

In each gridcell, the porosity at the reference pressure, and the pressure itself, are available within the simulator. The rock compressibility and its reference pressure are also available as they are specified in the ROCK keyword.

Rock compaction

If the simulation model contains the ROCKCOMP keyword, then rock compaction is used in the calculation of the effective porosity.

$$\phi_{eff} = \phi_0 C(P)$$

where

- $C(P)$ is the rock compaction pore volume multiplier for the gridcell (this is specified as tables of multipliers against pressure in the ROCKTAB keyword)
- P is the pressure in the gridcell

Effective fluid density

The effective fluid density is the saturation weighted average of the individual fluid component densities, that is,

$$\rho_{fluid} = S_o\rho_o + S_w\rho_w + S_g\rho_g \qquad \text{[EQ 51.11]}$$

where

- $S_{o/w/g}$ is the saturation of oil/water/gas in the gridcell
- $\rho_{o/w/g}$ is the density of oil/water/gas in the gridcell

These quantities are all available within the simulator for each gridcell.

Effective bulk modulus of the fluid

Two methods have been implemented for calculating the effective bulk modulus of the fluid.

The choice of which method to use is supplied directly as an input for each petro-elastic region as part of the Petro-elastic model definition in the simulation model.

Generalized Wood's Law

This is simply a saturation weighted harmonic average of the individual fluid bulk moduli, that is

$$\frac{1}{K_{fluid}} = \frac{S_o}{K_o} + \frac{S_w}{K_w} + \frac{S_g}{K_g} \qquad \text{[EQ 51.12]}$$

where

- $K_{o/w/g}$ is the bulk modulus of the oil/water/gas in the gridcell

Wood's Law with Brie et al

This uses a saturation weighted harmonic average for the liquid phases (that is, oil and water) with a separate equation used to incorporate the effect of the gas phase, that is

$$K_{fluid} = (K_{liquid} - K_g)(1 - S_g)^e + K_g \qquad \text{[EQ 51.13]}$$

where

- e is the "patchy saturation" exponent
- K_{liquid} is the combined bulk modulus of the liquid phases, given by

$$\frac{S_o + S_w}{K_{liquid}} = \frac{S_o}{K_o} + \frac{S_w}{K_w} \qquad \text{[EQ 51.14]}$$

The patchy saturation exponent is supplied directly as an input for each petro-elastic region as part of the petro-elastic model definition in the simulation model. This equation was first presented in [Ref. 91].

Effective bulk modulus of the minerals

A linear function of effective porosity is used to specify the effective bulk modulus of the minerals in the gridcell, that is

$$K_{minerals} = C_1 \phi_{eff} + C_0 \qquad \text{[EQ 51.15]}$$

where

- C_0 is the zeroth order coefficient of effective porosity
- C_1 is the first order coefficient of effective porosity

The zeroth and first order coefficients of effective porosity are supplied directly as an input for each petro-elastic region as part of the petro-elastic model definition in the simulation model.

Effective shear modulus of the minerals

The effective shear modulus of the minerals, $G_{minerals}$, is supplied directly as an input for each petro-elastic region as part of the petro-elastic model definition in the simulation model.

Effective bulk modulus of the frame

The effective bulk modulus of the dry rock frame is given by

$$(K_{frame})^{\alpha} = C_K(K_{minerals})^{\alpha} + \sum_{i=0}^{I} \{C_{K,i}(P) \cdot \phi_{eff}^{i}\} \qquad \text{[EQ 51.16]}$$

where

- α is a specified exponent
- C_K is the coefficient of the $K_{minerals}$ term in the calculation of K_{frame}
- $C_{K,i}(P)$ are specified tabular functions of pressure

α is usually -1 or +1 and specifies whether the dry rock frame and mineral bulk moduli are related harmonically or arithmetically. Both α and the tabular functions are supplied directly as an input for each petro-elastic region as part of the petro-elastic model definition in the simulation model.

Effective shear modulus of the frame

The effective shear modulus of the dry rock frame is given by

$$(G_{frame})^{\alpha} = C_G(G_{minerals})^{\alpha} + \sum_{i=0}^{I} \{C_{G,i}(P) \cdot \phi_{eff}^{i}\} \qquad \text{[EQ 51.17]}$$

where

- α is the same exponent as specified in [EQ 51.16]
- C_G is the coefficient of the $G_{minerals}$ term in the calculation of G_{frame}
- $C_{G,i}(P)$ are specified tabular functions of pressure

Both α and the tabular functions are supplied directly as an input for each petro-elastic region as part of the petro-elastic model definition in the simulation model.

Level 4: Individual fluid phase properties

The calculations for the bulk moduli of the oil, water and gas in a gridcell are based on the empirical equations given in [Ref. 90].

Note In the descriptions of the empirical equations in this section, the relevant units for physical quantities are given. These units are only used for internal calculation purposes. You do not have to input your petro-elastic property data in these units.

Bulk modulus of oil

The bulk modulus of oil in a gridcell is given by

$$K_o = \rho_o V_o^2 \qquad \text{[EQ 51.18]}$$

where

- ρ_o is the density of oil (g/cm^3) in the gridcell
- V_o is the velocity of sound in oil in the gridcell given (in m/s) by

$$V_o = 2096\sqrt{\frac{\rho_{o,R}}{2.6-\rho_{o,R}}} - (3.7\cdot T) + (4.64\cdot P)$$

$$+ 0.0115\cdot\left\{\left(4.12\sqrt{\frac{1.08}{\rho_{o,R}}-1}\right)-1\right\}\cdot T\cdot P \qquad \text{[EQ 51.19]}$$

where

- T is the temperature (^{o}C) in the gridcell. This is supplied directly as an input for each petro-elastic region as part of the petro-elastic model definition in the simulation model.
- P is the pressure (MPa) in the gridcell
- $\rho_{o,R}$ is the density of oil (g/cm^3) in the gridcell at reference conditions (15.6^{o}C and atmospheric pressure). This is calculated by first correction for temperature effects

$$\rho_{o,P} = \rho_o\{0.972 + 3.81\times 10^{-4}(T+17.78)^{1.175}\} \qquad \text{[EQ 51.20]}$$

and then correcting for pressure effects using

$$\rho_{o,P} = \rho_{o,R} + (0.00277P - 1.71\times 10^{-7}P^3)(\rho_{o,R}-1.15)^2 + 3.49\times 10^{-4}P$$

Bulk modulus of water

The bulk modulus of water in a gridcell is given by

$$K_w = \rho_w V_w^2 \qquad \text{[EQ 51.21]}$$

where

- V_w is the velocity of sound waves in water in the gridcell given (in m/s) by

$$V_w = V_{pw} + C_S\cdot S + (780 - 10P + 0.16P^2)\cdot S^{1.5} - 1820S^2 \qquad \text{[EQ 51.22]}$$

where

- S is the salinity of the water in the gridcell (in ppm/10^6, that is, as a fraction of 1)
- V_{pw} is the velocity of sound waves in pure water in the gridcell

Again, as this is an empirical formula, P must be in MPa. S is supplied directly as an input for each petro-elastic region as part of the petro-elastic model definition in the simulation model, while C_S and V_{pw} are given by

$$C_S = 1170 - 9.6T + 0.055T^2 - 8.5\times 10^{-5}T^3 + 2.6P - 0.0029TP - 0.0476P^2$$

where T is in ^{o}C, and

$$V_{pw} = \sum_{i=0}^{4} \left\{ \sum_{j=0}^{3} (w_{ij} T^i P^j) \right\} \qquad \text{[EQ 51.23]}$$

where

$$\begin{aligned}
w_{00} &= 1402.85 & w_{02} &= 3.437\times10^{-3} \\
w_{10} &= 4.871 & w_{12} &= 1.739\times10^{-4} \\
w_{20} &= -0.04783 & w_{22} &= -2.135\times10^{-6} \\
w_{30} &= 1.487\times10^{-4} & w_{32} &= -1.455\times10^{-8} \\
w_{40} &= -2.197\times10^{-7} & w_{42} &= 5.230\times10^{-11} \\
w_{01} &= 1.524 & w_{03} &= -1.197\times10^{-5} \\
w_{11} &= -0.0111 & w_{13} &= -1.628\times10^{-6} \\
w_{21} &= 2.747\times10^{-4} & w_{23} &= 1.237\times10^{-8} \\
w_{31} &= -6.503\times10^{-7} & w_{33} &= 1.327\times10^{-10} \\
w_{41} &= 7.987\times10^{-10} & w_{34} &= -4.614\times10^{-13}
\end{aligned} \qquad \text{[EQ 51.24]}$$

Bulk modulus of gas

The bulk modulus of gas in a gridcell is given (in MPa) by

$$K_g = \frac{P\gamma}{1 - \left(\frac{P_r}{Z} \cdot f\right)} \qquad \text{[EQ 51.25]}$$

where

$$GRAV = \frac{\rho_{gas|surface}}{\rho_{air|surface}}$$

$$P_r = \frac{P}{4.892 - (04048 \cdot GRAV)}$$

$$T_r = \frac{T + 273.15}{94.72 + (170.75 \cdot GRAV)}$$

$$f = c \cdot d \cdot m + a$$

$$Z = (a \cdot P_r) + b + (c \cdot d)$$

$$a = 0.03 + 0.00527(3.5 - T_r)^3$$

$$b = 0.642 T_r - 0.007 T_r^4 - 0.52$$

$$c = 0.109(3.85 - T_r)^2$$

$$d = \exp\left\{-\left[0.45 + 8\left(0.56 - \frac{1}{T_r}\right)^2\right]\frac{P_r^{1.2}}{T_r}\right\}$$

$$m = -1.2\left\{0.45 + 8\left(0.56 - \frac{1}{T_r}\right)^2\right\}\frac{P_r^{0.2}}{T_r}$$

$$\gamma = 0.85 + \frac{5.6}{P_r + 2} + \frac{27.1}{(P_r + 3.5)^2} - 8.7\exp\{-0.65(P_r + 1)\}$$

$GRAV$ is available within the simulator. Again, since this is an empirical formula, P is in MPa, and T is in ^{o}C.

Implementation of the Petro-elastic model

User Interface

Inputs

The following values, which are not normally present in the simulation model, are required to specify the Petro-elastic model:

These are specified as constants on each petro-elastic region

- S - the salinity of the water in [EQ 51.22]
- T - the temperature in [EQ 51.19], [EQ 51.22], and [EQ 51.25] (this may be available elsewhere in the simulation model but the petro-elastic temperature is specified separately)
- $\rho_{minerals}$ - the effective density of the minerals in [EQ 51.8]
- C_1 - the first order coefficient of effective porosity in the calculation of the effective bulk modulus of the minerals in [EQ 51.15]
- C_0 - the zeroth order coefficient of effective porosity in the calculation of the effective bulk modulus of the minerals in [EQ 51.15]
- C_K is the coefficient of the $K_{minerals}$ term in the calculation of K_{frame} in [EQ 51.16]
- C_G - the coefficient of the $G_{minerals}$ term in the calculation of G_{frame} in [EQ 51.17]
- $G_{minerals}$ - the effective shear modulus of the minerals in [EQ 51.7]
- α - the exponent in the calculation of the effective bulk/shear modulus of the dry rock frame in [EQ 51.16] and [EQ 51.17]
- e - the “patchy saturation” exponent in [EQ 51.13] (only used if Wood’s Law with Brie et al is requested)
- Switch to select between Generalized Wood’s Law and Wood’s Law with Brie et al in the calculation of the effective bulk modulus of the fluid
- $C_{K,i}(P)$ - the tabular functions of pressure in the calculation of the effective bulk modulus of the dry rock frame in [EQ 51.16]
- $C_{G,i}(P)$ - the tabular functions of pressure in the calculation of the effective shear modulus of the dry rock frame in [EQ 51.17]

Outputs

The output from the petro-elastic model depends on which simulated seismic-derived properties are requested (from the list of P-wave impedance, S-wave impedance and Poisson’s ratio). When a simulated seismic-derived property is requested, it is output as a value on each active gridcell at each report time.

One use of these seismic-derived quantities is in history-matching using seismic data. In this application, gradients of simulated seismic-derived quantities with respect to uncertain reservoir model parameters may be required. In addition to simulated seismic-derived quantities, the Petro-elastic model can calculate and output these gradients. For these calculations to be performed, you must specify a petro-elastic model (using the keywords described in "Summary of keywords" on page 774) and request gradients. For further information on requesting gradients, see "The Gradient Option" on page 463.

Summary of keywords

RUNSPEC section

The keywords are:

Table 51.1 RUNSPEC section

Keyword	Description
PEDIMS	Sets the number of petro-elastic regions in the simulation, sets table dimensions, in terms of the number of rows, for the PEKTABx and PEGTABx keywords.

REGIONS section

The keywords are:

Table 51.2 REGIONS section

Keyword	Description
PENUM	Specifies the petro-elastic region number for each gridcell in the simulation model.

PROPS section

The keywords are:

Table 51.3 PROPS section

Keyword	Description
PECOEFS	Specifies a variety of fluid and rock properties, and also options to be used in the petro-elastic calculations, for each petro-elastic region in the simulation model.
PEKTABx	Specifies the frame bulk modulus functions for each petro-elastic region in the simulation model.
PEGTABx	Specifies the frame shear modulus functions for each petro-elastic region in the simulation model.

SOLUTION/SUMMARY section

The keywords are:

Table 51.4 SOLUTION/SUMMARY section

Keyword	Description
RPTRST	Controls output of properties (including simulated seismic-derived properties) to RESTART files. The petro-elastic model specific mnemonics for this keyword are shown in Table 51.5.
GRADRESV	Specifies which gradients of solution data are output to the RESTART files. The Petro-elastic model specific mnemonics for this keyword are shown in Table 51.6.

Note For each of these properties, you must also request the DEN item in the RPTRST keyword.

Table 51.5 RPTRST options

Input Keyword Or Mnemonic	Output	E100	E300
ACIP	Output pressure-wave impedance	x	
ACIS	Output shear-wave impedance	x	
POIS	Output Poisson's ratio	x	

.

Table 51.6 GRADRESV options

Input Keyword Or Mnemonic	Output	E100	E300
ACIP	Output gradient of pressure-wave impedance	x	
ACIS	Output gradient of shear-wave impedance	x	
POIS	Output gradient of Poisson's ratio	x	

The Polymer Flood Model

Chapter 52

Introduction

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL

The main objective of polymer injection during water flooding of oil reservoirs is to decrease the mobility of the injected water. This decrease results in a more favorable fractional flow curve for the injected water, leading to a more efficient sweep pattern and reduced viscous fingering. Certain plugging effects within highly permeable layers may also occur and result in the diversion of the injected water into less permeable zones of the reservoir.

The mobility decrease of the injected water resulting from the addition of polymer is due to two effects. Firstly, the viscosity of the polymer solution is higher than that of pure water (the viscosity of the polymer solution increases as the concentration of the polymer in the water increases). Secondly, the rock permeability to water is reduced after the passage of a polymer solution through the rock material (the permeability to oil is, however, largely unaffected). Both effects combine to reduce the value of the water mobility while that for the oil is unaltered.

To achieve maximum efficiency, the polymer solution is often applied in the form of a tapered slug. At the front edge of the slug, the displacement is stable but the interface between the water and the polymer solution smears due to physical dispersion of the polymer. At the rear edge, the mobility ratio is unfavorable and is dominated by viscous fingering. Both effects cause deterioration of the slug, and are modeled in ECLIPSE by means of a mixing parameter applied to the viscosity terms in the fluid flow equations.

Certain polymer solutions, such as those of the PA type, exhibit a strong sensitivity to the presence of certain salts. In particular, sodium chloride can influence the viscosity characteristics of PA solutions to such an extent that in highly saline reservoirs it is often necessary to preflush with fresh water to reduce the exposure of the polymer solution to the reservoir brine. The ECLIPSE model allows you to investigate the effect of varying brine concentrations on the efficiency of the polymer flood. The study of the interaction of brine with the polymer solution is, however, optional within the model. It should be noted that the effect of temperature variations on the behavior of the polymer solution is currently ignored.

When a polymer solution is injected into the reservoir some of the long chain molecules constituting the polymer are adsorbed onto the rock surfaces. Mechanical entrapment of some of the large molecules at the entrance to small pore throats may also occur and account for an apparent loss of polymer from the invading solution. Experimentally, the reservoir rock material is believed to retain a specific capacity of polymer. The main effects of polymer loss occur at the leading edges of the polymer slug where a stripped water bank is created and the slug width is gradually reduced in time. Some desorption effects can occur as the trailing edge of the slug passes but these effects are usually small compared with the adsorption losses. ECLIPSE allows you to either neglect the desorption effect or to allow the adsorption isotherm to be retraced after passage of the polymer slug.

A further effect caused by the adsorption and entrapment processes is a reduction in the relative permeability of the polymer solution. The reduction results from an interaction between the aqueous solution and the polymer retained by the rock material. For modeling purposes it will be assumed that the reduction in permeability to the polymer solution is proportional to the quantity of polymer lost to the rock material. The permeability of the rock to water is thus permanently reduced after the passage of a polymer slug compared to its value before the passage. Experimentally, it is found that only a very small change occurs to the hydrocarbon relative permeability and the ECLIPSE model assumes that the change is negligible.

In core flooding experiments, it is often observed that injected polymer slugs break through to producers earlier than tracer slugs (for example, NaCl). The polymer fluid velocity is higher than that of the tracer fluid within the porous medium and this is due to the fact that only a fraction of the total pore space is available to the polymer fluid. As the inaccessible pore space to the polymer fluid increases, the effective polymer velocity through the rock increases and leads to a faster breakthrough of polymer. This chromatographic effect is modeled by assuming that the dead pore space is constant for each rock type:

- and either does not exceed the corresponding irreducible water saturation (pre 2008.2).
- or independent of the water saturation (post 2008.2 included)

The rheology of polymer solutions is not simple. At low flow rates the viscosity of the solution is approximately constant and depends only on the concentration of polymer in the solution. At higher flow rates the solution viscosity reduces in a reversible (elastic) manner. At even higher velocities the large polymer molecules begin to break up, and the viscosity reduction becomes irreversible (plastic). The effects tend to be greatest in the vicinity of injection wells where the fluid velocity is greatest, and so is the shear rate.

The polymer flood simulation model

The flow of the polymer solution through the porous medium is assumed to have no influence on the flow of the hydrocarbon phases. The standard black-oil equations are therefore used to describe the hydrocarbon phases in the model.

Modification is required to the standard aqueous (water) equation and additional equations are needed to describe the flow of polymer and brine within the finite difference grid. The water, polymer and brine equations used in the model are as follows:

$$\frac{d}{dt}\left(\frac{VS_w}{B_rB_w}\right) = \sum\left[\frac{Tk_{rw}}{B_w\ \mu_{w\ \mathrm{eff}}R_k}(\delta P_w - \rho_w g D_z)\right] + Q_w \qquad \text{[EQ 52.1]}$$

$$\frac{d}{dt}\left(\frac{V^*S_wC_p}{B_rB_w}\right) + \frac{d}{dt}\left(V\rho_rC_p^a\frac{1-\phi}{\phi}\right) = \sum\left[\frac{Tk_{rw}}{B_w\ \mu_{p\ \mathrm{eff}}R_k}(\delta P_w - \rho_w g D_z)\right]C_p + Q_wC_p \qquad \text{[EQ 52.2]}$$

$$\frac{d}{dt}\left(\frac{VS_wC_n}{B_rB_w}\right) = \sum\left[\frac{Tk_{rw}C_n}{B_w\ \mu_{s\ \mathrm{eff}}R_k}(\delta P_w - \rho_w g D_z)\right] + Q_wC_n \qquad \text{[EQ 52.3]}$$

$$V^* = V\left(1 - \frac{S_{\mathrm{dpv}}}{S_w}\right) \qquad \text{pre-2008.2} \qquad \text{[EQ 52.4]}$$

$$V^* = V(1 - S_{\mathrm{dpv}}) \qquad \text{post-2008.2} \qquad \text{[EQ 52.5]}$$

where

S_{dpv} denotes the dead pore space within each grid cell

C_p^a denotes the polymer adsorption concentration

ρ_r denotes the mass density of the rock formation

ϕ denotes the porosity

ρ_w denotes the water density

Σ denotes the sum over neighboring cells

R_k denotes the relative permeability reduction factor for the aqueous phase due to polymer retention

C_p, C_n denote the polymer and salt concentrations respectively in the aqueous phase

$\mu_{a\mathrm{eff}}$ denotes the effective viscosity of the water (a=w), polymer (a=p) and salt (a=s).

D_z is the cell center depth.

B_r, B_w are the rock and water formation volumes

T is the transmissibility

k_{rw} is the water relative permeability

S_w is the water saturation

V is the block pore volume

Q_w is the water production rate

P_w is the water pressure

g is the gravity acceleration

The model makes the assumption that the density and formation volume factor of the aqueous phase are independent of the polymer and salt concentrations. The polymer solution, reservoir brine and the injected water are represented in the model as miscible components in the aqueous phase, where the degree of mixing is specified through the viscosity terms in the conservation equations.

The principal effects of polymer and salt on the flow of the aqueous phase are represented by equations [EQ 52.1] to [EQ 52.4] above. The fluid viscosities ($\mu_{w\ \mathrm{eff}}$, $\mu_{p\ \mathrm{eff}}$, $\mu_{s\ \mathrm{eff}}$) are dependent on the local concentrations of salt and polymer in the solution. Polymer adsorption is represented by the additional mass accumulation term on the left hand side of equation [EQ 52.2]. The adsorption term requires the user to specify the adsorption isotherm, C_p^a for each rock type. The effect of pore blocking and adsorption on the aqueous phase relative permeability is treated through the term, R_k, which requires the input of a residual resistance factor for each rock type.

The equations solved for the polymer model are a discretized form of the differential equations [EQ 52.1] - [EQ 52.4]. In order to avoid numerical stability problems that could be triggered by strong changes in the aqueous phase properties over a time step (resulting from large changes in the local polymer/salt concentrations) a fully implicit time discretization is used, which is therefore free from this type of instability.

Treatment of fluid viscosities

The viscosity terms used in the fluid flow equations contain the effects of a change in the viscosity of the aqueous phase due to the presence of polymer and salt in the solution. However, to incorporate the effects of physical dispersion at the leading edge of the slug and also the fingering effects at the rear edge of the slug the fluid components are allocated effective viscosity values that are calculated using the Todd-Longstaff technique.

Consider the effective polymer viscosity term. The viscosity of a fully mixed polymer solution as an increasing function of the polymer concentration in solution ($\mu_m(C_p)$) must be input. The viscosity of the solution at the maximum polymer concentration also needs to be specified and denotes the injected polymer concentration in solution (μ_p). The effective polymer viscosity is taken to be

$$\mu_{p,\,\mathrm{eff}} = \mu_m(C_p)^{\omega} \cdot \mu_p^{1-\omega} \qquad \text{[EQ 52.6]}$$

where

ω is the Todd-Longstaff mixing parameter.

The mixing parameter is useful in modeling the degree of segregation between the water and the injected polymer solution. If $\omega = 1$ then the polymer solution and water are fully mixed in each block. If $\omega = 0$ the polymer solution is completely segregated from the water.

The partially mixed water viscosity is calculated in an analogous manner using the fully mixed polymer viscosity and the pure water viscosity (μ_w),

$$\mu_{w,e} = \mu_m(C_p)^{\omega} \cdot \mu_w^{1-\omega} \qquad \text{[EQ 52.7]}$$

In order to calculate the effective water viscosity to be inserted into [EQ 52.1], the total water equation is written as the sum of contributions from the polymer solution and the pure water. The following expression then gives the effective water viscosity to be inserted into [EQ 52.1]:

$$\frac{1}{\mu_{w,\,\mathrm{eff}}} = \frac{1-\overline{C}}{\mu_{w,e}} + \frac{\overline{C}}{\mu_{p,\,\mathrm{eff}}} \qquad \text{[EQ 52.8]}$$

$$\overline{C} = \frac{C_p}{C_{p,\,\max}} \qquad \text{[EQ 52.9]}$$

where

$\overline{C}$ is the effective saturation for the injected polymer solution within the total aqueous phase in the cell.

If the brine option is active, the above expressions still hold for the effective polymer and water viscosity terms. The injected salt concentration needs to be specified in order to evaluate the maximum polymer solution viscosity, μ_p. The effective salt component viscosity to be used in [EQ 52.3] is set equal to the effective water viscosity.

Treatment of polymer adsorption

Adsorption is treated as an instantaneous effect in the model. The effect of polymer adsorption is to create a stripped water bank at the leading edge of the slug. Desorption effects may occur as the slug passes. The isotherm adsorption can be specified as:

- either a look-up table of adsorbed alkaline as a function of alkaline concentration using the PLYADS keyword
- or by a generic analytical adsorption model using the ADSORP keyword with POLYMER as an argument. This model enables the adsorption to depend on the salinity and rock permeability. When used in conjunction with the Multi-Component Brine model (ECLMC keyword in the RUNSPEC section) an effective salinity for polymer is calculated, which can then be used in the adsorption isotherm. Because the standard brine option and the multi-component brine use different salinity concentration definitions (massic concentration versus molar concentration), care is needed in the interpretation of the coefficients in this model.

If desorption is prevented (4th argument of the PLYROCK keyword) then the adsorbed polymer concentration may not decrease with time. If desorption is allowed then each grid block retraces the adsorption isotherm as the alkaline concentration rises and falls in the cell.

Treatment of permeability reductions and dead pore volume

The adsorption process causes a reduction in the permeability of the rock to the passage of the aqueous phase and is directly correlated to the adsorbed polymer concentration. In order to compute the reduction in rock permeability, you are required to specify the residual resistance factor (RRF) for each rock type (2nd argument of the PLYROCK keyword). The actual resistance factor can then be calculated:

$$R_k = 1.0 + (\mathrm{RRF} - 1.0)\frac{C_p^a}{C_p^{a\max}} \qquad \text{[EQ 52.10]}$$

The value of the maximum adsorbed concentration, $C_p^{a\max}$, depends on the rock type and needs to be specified as the 5th argument of the PLYROCK keyword. This value must be non zero.

The dead pore volume (1st argument of the PLYROCK keyword) must also be specified for each rock type. It represents the amount of total pore volume in each grid cell that is inaccessible to the polymer solution. The effect of the dead pore volume within each cell is to cause the polymer solution to travel at a greater velocity than inactive tracers embedded in the water. This chromatographic effect is modeled by assuming that the dead pore space is constant for each rock type:

- and either does not exceed the corresponding irreducible water saturation (pre 2008.2)
- or is independent of the water saturation (post 2008.2 included).

Treatment of the non-Newtonian rheology

You can choose between two models to take into account some of the non-Newtonian rheology behavior reported for polymer solutions.

Model 1

This model targets the shear thinning of polymer that has the effect of reducing the polymer viscosity at higher flow rates. It assumes that shear rate is proportional to the flow viscosity. This assumption is not valid in general, as for example, a given flow in a low permeability rock will have to pass through smaller pore throats than the same flow in a high permeability rock, and consequently the shear rate will be higher in the low permeability rock. However for a single reservoir this assumption is probably reasonable.

Model 2

The second model is the Herschel-Bulkley model that can be used to model shear thinning and thickening as well as yield stress, dependent on polymer concentration. Please refer to "Non-Newtonian fluid flows" on page 723 for more details.

Model 1 description

Here we describe Model 1.

The water flow velocity is calculated as:

$$V_w = B_w \cdot \frac{F_w}{\phi A} \qquad \text{[EQ 52.11]}$$

where

F_w is the water flow rate in surface units

B_w is the water formation volume factor

ϕ is the average porosity of the two cells

A is the flow area between two cells.

The reduction in viscosity of the polymer solution is assumed to be reversible as a function of the water velocity. The resulting shear viscosity of the polymer solution is calculated as:

$$\mu_{sh} = \mu_{w,\,eff}\left[\frac{1+(P-1)M}{P}\right] \qquad \text{[EQ 52.12]}$$

where

μ_{sh} is the shear viscosity of the polymer solution (water+polymer)

$\mu_{w,\,eff}$ is the effective water viscosity (see [EQ 52.8])

P is the viscosity multiplier assuming no shear effect (entered using the `PLYVISC` or `PLYVISCS` keywords)

M is the shear thinning multiplier supplied in the `PLYSHEAR` keyword.

Note Note that for $M = 1$, or no shear thinning, we recover the effective water viscosity, and for $M = 0$, or maximum shear thinning, the shear viscosity is

---

$$\mu_{sh} = \frac{\mu_{w,\,eff}}{P} \qquad \text{[EQ 52.13]}$$

which corresponds to the minimum viscosity that can be obtained. If the polymer concentration is zero (P=1) we recover the effective water viscosity, which in that particular case equals the water viscosity.

The well inflows are treated in a manner analogous to the treatment of block to block flows. The viscosity of the polymer solution flowing into the well is calculated assuming a velocity at a representative radius from the well. The representative radius is taken to be:

$$R_r = \sqrt{R_w R_a} \qquad \text{[EQ 52.14]}$$

where

R_w well bore radius (taken from diameter input in COMPDAT)

R_a area equivalent radius of the grid block in which the well is completed.

In the present version of ECLIPSE, the radial inflow equation is not integrated over distance from the well to account for the local viscosity reduction due the local velocity.

Using the polymer flood model

The option is activated by the keyword POLYMER in the RUNSPEC section. If salt-sensitivity for polymer is required, this can be set by adding the keyword BRINE to the RUNSPEC section. By default the Polymer model is not salt-sensitive.

The mixing parameter data is obligatory and should be provided using the keyword TLMIXPAR. The maximum number of mixing parameter regions is set using the parameter NTMISC in the keyword MISCIBLE. The associated mixing region for each grid cell is specified using the MISCNUM keyword. The maximum polymer and salt concentrations to be used in calculating the effective fluid component viscosities are entered under the keyword PLYMAX.

The viscosity of a fully mixed polymer solution needs to be defined as function of the solution concentration. This should be done using the keyword PLYVISC. If the salt-sensitive option is activated (BRINE specified), then the keywords SALTNODE and PLYVISCS should be used to enter the two-dimensional table for the viscosity of the solution.

The polymer adsorption data should be entered using either the PLYADS keyword or the ADSORP keyword with POLYMER as argument in the PROPS section. When the ADSORP keyword is used, the polymer adsorption can be set to depend on the salinity and rock permeability. In this case there are two possibilities:

- With the standard brine option, the massic salt concentration is directly used in the analytic model, and therefore the units of the coefficients need to be interpreted accordingly.
- With the multi-component brine option (ECLMC keyword in the RUNSPEC section), the PLYESAL keyword needs to be added to trigger the computation of an effective salinity based on molar ions concentrations. This effective salinity is then used in the analytic model to compute the corresponding polymer adsorption. As for the previous approach, care needs to be taken in the interpretation of the units of coefficients in the analytical model.

Other polymer-rock parameters such as the rock mass density used in the adsorption calculation, the dead pore volume and the residual resistance factor are input using the keyword PLYROCK.

Non-Newtonian rheology can be activated either by setting the PLYSHEAR keyword in the PROPS section for Model 1, or by opting for the Herschel-Bulkley model as described in "Non-Newtonian fluid flows" on page 723.

If Model 1 is activated, the shear thinning data consists of tables of viscosity reduction as a function of local velocity. The values of $1/(\phi \cdot A)$ used to calculate the velocity can be printed out using the mnemonic POLYMER in the RPTGRID keyword. The representative radius used in the well inflow calculation is output with the well connection data.

Note however that if non-Newtonian rheology is not an important effect, it should be omitted as the calculation of the velocity dependent viscosity can significantly increase the CPU requirements for the run.

When the Brine option is activated in conjunction with Polymer, the initial salt concentrations are specified with either the SALT keyword (if the enumeration method is used for initialization) or the SALTVD keyword (if the EQUIL keyword is used). The initial polymer concentration and the initial adsorbed polymer concentration are both automatically set to zero.

If the model contains analytic aquifers, the initial salt concentration in each aquifer should be specified using the appropriate keywords AQUCT, AQUFET or AQUFETP. Since the aquifer volume is large, the polymer concentration is assumed to be always zero - any polymer entering the aquifer due to backflow is adsorbed by the aquifer rock.

If the standard brine is used, the polymer and salt well injection concentration should be set using the keyword WPOLYMER in the SCHEDULE section.

If the multi-component brine (ECLMC keyword in the RUNSPEC section) is used then injected polymer concentrations are still set using the WPOLYMER keyword (the second argument being ignored). The salt concentrations need to be set using the WSALT keyword.

Restrictions

The IMPES solution algorithm cannot currently be used with the Polymer option.

Wells are not allowed to crossflow when the Polymer Flood model is in use.

Changing the pore volume in the EDIT section using PORV is not allowed with the polymer flood model.

Summary of keywords

RUNSPEC section

The keywords are:

Table 52.1 RUNSPEC section

Keyword	Description
BRINE	Enables the Polymer Salt-sensitivity option.
MISCIBLE	Item 1 sets the maximum number of mixing parameter regions.
POLYMER	Enables the Polymer Flood model.

GRID section

The keywords are:

Table 52.2 GRID section

Keyword	Description
RPTGRID	Controls output from the GRID section. The POLYMER mnemonic outputs a map of the 1/(poro*area) values used for shear thinning calculations.

PROPS section

The keywords are:

Table 52.3 PROPS section

Keyword	Description
PLYADS	Polymer adsorption isotherms.
ADSORP	Analytical adsorption isotherms with salinity and permeability dependence.
PLYMAX	Polymer/salt concentrations for mixing calculations.
PLYROCK	Specifies the polymer-rock properties.
PLYSHEAR	Polymer shear thinning data.
PLYVISC	Polymer solution viscosity function.
PLYVISCS	Polymer/salt solution viscosity function.

Table 52.3 PROPS section

Keyword	Description
RPTPROPS	Controls output from the PROPS section. Mnemonics PLYVISC etc. output Polymer Flood model properties.
SALTNODE	Salt concentration nodes for polymer solution viscosity.
TLMIXPAR	Todd-Longstaff mixing parameter.

REGIONS section

The keywords are:

Table 52.4 REGIONS section

Keyword	Description
MISCNUM	Miscibility region numbers.
RPTREGS	Controls output from the REGIONS section. Mnemonic MISCNUM outputs mixing region numbers.

SOLUTION section

The keywords are:

Table 52.5 SOLUTION section

Keyword	Description
RPTSOL	Controls output from the SOLUTION section. Mnemonic - FIPPLY outputs polymer/salt fluid in place reports. - PBLK outputs initial grid block polymer concentration values. - PLYADS outputs the current adsorbed polymer concentration on the formation rock. - RK outputs the water relative permeability reduction factor. - SALT outputs the initial grid block salt concentration values. - SALTVD outputs the tables of initial salt concentration versus depth.
SALT	Initial salt concentrations.
SALTVD	Salt concentration versus depth for equilibration.

SUMMARY section

List of output for the Polymer Flood model:

Table 52.6 Polymer flood SUMMARY output controls

Field	Group	Well	Connection	Region	Block	Information
			CCFR			Polymer Flow Rate (+ or -)
FCPR	GCPR	WCPR				Polymer Production Rate
FCPT	GCPT	WCPT	CCPT			Polymer Production Total
FCIR	GCIR	WCIR				Polymer Injection Rate
FCIT	GCIT	WCIT	CCIT			Polymer Injection Total
					BCCN	Polymer Cell Concentration
FCIP				RCIP	BCIP	Polymer In Solution
				RCFT		Polymer Flow Total
FCAD				RCAD	BCAD	Polymer Adsorption total
			CSFR			Salt Flow Rate (+ or -)
FSPR	GSPR	WSPR				Salt Production Rate
FSPT	GSPT	WSPT	CSPT			Salt Production Total
FSIR	GSIR	WSIR				Salt Injection Rate
FSIT	GSIT	WSIT	CSIT			Salt Injection Total
					BSCN	Salt Cell Concentration
FSIP				RSIP	BSIP	Salt In Place
				RSFT		Salt Flow Total
					BEPVIS (BVPOLY)	Effective polymer solution viscosity [EQ 52.6]
					BEWVPL	Effective water viscosity [EQ 52.7]
					BEMVIS	Effective mixture viscosity [EQ 52.8]

SCHEDULE section

The keywords are:

Table 52.7 SCHEDULE section

Keyword	Description
RPTSCHED	Controls output from the SCHEDULE section. Mnemonic - FIPSALT outputs polymer/salt fluid in place reports. - PBLK outputs grid block polymer concentration values. - PLYADS outputs the current adsorbed polymer concentration on the formation rock. - RK outputs the water relative permeability reduction factor. - SALT outputs the grid block salt concentration values.
RPTRST	Mnemonic VISC allows to output extra viscosities when Polymer is active: - SHWATVIS: effective polymer viscosity due to shear thinning (PLYSHEAR keyword) - EPVIS: effective polymer viscosity (Polymer option) - EMVIS: effective mixture (water/polymer) viscosity (Polymer option) - EWV_POL: effective water viscosity (Polymer option) - EWV_SAL: effective water viscosity due to salt concentration (Brine option)
WPOLYMER	Sets polymer/salt concentrations for injection wells. The arguments can be used as User Defined Arguments (UDAs).

Example problems

Problem 1

Polymer flood model test problem.

This is a 1-D problem modeling water injection into oil at adverse mobility ratios. To aid recovery, a polymer slug is injected into the reservoir some time after the primary water drive. Water injection is resumed after the polymer injection period. The performance of the polymer drive is assumed not be salt sensitive in this study.

RUNSPEC

```
RUNSPEC  ============================================================
TITLE
Polymer flood model test problem - oil/water/polymer
DIMENS
 15    1    1  /
OIL
WATER
POLYMER
METRIC
TABDIMS
 1    1   50    2    2   20    1   20 /
REGDIMS
 2    1    0    0    0    1 /
WELLDIMS
 2    1    1    2 /
START
 1 'JAN' 1983  /
FMTOUT
FMTIN
```

GRID

```
GRID    ==============================================================
EQUALS
'DX'     100    /
'DY'     100    /
'DZ'     20     /
'PERMX'  100    /
'PERMY'  100    /
'TOPS'   3990.0 /
'PORO'   0.5    /
/
```

```
RPTGRID
 'PORO'  'PORV'  /
```

PROPS

```
PROPS   ============================================================
SWFN
    .20    .0    4.0
    .7    0.7    2.0
   1.0    1.0    0.0
 /
SOF2
    .3000  .0000
    .8000 1.0000
/
PVTW
  .0  1.0  3.03E-06  .5  0.0 /
PVDO
  .0     1.0     5.0
 8000.0   .92    5.0
/
ROCK
 4000.0    .30E-05 /
DENSITY
 52.0000  64.0000   .04400 /
```

Define the viscosity of the polymer solution as a function of the polymer concentration. Note that the function values refer to the multiplication factor for the pure water viscosity to obtain the true polymer solution viscosity.

```
PLYVISC
  0.0   1.0
  7.0  20.0 /
```

Define the polymer-rock properties. A series of entries is required for each rock region. These values represent the inaccessible pore volume, the residual resistance factor, the rock mass density, the adsorption index and the maximum polymer concentration, to be used in the calculation of the actual resistance factor.

```
PLYROCK
 0.15  2.67  1000.0  2  0.0035 /
```

Define the polymer adsorption properties. A table of values is required for each rock region. Each table represents the equilibrium concentration of polymer adsorbed on the rock and the concentration of polymer in the surrounding aqueous phase.

```
PLYADS
  0.0  0.000
  2.0  0.0015
  8.0  0.0025 /
```

Mixing parameter to indicate the degree of segregation between the polymer solution and the water at the trailing edge of the slug. In this case the polymer and water in each grid cell are fully mixed. A value of the mixing parameter should be entered for each mixing region.

```
TLMIXPAR
  1.0 /
```

Maximum value for the polymer and salt concentration values to be used in the mixing calculation of fluid viscosities. Since the salt option has been disabled in this run, the second value is ignored. A pair of entries is required for each mixing region.

```
PLYMAX
 5.0  0.0  /
```

The polymer properties can be echoed using keyword RPTPROPS.

```
RPTPROPS
 'PLYVISC'  /
```

REGIONS

```
REGIONS    ==========================================================
FIPNUM
1 1 1 1 1 1 1 2 2 2 2 2 2 2 2 /
```

SOLUTION

```
SOLUTION  ==============================================================
PRESSURE
 15*4000.0 /
SWAT
 15*0.2
```

Note that the mixing regions have not been specified in this run since the region numbers are all unity in this problem.

```
RPTSOL
 'PRES'  'SOIL'  'SWAT'  'RESTART=2'  'PBLK'
 'SALT'  'PLYADS'  'FIPPLY=2'  /
```

SUMMARY

```
SUMMARY
==========================================================
RUNSUM
```

SUMMARY file keywords are used to indicate which data are to be written to the ECLIPSE SUMMARY files at each time step. In this case, we are writing field water-cuts, polymer production rates, the cumulative polymer total and the cumulative oil production total at each time step.

```
FWCT
FCPR
FCPT
FOPT
RPTSMRY
 1 /
```

SCHEDULE

```
SCHEDULE ==========================================================
RPTSCHED
'PRES'  'SOIL'  'SWAT'  'RESTART=2'  'NEWTON=2'
'PBLK'  'SALT'  'PLYADS'  'RK'  /
WELSPECS
'I'  'G'   1  1  4000  'WAT'  0.0  'STD'  'SHUT'  'NO'  /
'P'  'G'  15  1  4000  'WAT'  0.0  'STD'  'SHUT'  'NO'  /
/
COMPDAT
'I       '   1   1   1   1 'OPEN'   0  .0   1.0 /
'P       '   15  1   1   1 'OPEN'   0  .0   1.0 /
/
WCONPROD
'P' 'OPEN' 'LRAT' 3* 1000.0 /
/
WCONINJ
'I' 'WAT' 'OPEN' 'RATE' 1000.0 /
/
```

Initial water flood

Specify polymer concentration in the injection stream of well ‘I’. The second figure refers to the injection salt concentration and is ignored in this run.

```
WPOLYMER
 'I'  0.0  0.0 /
 /
TSTEP
1.0 99.0 2*100.0  /
```

Polymer slug in fresh water

Specify the polymer concentration in the injection stream of well ‘I’.

```
WPOLYMER
 'I'  5.0  0.0   /
 /
TSTEP
 5*100.0 /
```

Chase water period

Specify the polymer concentration in the injection stream of well ‘I’.

```
WPOLYMER
 'I'  0.0  0.0   /
 /
TSTEP
10*100.0  /
END
```

Problem 2

Polymer flood model test problem.

This is a version of the previous example with the salt option activated to study the effect of the interaction between the injected polymer and the salt. The polymer is strongly salt-sensitive and the efficiency of the drive is expected to be lower than in the previous example.

RUNSPEC

```
RUNSPEC  ============================================================
TITLE
Polymer flood model test problem - oil/water/polymer/salt
DIMENS
 15    1    1  /
OIL
WATER
POLYMER
BRINE
METRIC
TABDIMS
 1    1   50    2    2   20    1   20 /
REGDIMS
 2    1    0    0    0    1 /
WELLDIMS
 2    1    1    2 /
START
 1 'JAN' 1983  /
FMTOUT
FMTIN
```

GRID

```
GRID    ============================================================
EQUALS
 'DX'      100     /
 'DY'      100     /
 'DZ'       20     /
 'PERMX'   100     /
 'PERMY'   100     /
 'TOPS'   3990.0   /
 'PORO'      0.5   /
/
RPTGRID
 'PORO'  'PORV'  /
```

PROPS

```
PROPS  ============================================================
SWFN
    .20    .0    4.0
    .7    0.7    2.0
   1.0    1.0    0.0
/
SOF2

    .3000  .0000
    .8000 1.0000
/
PVTW
  .0  1.0  3.03E-06  .5  0.0 /
PVDO
  .0     1.0     5.0
 8000.0   .92    5.0
/
ROCK
 4000.0    .30E-05 /
DENSITY
 52.0000  64.0000   .04400 /
```

Define the viscosity of the polymer solution as a function of both the polymer and the salt concentrations. A 2-dimensional table input structure is required. The salt concentration nodal values are defined using the SALTNODE keyword. The polymer concentration nodal values and the function values are entered using the PLYVISCS keyword. Note that the function values refer to the multiplication factor which is to be applied to the water viscosity to give the true solution viscosity.

```
SALTNODE
  0.0
  0.35      /
```

```
PLYVISCS
  0.0   1.0
        1.0 /
  7.0  20.0
        2.0 /
/
```

Define the polymer-rock properties. A series of entries is required for each rock region. These values represent the inaccessible pore space, the residual resistance factor, the rock mass density, the adsorption index and the maximum polymer concentration, to be used in the calculation of the actual resistance factor.

```
PLYROCK
  0.15  2.67  1000.0  2  0.0035 /
```

Define the polymer adsorption properties. A table of values is required for each rock region. Each table represents the equilibrium concentration of polymer adsorbed on the rock and the concentration of polymer in the surrounding aqueous phase.

```
PLYADS
   0.0  0.000
   2.0  0.0015
   8.0  0.0025 /
```

Mixing parameter to indicate the degree of segregation between the polymer solution and the water at the trailing edge of the slug. In this case, the polymer and water in each grid cell are fully mixed. A value of the mixing parameter should be entered for each mixing region.

```
TLMIXPAR
   1.0 /
```

Maximum value for the polymer and salt concentration values to be used in the mixing calculation of fluid viscosities. A pair of entries is required for each mixing region.

```
PLYMAX
  5.0  0.0  /
```

The polymer properties can be echoed using keyword RPTPROPS.

```
RPTPROPS
 'PLYVISC' /
```

REGIONS

```
REGIONS    ========================================================
FIPNUM
1 1 1 1 1 1 1 2 2 2 2 2 2 2 2 /
```

Define the mixing region for each grid cell

```
MISCNUM
 15*1 /
```

SOLUTION

```
SOLUTION  ==========================================================
PRESSURE
 15*4000.0 /
SWAT
 15*0.2  /
```

Define the initial salt concentration by enumeration.

```
SALT
 15*0.3  /
RPTSOL
'PRES'  'SOIL'  'SWAT'  'RESTART=2'  'PBLK'
'SALT'  'PLYADS'  'FIPPLY=2'  /
```

SUMMARY

```
SUMMARY  ===========================================================
RUNSUM
```

SUMMARY file keywords are used to indicate which data is to be written to the ECLIPSE summary files at each time step. In this case, we are writing field water-cuts, polymer production rates, salt production rates and cumulative oil production totals, at each time step.

```
FWCT
FCPR
FSPR
FOPT
RPTSMRY
 1 /
```

SCHEDULE

```
SCHEDULE  ==========================================================
RPTSCHED
'PRES'  'SOIL'  'SWAT'  'RESTART=2'  'NEWTON=2'
'PBLK'  'SALT'  'PLYADS'  'RK'  /
WELSPECS
'I'  'G'   1  1  4000  'WAT'  0.0  'STD'  'SHUT'  'NO'  /
'P'  'G'  15  1  4000  'WAT'  0.0  'STD'  'SHUT'  'NO'  /
/
COMPDAT
'I       '   1   1   1   1 'OPEN'   0  .0   1.0 /
'P       '   15  1   1   1 'OPEN'   0  .0   1.0 /
/
WCONPROD
'P' 'OPEN' 'LRAT' 3* 1000.0 /
/
WCONINJ
'I' 'WAT' 'OPEN' 'RATE' 1000.0 /
/
```

Initial water flood

Specify polymer and salt concentrations in the injection stream of well ‘I’.

```
WPOLYMER
 'I'  0.0  0.3 /
 /
TSTEP
1.0 99.0 2*100.0  /
```

Polymer slug in fresh water

Specify polymer and salt concentrations in the injection stream of well ‘I’.

```
WPOLYMER
 'I'  5.0  0.0   /
 /
TSTEP
 5*100.0 /
```

Chase water period

Specify polymer and salt concentrations in the injection stream of well ‘I’.

```
WPOLYMER
 'I'  0.0  0.3   /
 /
TSTEP
10*100.0  /
END
```

Potential Calculations

Chapter 53

Phase potentials

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

The phase potentials are the oil, water and gas phase pressures corrected to a datum depth. ECLIPSE can optionally calculate them for output. They may be written to the PRINT file and the SUMMARY file and, in ECLIPSE 100, the RESTART file.

The phase potential of a cell is calculated by correcting the appropriate phase pressure to the datum depth using the average phase density (at reservoir conditions) in the PVT region containing the cell,

$$\begin{aligned}\Phi_{\text{oil}} &= P_{\text{oil}} - \rho_o(D - \text{DATUM})\\ \Phi_{\text{wat}} &= P_{\text{wat}} - \rho_w(D - \text{DATUM})\\ \Phi_{\text{gas}} &= P_{\text{gas}} - \rho_g(D - \text{DATUM})\end{aligned} \qquad \text{[EQ 53.1]}$$

where

ρ_o = PVT region average oil density

ρ_w = PVT region average water density

ρ_g = PVT region average gas density

D = depth of the cell

DATUM = datum depth

The datum depth is either supplied using the DATUM keyword, or is taken to be the datum depth specified for the equilibration region 1, in the EQUIL keyword. If both the DATUM and EQUIL keywords are absent, the datum depth is set to zero.

The region and field average potentials are the phase pore volume weighted averages of the cell potentials within a region.

Hence, for a region or field, the average potential of phase p with saturation S_p is given by

$$\bar{\Phi}_p = \frac{\sum \Phi_p \cdot PV \cdot S_p}{\sum PV \cdot S_p} \qquad \text{[EQ 53.2]}$$

ECLIPSE 100

Setting `OPTIONS` keyword switch 31 non-zero and divisible by two, weights the potential by pore volume rather that phase pore volume.

The PVT region average density is always weighted by phase saturation, so that

$$\bar{\rho}_p = \frac{\sum PV \cdot S_p \cdot \rho_p}{\sum PV \cdot S_p} \qquad \text{[EQ 53.3]}$$

where the sum is over a PVT region, and PV is the reference pore volume. When the phase saturation is zero within a PVT region (for example, oil phase saturation for a gas condensate above the dew point pressure), a pore volume weight is used instead to obtain the average density which is only used to report what the potential would be if the phase occupied the PVT region.

Initial contact corrected potential (ECLIPSE 100)

ECLIPSE 100 also calculates a representative potential, where the pressure is corrected to the datum depth using the water density below the initial WOC, the gas density above the initial GOC, and the oil density between the two initial contacts.

Figure 53.1 Calculating the initial contact corrected potential

This representative potential can be thought of as the potential of the phase occupying the region at initial equilibrium conditions.

Output of potentials

The potentials can be output using the following keywords.

ECLIPSE 100

To the `PRINT` file:

```
RPTSOL
'POTO' 'POTW' 'POTG' 'POTC' /
RPTSCHED
'POTO' 'POTW' 'POTG' 'POTC' /
```

ECLIPSE 300

To the PRINT file:

```
RPTSOL
'OILPOT' 'GASPOT' 'WATPOT' /
RPTSCHED
'OILPOT' 'GASPOT' 'WATPOT' /
```

To the SUMMARY file:

FPPO	RPPO	BPPO	Oil Potential
FPPW	RPPW	BPPW	Water Potential
FPPG	RPPG	BPPG	Gas Potential

ECLIPSE 100

Again, for the SUMMARY file:

FPPC	RPPC	BPPC	Initial contact corrected potential

ECLIPSE 100

To the RESTART file:

```
RPTRST
'POT' /
```

Well potentials

Well potentials are flow rates rather than pressures. They represent the production or injection rate that a well would achieve in the absence of any rate constraints, at the current grid block conditions. The acting constraint for a well's potential is thus either its BHP limit or its THP limit, whichever is the more restrictive. If a well has a drawdown limit, that constraint may optionally be included in its potential calculation by setting the flag in item 4 of keyword WELDRAW.

ECLIPSE 100

There is an option to ignore THP limits when calculating the potential, which may be useful if the potentials are not used for anything important and the potential flows would extrapolate the VFP tables (see OPTIONS keyword switch 40).

Note that if a well has no VFP table and its BHP limit is defaulted, its potential will not be particularly meaningful.

The calculated well potential may be somewhat optimistic. This is because the grid block conditions at which the potential is calculated are not changed to take account of the well's higher flow rate. In reality, increasing a well's production from its actual rate to its potential would cause the pressure in its connecting grid blocks to decrease, thus decreasing its potential. The effect may be insignificant for wells completed in large grid blocks, but it could be appreciable for wells completed in small grid blocks, especially when using local grid refinement. The smaller the connecting grid blocks are, the more optimistic will be the potential.

On a similar note, the potential of a well belonging to a network (see "The Network Option" on page 681) may also be optimistic for another reason. The THP limit at which the well's potential is calculated is the same as for its normal flowing rate. In reality, increasing a well's production from its actual rate to its potential would cause its THP limit to increase due to the greater pressure losses in the network, thus decreasing its potential.

ECLIPSE uses well potentials in the following calculations and facilities:

- To set guide rates for group control, for wells that have not been given specific guide rates in keyword WGRUPCON. In ECLIPSE 100, the well and group guide rates can be set according to a function of their potentials (see keyword GUIDERAT).
- When using prioritization group control, well priorities can be set according to a function of their potentials (see keyword PRIORITY).

ECLIPSE 100

- Wells can be closed automatically when their potential production or injection rates fall below an economic limit (see keywords WECON and WECONINJ).

ECLIPSE 100

- A well may be opened from the drilling queue when required to maintain a group's production or injection potential above a specified value (see keyword GDRILPOT).
- In a Prioritized Drilling Queue, well drilling priorities can be set according to a function of their potentials (see keyword DRILPRI).

ECLIPSE 100

- Well potentials may be written to the PRINT file, in place of their PI, by setting a negative value for the drainage radius in item 7 of keyword WELSPECS. Well, connection and group potentials can also be written to the SUMMARY file, using the mnemonics:

FOPP	GOPP	WOPP	COPP	Oil Potential Production rate
FOPI	GOPI	WOPI	COPI	Oil Potential Injection rate
FWPP	GWPP	WWPP	CWPP	Water Potential Production rate
FWPI	GWPI	WWPI	CWPI	Water Potential Injection rate
FGPP	GGPP	WGPP	CGPP	Gas Potential Production rate
FGPI	GGPI	WGPI	CGPI	Gas Potential Injection rate

ECLIPSE 300

Well potentials can be written to the SUMMARY file, using the mnemonics

WOPP	Oil Potential Production rate
WWPP	Water Potential Production rate
WWIP	Water Potential Injection rate
WGPP	Gas Potential Production rate
WGIP	Gas Potential Injection rate

The field and group potentials are the sum of their subordinate wells' potentials. The connection potentials are the individual connection flow rates when the well is operating at its potential.

Recovery Mechanisms

Chapter 54

Introduction

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL

The oil production from a reservoir will typically be due to a combination of different recovery mechanisms, for example rock compaction and water influx. ECLIPSE 100 provides a facility to quantify the proportion of oil produced by each physical process, accumulated during the simulation. This information is important both to understand the behavior of the reservoir and to allow greater investigation of the data uncertainty associated with each production mechanism.

The oil in place within a grid block is:

$$\text{Total oil in place} = \text{PV} \cdot S_o \cdot \frac{1}{B_o} \qquad \text{[EQ 54.1]}$$

$$S_o = 1 - S_w - S_g$$

where

PV is the pore volume

S_o is the oil saturation

B_o is the oil formation volume factor

S_w is the water saturation

S_g is the gas saturation

The change in oil in place can be attributed as follows:

$$
\begin{aligned}
\mathrm{DC} &= -d(\mathrm{PV}) \cdot S_o \frac{1}{B_o} \\
\mathrm{DW} &= \mathrm{PV} \cdot d(S_w) \frac{1}{B_o} \\
\mathrm{DG} &= \mathrm{PV} \cdot d(S_g) \frac{1}{B_o} \\
\mathrm{DE} &= \mathrm{PV} \cdot S_o \cdot d\left(\frac{1}{B_o}\right)
\end{aligned}
\qquad \text{[EQ 54.2]}
$$

where

DC is the production due to rock compaction

DW is the production due to water influx

DG is the production due to gas influx

DE is the production due to oil expansion

$d(\mathrm{PV})$ is the change in pore volume over the time step

$d(S_w)$ is the change in water saturation over the time step

$d(S_g)$ is the change in gas saturation over the time step

$d(1/B_o)$ is the change in formation volume factor over the time step.

In the case of a saturated cell reducing in pressure, the oil expansion term, DE, will be negative as $1/B_o$ will increase. Hence if a cell is saturated then the (negative) production is accumulated with the gas influx:

If a cell is saturated then:

$$
\begin{aligned}
\mathrm{DG} &= \mathrm{DG} - \mathrm{DE} \\
\mathrm{DE} &= 0
\end{aligned}
\qquad \text{[EQ 54.3]}
$$

The oil production from each mechanism (DC, DW, DG, DE) is summed in each fluid-in-place region and also summed over time steps. A table of production by each recovery mechanism can be output by using the `RECOV` mnemonic in the `RPTSCHED` keyword, and appears under the heading of `RECOVERY`.

Limitations

At present the facility cannot be used in runs containing vaporized oil, (that is, with `VAPOIL` present in the `RUNSPEC` section).

Solution gas drive

Dividing the oil production between the above set of recovery mechanisms will give the total oil production due to gas influx, but this does not differentiate between the oil production associated with free gas influx and that associated with solution gas.

The oil production due to gas influx can be sub-divided into oil produced by solution gas drive and oil produced by free gas, by using the Tracer Tracking facility. If a gas tracer is defined and initialized to be 1.0 for gas dissolved in the oil, and 0.0 for free gas, then the recovery mechanism option will use the tracer information to calculate the split between solution gas and free gas.

To use this facility the following steps must be taken:

1 Increase the number of gas tracers by one in the TRACERS keyword in the RUNSPEC section.

2 Add a tracer called 'DGS' in the PROPS Section. The special name 'DGS' **must** be used to enable the recovery mechanism to recognize the tracer.

```
TRACER
 'DGS' 'GAS' /
/
```

3 Initialize the tracer in the SOLUTION section, giving the solution gas a value of 1.0 and the free gas a value of 0.0, for example:

```
TBLKFDGS
 300*0 /
TBLKSDGS
 300*1 /
```

Two new oil production values associated with solution gas and free gas will be calculated as follows:

$$\mathrm{DS} = \mathrm{PV} \cdot F_s \cdot d(S_g) \cdot \frac{1}{B_o}$$
$$\mathrm{DF} = \mathrm{PV} \cdot (1 - F_s) \cdot d(S_g) \cdot \frac{1}{B_o} \qquad \text{[EQ 54.4]}$$

where

DS is the production due to solution gas

DF is the production due to free gas

F_s is the fraction of S_g that was originally solution gas.

Water drive

A similar procedure is available to distinguish production from two different water sources, for example injected water and aquifer water. Again, a tracer is used to track (say) the aquifer water, and the production is sub-divided into production associated with 'traced' water and production associated with the remaining water.

To use this facility the following steps must be taken:

1 Increase the number of water tracers by one in the TRACERS keyword in the RUNSPEC section.

2 Add a tracer called 'WTR' in the PROPS Section. The special name 'WTR' **must** be used to enable the recovery mechanism to recognize the tracer.

```
TRACER
 'WTR' 'WAT' /
/
```

3 Initialize the tracer in the SOLUTION section in the usual way using the TBLK or TVDP keywords, and set the tracer concentration for any analytic aquifers using the AQANTRC keyword.

   The initial tracer concentration should be either 1.0 for 'traced' water, or 0.0 for the rest.

Two new oil production values associated with the traced water and the remaining water will be calculated as follows:

$$\begin{aligned} \mathrm{DWT} &= \mathrm{PV} \cdot F_t \cdot d(S_w) \cdot \frac{1}{B_o} \\ \mathrm{DWR} &= \mathrm{PV} \cdot (1 - F_t) \cdot d(S_w) \cdot \frac{1}{B_o} \end{aligned} \qquad \text{[EQ 54.5]}$$

where

DWT is the production due to traced water

DWR is the production due to remaining water

F_t is the fraction of S_w that is 'traced' water.

Summary output

The oil production attributable to each mechanism, and the corresponding fraction of the total oil production, can also be output to the `SUMMARY` file using the following mnemonics

Table 54.1 Mnemonics for reporting recovery mechanism SUMMARY data

Field	Region	Description
FORMR	RORMR	Stock tank oil produced by rock compaction
FORMW	RORMW	Total stock tank oil produced by water influx
FORMG	RORMG	Total stock tank oil produced by gas influx
FORME	RORME	Stock tank oil produced by oil expansion
FORMS	RORMS	Stock tank oil produced by solution gas
FORMF	RORMF	Stock tank oil produced by free gas influx
FORMX	RORMX	Stock tank oil produced by ‘traced’ water influx
FORMY	RORMY	Stock tank oil produced by other water influx
FORFR	RORFR	Fraction of total oil produced by rock compaction
FORFW	RORFW	Fraction of total oil produced by water influx
FORFG	RORFG	Fraction of total oil produced by gas influx
FORFE	RORFE	Fraction of total oil produced by oil expansion
FORFS	RORFS	Fraction of total oil produced by solution gas
FORFF	RORFF	Fraction of total oil produced by free gas influx
FORFX	RORFX	Fraction of total oil produced by ‘traced’ water influx
FORFY	RORFY	Fraction of total oil produced by other water influx

Details of the recovery calculation

ECLIPSE solves the oil conservation equation:

$$d(\mathrm{OIP}) = \mathrm{PV}^{t+dt} \cdot S_o^{t+dt} \cdot \left[\frac{1}{B_o}\right]^{t+dt} - \mathrm{PV}^t \cdot S_o^t \cdot \left[\frac{1}{B_o}\right]^t \qquad \text{[EQ 54.6]}$$

Some care is needed to associate the production mechanisms such that the total oil produced equals the change in oil-in-place, $d(\mathrm{OIP})$. Below we show how the production associated with each recovery mechanism is calculated.

Let:

$$\begin{aligned}
\mathrm{PV}^{t+dt} &= \mathrm{PV}^t + d(\mathrm{PV}) \\
S_o^{t+dt} &= S_o^t + d(S_o) \\
\left[\frac{1}{B_o}\right]^{t+dt} &= \left[\frac{1}{B_o}\right]^t + d\left[\frac{1}{B_o}\right] \\
d(\mathrm{OIP}) &= [\mathrm{PV} + d(\mathrm{PV})][S_o + d(S_o)]\left[\frac{1}{B_o} + d\left(\frac{1}{B_o}\right)\right] - \left[\mathrm{PV} \cdot S_o \cdot \frac{1}{B_o}\right]
\end{aligned} \qquad \text{[EQ 54.7]}$$

Expanding the conservation equation,

$$\begin{aligned}
d(\mathrm{OIP}) = {} & d(\mathrm{PV}) \cdot S_o^t \cdot \left[\frac{1}{B_o}\right]^t + d(\mathrm{PV}) \cdot S_o^t \cdot d\left[\frac{1}{B_o}\right] \\
& + \mathrm{PV}^t \cdot d(S_o) \cdot \left[\frac{1}{B_o}\right]^t + d(\mathrm{PV}) \cdot d(S_o) \cdot \left[\frac{1}{B_o}\right]^t + \mathrm{PV}^t \cdot S_o^t \cdot d\left[\frac{1}{B_o}\right] \\
& + \mathrm{PV}^t \cdot d(S_o) \cdot d\left[\frac{1}{B_o}\right] + d(\mathrm{PV}) \cdot d(S_o) \cdot d\left[\frac{1}{B_o}\right]
\end{aligned} \qquad \text{[EQ 54.8]}$$

The terms in this equation that are associated with the various production mechanisms are:

$$\begin{aligned}
\text{Rock Compaction} &= \mathrm{DC} = -d(\mathrm{PV}) \cdot S_o^t \cdot \left[\frac{1}{B_o}\right]^{t+dt} \\
\text{Saturation Change} &= \mathrm{DS} = -\mathrm{PV}^{t+dt} \cdot d(S_o) \cdot \left[\frac{1}{B_o}\right]^t \\
\text{Oil Expansion} &= \mathrm{DE} = -\mathrm{PV}^t \cdot S_o^{t+dt} \cdot d\left[\frac{1}{B_o}\right] - d(\mathrm{PV}) \cdot d(S_o) \cdot d\left[\frac{1}{B_o}\right]
\end{aligned} \qquad \text{[EQ 54.9]}$$

Now, $d(S_o) = -d(S_w) - d(S_g)$, giving the final breakdown of the production:

$$\begin{aligned}
\text{Rock Compaction} &= \mathrm{DC} = -d(\mathrm{PV}) \cdot S_o^t \cdot \left[\frac{1}{B_o}\right]^{t+dt} \\
\text{Water Influx} &= \mathrm{DW} = \mathrm{PV}^{t+dt} \cdot d(S_w) \cdot \left[\frac{1}{B_o}\right]^t \\
\text{Gas Influx} &= \mathrm{DG} = \mathrm{PV}^{t+dt} \cdot d(S_g) \cdot \left[\frac{1}{B_o}\right]^t \\
\text{Oil Expansion} &= \mathrm{DE} = -\mathrm{PV}^t \cdot S_o^{t+dt} \cdot d\left[\frac{1}{B_o}\right] - d(\mathrm{PV}) \cdot d(S_o) \cdot d\left[\frac{1}{B_o}\right]
\end{aligned} \qquad \text{[EQ 54.10]}$$

When the pressure falls below the bubble point the oil expansion term will usually be negative, as the oil production will then be by solution gas drive. In this case the oil expansion term (DE) is added to the gas influx term (DG):

If gas saturation is greater than zero then:

$$\begin{aligned}
\mathrm{DG} &= \mathrm{DG} + \mathrm{DE} \\
\mathrm{DE} &= 0.0
\end{aligned} \qquad \text{[EQ 54.11]}$$

Reservoir Coupling

Chapter 55

Introduction

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL

The Reservoir Coupling facility is a Special Extension to ECLIPSE 100; it is not available in ECLIPSE 300. It provides a means of coupling a number of separate ECLIPSE 100 simulation models to account for constraints on overall production and injection rates, and optionally the sharing of a common surface network.

The individual simulation models are run as separate processes, each having its own standard ECLIPSE data file. One model is chosen to be the “master”, and the other models are defined as “slaves”. The slave processes are activated by the master, and run in synchronization with it. The master process imposes production and injection constraints on the slaves to meet overall targets. On suitable systems, the slave processes may be run in parallel with each other on separate workstations. Communication between the processes is handled by the message-passing system MPI, which must be present on the system.

A typical situation in which the Reservoir Coupling facility would be useful is as follows. Imagine you are responsible for an area that contains a number of separate independent reservoirs. For each reservoir, you have a separate simulation model, which you have used to history match the reservoirs independently. The simulation models may have different characteristics. For example, some may be 3-phase while others are 2-phase. One model may perhaps use the Vertical Equilibrium option, while the others use dispersed flow. There are now plans to produce these reservoirs into common surface facilities, and constraints on overall production and injection rates must be observed. Also, perhaps, the Network model is to be used to determine the pressure constraints in a common surface network linking the reservoirs.

Without the Reservoir Coupling facility, the above scenario would require the various simulation grids to be amalgamated into a single, huge model. It would have to be 3-phase if each phase was active in one or more of the reservoir models, and if one of them used the V.E. option then the whole amalgamated model would also have to use it. The task of amalgamating the models would be time consuming, and the resulting single model would be slow and expensive to run. The Reservoir Coupling facility offers an efficient solution to this problem, allowing the reservoirs to be represented by their original simulation models, but these are now coupled through their production and injection targets and network nodal pressures.

Master and slave reservoirs

Each slave reservoir consists of an ECLIPSE 100 simulation model with its own input data file. They may be restarts, if required, of simulations that have previously been run independently. Each must have one or more slave groups, which act as the path for communicating flow rates and constraints with the master process. For example, a reservoir may have its FIELD group as a single slave group, or alternatively all its Level 1 groups may be slave groups. The slave groups need not all be on the same level, but one slave group cannot be subordinate to another slave group.

The master reservoir is an additional ECLIPSE 100 simulation model, again with its own input data file. It must contain the group hierarchy of the whole system down as far as the slave groups in the slave reservoirs. The slave groups are represented in the master reservoir by master groups, which have no subordinate wells or groups. These are similar to satellite groups in concept (see keyword `GSATPROD`), except that the flows of satellite groups are set by the user, whereas the flows of master groups are determined by the coupled slave reservoir simulations.

The master reservoir may represent one of the reservoirs to be simulated. If so, its grid data, fluid properties, scheduling data, etc. will be entered in the normal way in its data file. Alternatively, all the reservoirs to be simulated may be represented by slave reservoirs. In this case, the master reservoir should just be a dummy reservoir for example a single grid block), containing no wells but having the group hierarchy down to the master groups.

Figure 55.1 Three coupled reservoirs; one is the master

Figure 55.2 Three reservoirs coupled by a “dummy reservoir” master

Figure 55.1 and Figure 55.2 illustrate these two cases applied to three coupled reservoirs. In Figure 55.1, one of these reservoirs R1 is chosen as the master. Its own wells are under groups R1-A and R1-B. The other two reservoirs are represented as slaves. The master reservoir controls the slave reservoirs through the groups R2, R3-A and R3-B. These are declared as master groups in the master reservoir, and do not have any subordinate wells. The master group R2 is coupled to the FIELD group in the slave reservoir R2. The master groups R3-A and R3-B are coupled to the level 1 groups of the same name in the slave reservoir R3. In the slave reservoir R2, the FIELD group is declared as a slave group, coupled to group R2 in the master reservoir. In the slave reservoir R3, the groups R3-A and R3-B are declared as slave groups, coupled to the groups of the same name in the master reservoir. This example shows that a given slave reservoir may be coupled to the master by more than one group, which need not necessarily belong to the same parent group in the master reservoir.

Figure 55.2 shows the same set of reservoirs, but this time all three of them are represented as slave reservoirs. The master reservoir is a dummy reservoir with no wells. It contains the group hierarchy down to the master groups. Note that groups R1-A and R1-B are now master groups, being coupled to slave groups of the same name in reservoir R1 which is now a slave reservoir.

Coupling and synchronization

At the start of the master run, the master enquires the start date of each slave run. A slave may, if necessary, be set to start later than the master. In this case the slave will be “dormant”, not contributing any production or injection, until its start date is reached in the master run.

The events that couple the master and the slave runs are synchronized with the time steps of the master run. Note that these are the **actual** time steps, rather than the user-defined **reporting** steps. Thus the coupling events can be made to occur more frequently (for greater accuracy) by reducing the maximum time step size in the master reservoir (keyword `TUNING`).

For each time step in the master run:

1. The master must first decide the length of the time step. This is done in the usual way, but subject to two additional constraints:
   a. The time step is limited so as not to overshoot a reporting time in any of the slave runs. (Changes to wells may be scheduled to occur at these times.) The master asks each of the slaves when its next report time is. Time steps limited in this way are identified by the time step control mnemonic `SLVR` in the time step summary written to the Print file.
   b. The time step is limited to keep the expected change in flow rate from the slave groups occurring over its length to within certain bounds. This limit is estimated from the changes that occurred over the previous time step. (See keywords `GRUPMAST` and `RCMASTS`.) Time steps limited in this way are identified by the time step control mnemonic `SLVC` in the time step summary written to the Print file.
2. At the beginning of the time step, the master enquires from the slaves what their slave groups' production and injection potentials are. These are determined from the reservoir conditions at the beginning of the time step. They may be used to calculate the guide rates of the master groups, for apportioning the group flow targets.
3. The master allocates rate targets to all its master groups, based on their guide rates. This is done according to the usual procedure for guide rate group control. When a master group is given a rate target, it sends it to its equivalent slave group in the slave reservoir. The slave run calculates the flows of the group under this constraint, at the conditions prevailing at the beginning of the time step, and tells the master.
4. The master completes its own time step, assuming that the production and injection rates of the slave groups remain constant over the time step.
5. When the master has completed its time step, all the slave reservoirs are advanced to the end of the master's time step, using the rate targets allocated to the slave groups as constraints. The slaves may take one or more of their own time steps to reach this time. The time step that is limited to reach this time is identified by the time step control mnemonic `MAST` in the time step summary written to the Print file. `RESTART` and `SUMMARY` files are written at the appropriate times by each slave process, as they would be if the slaves were being run independently. The slave simulation processes can be run in parallel with each other, if they are located on different workstations. Note that events occurring in a slave reservoir cannot influence the master or the other slaves until the next master time step.
6. When the slave reservoirs have advanced to the new time, they send the new cumulative production and injection totals of the slave groups to the master. From these the master computes the average flow rates of each master group over its time step. The master sums these flows up to its FIELD level, and updates its cumulative totals. The master writes its own `SUMMARY` and `RESTART` files as required, at the end of its time step.

This procedure continues until the master run finishes. Its final task is to send a signal to the slave runs to terminate. If a slave run finishes before the master run, the master run will continue without any production or injection from the slave. However, the master run can be made to stop in these circumstances by setting the "end run" flag of its master groups in item 8 of keyword `GECON`. Of course, if a slave run stops due to an error condition, the master and all the other slave runs will also stop.

Group control

The reservoirs are coupled by allocating production and injection rate targets to the slave groups, to meet global production and injection targets specified in the master reservoir. The allocation is made in proportion to the guide rates of the master groups, following the normal procedure of guide rate group control. Thus the master groups must be given guide rates in the master reservoir's data file for production control and any injected phases. The group guide rates may optionally be set as a function of their potential rates (for production), or equal to their voidage replacement needs (for injection).

In each slave reservoir, the target rates of the slave groups are communicated from the master run. These targets can be allocated to subordinate wells or groups by either guide rate control or prioritization. Additional constraints may be applied to subordinate groups in the usual way. However, any groups in the slave reservoir that are superior to the slave groups must not have any constraints on them, as these would interfere with the targets of the slave groups.

If the slave reservoirs were previously simulated independently, they may already have rate targets for the slave groups set in their data files. Accordingly, an option is provided either to ignore any limits specified for the slave groups in their data files, or to take the minimum of these limits and the targets set by the master. The option is independently selectable for each produced and injected phase.

If a rate constraint is applied to a phase that is not an active phase in the slave run, ECLIPSE transforms it into a constraint on an associated phase, using the latest available production ratios. This may happen, for example, if gas is an active phase in the master run, but not in the slave run, which instead has a constant Rs specified with keyword `RSCONST` or `RSCONSTT`. If the master imposes a gas rate target on a slave group in this reservoir, it is transformed into an oil rate target using the previously calculated GOR of the group.

Overall rate targets applied to the coupled system in the master run will not be met exactly. This is because the master run apportions the rate targets among the master groups according to their slave groups' flow capabilities at the **start** of the time step, but the flows are reported at the **end** of the time step. If the flow rate of a slave group changes over the length of the master's time step, it causes the flow totals in the master reservoir to drift away from their targets. Obviously, the amount of drift increases with the length of the master run's time steps. There is an option to limit the master run's time steps to keep the expected drift within a specified tolerance (see item 4 of keyword `GRUPMAST`). The drift also occurs with reinjection and voidage replacement targets applied in the master run. If you wish to impose reinjection or voidage replacement targets on a slave reservoir, they will be met more accurately if they are imposed directly on the slave groups in the slave reservoir's data file than if they are imposed in the master reservoir.

Flowlines and surface networks

If the coupled reservoirs produce into the same flowline or surface network, the Reservoir Coupling facility can be used in conjunction with the Network option to include the network conditions in the constraints that couple the reservoirs (see "The Network Option" on page 681). Specifically, the networks in each of the reservoirs are coupled by including the **nodal pressures** of the master groups in the set of constraints passed by the master to the slave reservoirs.

The global network should be constructed as follows:

1 The master reservoir should contain a representation of the global network down to the master groups (and its own well groups also, if the master represents an actual reservoir).

2 In each slave reservoir, the slave groups should be defined as fixed pressure nodes, and the network extended down from them to the well groups. Any groups superior to the slave groups should not be part of the network in the slave reservoirs.

The master and each slave may use either the Standard Network model or the Extended Network model, which allows the network structure to differ from the GRUPTREE hierarchy (see "The extended network model" on page 693). The Extended Network structure and the GRUPTREE hierarchy must however coincide at the master groups (in the master reservoir) and the slave groups (in the slave reservoir). Specifically, the requirements for the Extended Network structure are:

1 If the master reservoir uses the Extended Network model, the master groups must coincide with source nodes in the network. For each master group there must be a corresponding network source node with the same name.

2 If a slave reservoir uses the Extended Network model, the slave groups must coincide with fixed-pressure nodes in the network. For each slave group there must be a corresponding network fixed-pressure node with the same name.

The networks in each reservoir should be set up to be balanced at each time step (which is the default in keyword NETBALAN). Thus all networks will be balanced simultaneously at each time step of the master process. At each iteration of the network balancing calculation in the master:

1 The master sends the nodal pressures of the master groups to the respective slave groups in the slave reservoirs.

2 The slave reservoirs balance their own networks, using the nodal pressures passed from the master as the “fixed” pressures of the slave groups.

3 The slave processes send back to the master the flows of the slave groups at the current conditions.

4 The master sums the flows up to its FIELD level, and updates the pressures in the nodes of its network in the usual way.

Note that at each balancing iteration in the master run, the slaves balance their own networks to convergence.

If the slave reservoirs take more than one time step to advance to the end of the master time step, they will rebalance their own networks at each step. However, the “fixed” pressures of the slave groups will remain constant until they are updated again at the next master time step.

Reservoir coupling files

During the simulation of the coupled system, the master run can be made to write the flow rates and coupling constraints of the master groups to a **reservoir coupling file**. At each time step of the master run, the following information is written to this file for each master group:

1 The group’s coupling constraints. These include the rate target applied to the group by the master run. Other rate limits are included, to prevent the group increasing its flow sufficiently to violate any rate limits imposed on higher level groups, assuming the flows of all other groups remain constant. If the group is part of a pipeline network, its network nodal pressure is also included.

2 The group’s potential flow rates. These are the flow rates that the coupled slave group’s subordinate wells would achieve, at the current grid block conditions, if all rate limits were ignored and only their THP and BHP limits were observed. The potentials may be used to determine guide rates for group control.

3 The group's production and injection rates. These are the rates that the coupled slave groups have achieved, averaged over the length of the master run's current time step. These rates are reported in the master run.

The Reservoir Coupling file will be written by the master run if the keyword DUMPCUPL is present in its data file. There is a choice to make this either a formatted or an unformatted file. Formatted files occupy much more disk space than unformatted files, but should be chosen if the Reservoir Coupling file is going to be used on a system that does not have binary compatibility with the one on which it was created.

The Reservoir Coupling file may subsequently be used for either of two purposes:

1 To substitute for one or more of the slave reservoirs in the master run. The flow rates of the slave groups will be read from the file and incorporated into the master groups' flows.

   This would allow the master to be run with a reduced number of coupled slaves. In fact, if all the slaves were replaced by the data on the Reservoir Coupling file, the master could be run on its own without requiring MPI to activate the slave runs.

   Obviously, the slave group flows read from the file cannot be influenced by any changes made to the coupled system. So be careful about making changes that would have significant repercussions on the slave reservoirs. Use of the Reservoir Coupling file for this purpose has similarities with the Flux Boundary Conditions option.

2 To drive a slave reservoir simulation on its own, supplying the rate (and perhaps network pressure) constraints that were originally dictated by the master. MPI is not required for this task.

   At a time corresponding to the start of each time step of the original master run, the rate (and network pressure) constraints of the reservoir's slave groups are read from the file. These are applied to the groups until the next master time step. For additional flexibility, the user can choose to ignore limits for particular produced or injected phases. For each produced and injected phase, there is an option to take the rate constraints as specified in the slave's data file, or to take the rate constraints from the Reservoir Coupling file, or to take the minimum of the two constraints. (See keyword GRUPSLAV.)

   A set of SUMMARY section keywords is provided to output the rate limits the master groups are subject to, as a function of time. This offers a useful way of displaying the rate limits imposed by the Reservoir Coupling file.

When a Reservoir Coupling file is used for either of the two purposes described above, the time steps of the simulation are limited so as not to overshoot the time steps of the original master run that produced the file. At these times the flow rates and the constraints on the file may change. Time steps limited in this way are identified by the time step control mnemonic MAST in the time step summary written to the Print file.

If the original coupled runs were performed over a series of restarts, there should be a sequence of Reservoir Coupling files, each having the root name of the data file of the master run that wrote it. Each file has a header that contains the root name of the data file from which it was restarted. Thus ECLIPSE is able to establish the chain of Reservoir Coupling files to be used, provided that they are all present.

Current restrictions

1. The global constraints that couple the reservoirs are limited to group rate control (using group guide rates) and network pressures. Other features (for example prioritization, economic limits, automatic drilling and workover facilities, group production rules) can be used within individual reservoirs, but cannot be employed globally. The master run does not have information about the individual wells in the slave reservoirs, and cannot directly command drilling, workover, shut-in or production rule actions (`PRORDER`) on these wells. In particular, `GECON` should not be used to apply economic limits to master groups or their superiors, and no procedure other than RATE should be specified for these groups in `GCONPROD`. Prioritization (keyword `GCONPRI`) cannot be applied to these groups.
2. Global rate targets and limits will not be met exactly, due to the flows of the slave groups changing over the length of the master time step.
3. It is not possible to guarantee that all current and future ECLIPSE facilities may be used in coupled reservoirs. In particular, Gas Lift Optimization cannot be used at present (see "Gas Lift Optimization" on page 385). However, individual reservoirs may use Local Grid Refinement, Vertical Equilibrium, the Flux option, tracers, and polymer or surfactant floods. The Gas Field Operations model may be used to a limited extent with Reservoir Coupling (see "Using the reservoir coupling facility" on page 819).
4. The master time step cannot be influenced by events happening in the slave reservoirs, other than observing their report times. For example, automatic drilling or workovers happening in a slave reservoir will not cause the master to chop or repeat its time step.
5. The master run and all the slave runs must have the same unit convention. The slave runs need not all have the same set of active phases, but all phases active in any slave must be active in the master.
6. The order of solving the different types of wells (for example producers, water injectors, gas injectors) must be the same in all the coupled reservoirs. Thus, all reservoirs must have the same top-up phase for voidage replacement or reservoir volume rate injection targets, as these injectors are solved last. For example, one reservoir cannot have water voidage replacement targets if another has gas voidage replacement targets. Similarly, either none or all of them must use production balancing, which requires the producers to be solved after the injectors.
7. Automatic compressors in an Extended Network (see keyword `NETCOMPA`) cannot respond to falling production from a group in a different reservoir. For example, compressors in a slave reservoir cannot be activated by a group in the master reservoir failing to meet its `GCONPROD` target.

Using the reservoir coupling facility

The Reservoir Coupling facility uses the MPI message passing system to send messages and data between the master and the slave runs.

Reservoir coupling using MPI

To run a coupled reservoir simulation MPI must first be installed on your system. Refer to the "ECLIPSE Suite Installation Guide" for instructions on how to install MPI.

By using MPI as the message passing system slave reservoirs may be run either as serial processes or in parallel. To run a slave reservoir in parallel the number of processors on which it is to be run must be specified in the master data set using the SLAVES keyword. The PARALLEL keyword should also appear in the slave data set with the number of processors on which the slave is to be run as the first argument.

It is not possible to run a master reservoir in parallel.

Starting a reservoir coupling simulation using MPI is achieved using the parallel ECLIPSE macro, @mpieclipse. The total number of processors to be used by the master reservoir and combined slave reservoirs should be provided as the argument to @mpieclipse.

for example, @mpieclipse -procs X

The master reservoir

The master reservoir is represented by a standard ECLIPSE data file. It may, if required, be a restart from a previous run. It must have as active phases all the phases that are active in any of the slave runs. It must contain the group hierarchy down to the master groups that couple it to the slaves, and of course the hierarchy must include its own well groups if it has any. The group hierarchy is defined in the usual way with keyword GRUPTREE.

The master does not necessarily have to represent a real reservoir. It may just be a “dummy reservoir” whose purpose is to couple the slave reservoirs through its production and injection constraints (and optionally a pipeline network). In this case, the data file should still contain enough information to run ECLIPSE. At least one grid block is necessary. Fluid properties should be provided as required for its active phases. No wells should be defined, but the group hierarchy must be defined down to the master groups.

The SLAVES keyword must be used in the SCHEDULE section to activate the coupled slave runs. The keyword data specifies the machine on which each slave process is to be run, and the directory where their data files are located. For example, for Figure 55.1:

```
SLAVES
--    slave   datafile    machine        directory
--    name    root        hostname       of data file
    'RES-R2'  'base'      'rios'        '/usr/models/res2' /
    'RES-R3'  'base'      'sg-indigo'   '/usr/models/res3' /
/
```

If you wish the master run to create a Reservoir Coupling file containing the rates and constraints of all the master groups, enter the keyword DUMPCUPL:

```
DUMPCUPL
  'U'  /
```

After the group hierarchy has been set up with keyword GRUPTREE, the master groups must be identified with keyword GRUPMAST. The keyword also specifies the slaves to which the master groups are coupled, and sets a limiting fraction for flow rate change in a single time step. For example, for Figure 55.1:

```
GRUPMAST
--  master      slave       associated      limiting
--  group     reservoir       slave        rate change
--  name        name          group         fraction
   'R2'       'RES-R2'       'FIELD'           0.1  /
   'R3-A'     'RES-R3'         1*              0.1  /
   'R3-B'     'RES-R3'         1*              0.1  /
/
```

If limiting rate change fractions have been set for the master groups, it is advisable to set a minimum length to which the time step can be reduced by this mechanism, using keyword RCMASTS:

```
RCMASTS
 10.0  /
```

Set the global production rate targets and limits with keyword GCONPROD, with action 'RATE' selected to activate group control if limits are exceeded. Other actions, for example 'WELL' or 'CON' will not work with master groups, because the master run cannot request workovers and shut-ins for wells in the slaves. For similar reasons, prioritization group control (keyword GCONPRI) cannot be used for master groups and any of their superior groups. The master groups must have their production guide rates set, so that higher level rate targets can be apportioned among these groups (rather than their wells). The group guide rates can either be set to a fixed value, or calculated as a function of their production potentials (see keyword GCONPROD items 9 and 10).

```
GCONPROD
-- group   ctl   oil   wat   gas   liq   limit   field  guide  g-r
-- name    mode  rate  rate  rate  rate  action  ctrl?  rate   phase
  'FIELD' 'ORAT' 4.E5  /
  'PLAT*' 'NONE' 3*                3.E5  'RATE' /
  'R1-*'  'FLD'  6*                                    100   'LIQ' /
  'R3-*'  'FLD'  6*                                    100   'LIQ' /
  'R2'    'FLD'  6*                                    200   'LIQ' /
/
```

If there are any global injection targets or limits to be applied to the coupled system, set them with keyword GCONINJE in the usual way. Any master group subordinate to a group with an injection target or limit must be given an injection guide rate for that phase, so that the higher level target can be apportioned down to it. The guide rate can optionally be set equal to the group's voidage replacement requirements (GCONINJE item 10). If reinjection or voidage replacement is to be performed independently in each slave, and there are no global injection constraints, the targets will be met more accurately if they are specified in the slave data files rather than the master, because of the "drift" caused by flow rate changes over the master time step.

```
GCONINJE
-- group  inj    ctl   surf  resv  reinj  vrep  field  guide  g-r
-- name   phase  mode  rate  rate  frac   frac  ctrl?  rate   defn
  'PLAT*' 'WAT' 'REIN'  2*          1.0  /
  'R*'    'WAT' 'FLD'   6*                                 'VOID'  /
/
```

If there is a pipeline network coupling the reservoirs, specify it in the usual way with keyword GRUPNET (or BRANPROP and NODEPROP for an Extended Network), defining it down to the master groups. If the Extended Network model is used, the master groups must coincide with source nodes in the network; for each master group there must be a corresponding network source node with the same name. If using keyword NETBALAN, ensure it is set up to balance the network at every time step (which is the default).

If you wish the master run to stop when a slave run finishes for any reason, set its master group's end-run flag in item 8 of keyword GECON. Do **not** set any of the other GECON limits, as the master groups effectively contain no wells, and the master run cannot control workovers in the slaves. Economic limits can, of course, be set in the slave data files.

Set the report times or dates in the usual way. For easier synchronization it is best to use DATES, especially if the slave runs start on different dates. If this is the case, it is also advisable to use the DATE keyword in the SUMMARY section. Aim to set common report dates in each coupled simulation. This is especially important for restarts, as the coupled simulations can only be restarted correctly from RESTART files written on the same date.

The slave reservoirs

The slave reservoirs are represented by standard ECLIPSE data files. These may be restarts from previous runs. You should ensure that they are set up to start at the required date. If they are restart runs, ensure also that the correct file name and report number are entered in the RESTART keyword.

A slave run may be set to start at a date later than the start of the master run. In this case the slave will be "dormant", having no production or injection, until its start date is reached in the master run. Note that the simulation time will not be the same as in the master run, so it is best to use the DATES keyword to specify report times, and also to include the DATE keyword in the SUMMARY section to present the results by calendar date rather than simulation time. An error message will be issued if the slave run is set to start earlier than the master run.

The only additional keyword required in a slave run's data file is GRUPSLAV. This identifies the slave groups, and the associated master groups to which they are coupled. You can also set instructions to ignore any rate limits set in the slave run's data file, or take the minimum of these limits and the limits imposed by the master run. For example, for slave reservoir R2 in Figure 55.1, using the default option of observing only the rate limits communicated from the master for all phases:

```
GRUPSLAV
--
  slave   assoc   oil     wat/liq  gas     resv    oil     wat     gas
--
  group   master  prod    prod     prod    prod    inj     inj     inj

--
  name    group   limits  limits   limits  limits  limits  limits  lim
its
   'FIELD'  'R2'   /
/
```

The slave groups' production rate can be controlled by either guide rate control or prioritization. If you wish to use guide rate control, reference the slave groups with keyword GCONPROD, selecting 'RATE' as the action to be performed on violating any limits:

```
GCONPROD
-- group   ctl   oil   wat   gas   liq   limit   field  guide  g-r
-- name    mode  rate  rate  rate  rate  action  ctrl?  rate   phase
  'FIELD' 'ORAT' 1.0    3*              'RATE'  /
/
```

The default option in GRUPSLAV will over-write any rate limits specified here. Alternatively, if you wish to use prioritization, reference the slave groups with keyword GCONPRI, selecting 'PRI' as the action to be performed on violating each limit (the PRIORITY keyword must previously have been entered):

```
GCONPRI
--
  GROUP    OIL   OIL   WAT   WAT   GAS   GAS   LIQ   LIQ   RESV  PBAL
--
  NAME     RATE  ACTN  RATE  ACTN  RATE  ACTN  RATE  ACTN  RATE  FRAC
   'FIELD'    1*  'PRI'   1*   'PRI'  1*   'PRI'  1*   'PRI'  /
/
```

It is not strictly necessary to set up injection controls for the slave groups with keyword GCONINJE, as injection rate limits imposed by the master will automatically be handled by guide rate control. If injection constraints are already specified in the data file, you can choose either to ignore them or take their minimum with the limits imposed by the master run (see keyword GRUPSLAV). Reinjection and voidage replacement targets applied in the master reservoir to groups superior to the master groups are translated into surface rate and reservoir volume rate targets respectively at the slave groups. If you wish to apply reinjection or voidage replacement targets directly to the master groups, they will be observed more accurately if they are applied instead directly to their corresponding slave groups in the slave reservoir.

Additional production and injection constraints may be applied to subordinate groups in the usual way. But any groups in the slave reservoir that are superior to the slave groups (for example 'FIELD' in slave reservoir R3) must not have any constraints on them, as these would interfere with the targets of the slave groups.

If there is a pipeline network in the slave reservoir, specify it in the usual way with keyword GRUPNET (or BRANPROP and NODEPROP for an Extended Network), defining it from the slave group(s) down to the well groups. The network in the slave reservoir must not extend higher than the slave groups. To couple the network to the master reservoir's network, the slave groups must be defined as fixed-pressure nodes in GRUPNET or NODEPROP. In an Extended Network, for each slave group there must be a corresponding network fixed-pressure node with the same name. The value of the fixed pressure specified here does not matter, as it will be over-written by the nodal pressure communicated from the master. If using keyword NETBALAN, ensure it is set up to balance the network at every time step (which is the default).

There should be sufficient DATES keywords to advance the simulation up to the end of the run. If you "overshoot", the slave run is automatically terminated when the master run ends. If you "undershoot", the slave run will stop at its last report date, and the master run will continue without any production or injection from this slave (unless told to stop with an "end-run" flag in keyword GECON).

Choose the report dates carefully. The time steps in the master run are synchronized so as not to overshoot a report date in any slave. Thus if the report dates differ slightly between the slaves and the master run, this will restrict the time step size throughout the set of coupled reservoirs. So it is best to have a common set of report dates. You will certainly need to synchronize the report dates on which RESTART files are to be written, if you wish subsequently to restart the coupled simulation.

Reading a reservoir coupling file in the master reservoir

A Reservoir Coupling file can be used to take the place of one or more of the slave reservoirs. Instead of activating their slave runs, ECLIPSE reads their flow rates from this file. To use this option, make the following changes to the master reservoir's data file:

- Add the keyword USECUPL, to specify the name and format of the Reservoir Coupling file to be read. If the run spans a sequence of Reservoir Coupling files generated by a series of restarts of the master run, enter the name of the last file in the sequence.
- Remove the particular slave reservoir's data record from the SLAVES keyword. If there are no more slave reservoirs to be actively coupled to the master, the SLAVES keyword should be removed.
- In keyword GRUPMAST, the master groups associated with the substituted slave reservoir must have their slave reservoir name in item 2 defaulted.

For example, to substitute slave reservoir R3 in Figure 55.1 with a Reservoir Coupling file written by the run 'master.data':

```
USECUPL
 'master'  'U' /
SLAVES
--    slave   datafile    machine        directory
--    name    root        hostname       of data file
--
    'RES-R2'  'base'      'rios'        '/usr/models/res2' /
/
GRUPMAST
--  master      slave       associated      limiting
--  group     reservoir       slave        rate change
--  name        name          group         fraction
   'R2'       'RES-R2'       'FIELD'           0.1  /
   'R3-A'        1*            1*     /
   'R3-B'        1*            1*     /
/
```

Driving a slave reservoir with a reservoir coupling file

A slave run may be performed on its own, instead of being activated by the master run. It will read the Reservoir Coupling file to obtain the constraints that the master run originally applied to its slave groups over the course of the simulation. MPI is not required for this option.

To use this option, set up the slave's data file as though it were going to be activated by the master, but in addition include the keyword USECUPL as described above. The slave groups are identified with the keyword GRUPSLAV, and you can choose whether to take their rate constraints from the Reservoir Coupling file, the slave's own data file, or the minimum of both.

You must set up the slave groups for production control (with keyword GCONPROD for guide rate control or keyword GCONPRI for prioritization), and for injection control of the injected phases (with keyword GCONINJE). Reinjection and voidage replacement targets applied in the master reservoir to groups superior to the master groups are translated into surface rate and reservoir volume rate targets respectively at the slave groups.

If the master and slave reservoirs are coupled by a pipeline network, the nodal pressures of the slave groups that were set by the master in the original coupled run will also be read from the Reservoir Coupling file. The slave groups must of course be defined as fixed-pressure nodes in keyword GRUPNET.

A useful way of monitoring the production and injection constraints applied to the slave groups is to write them to the SUMMARY file using the SUMMARY section keywords GOPRL, GOIRL, GWPRL, GWIRL, GGPRL, GGIRL, GLPRL, GVPRL, GVIRL. See the SUMMARY section description in this manual. For example, to drive slave reservoir R3 in Figure 55.1 from the Reservoir Coupling file written by the run 'master.data', but imposing an additional constraint on the gas production:

```
USECUPL
 'master'  'U' /
GRUPSLAV
--
  slave   assoc   oil     wat/liq  gas     resv    oil     wat     gas
--
  group   master  prod    prod     prod    prod    inj     inj     inj

--
  name    group   limits  limits   limits  limits  limits  limits  lim
its
   'R3-A'     1*      1*      1*     'BOTH'  /
   'R3-B'     1*      1*      1*     'BOTH'  /
/
GCONPROD
-- group   ctl   oil   wat   gas   liq   limit   field  guide  g-
r    .....
--
 name    mode  rate  rate  rate  rate  action  ctrl?  rate   phase  ..
...
  'R3-*'  'ORAT' 1.0    1*   2.E5   1*   'RATE'  /
/
GCONINJE
-- group  inj    ctl   surf  resv  reinj  vrep  field  guide  g-r
-- name   phase  mode  rate  rate  frac   frac  ctrl?  rate   defn
  'R3-*'  'WAT' 'RATE'  10       /
/
```

Using the gas field operations model in coupled reservoirs

The Gas Field Operations model (see "Gas Field Operations Model" on page 359) may be used to a limited extent in coupled reservoirs. The master run handles all the decisions concerning the calculation of DCQs, while the slaves take the more passive role of communicating to the master their production rates over each pass of the contract year, and their delivery capacities. The master run synchronizes with the slaves to start each pass of the contract year, and sets the target rate of each slave group every month for calculating the swing, the profile, and delivery capacities.

The restrictions on running the model are as follows.

1 All contract groups must be situated in the master reservoir; there must be no contract groups in the slaves. The master run should contain the keyword SWINGFAC (for a single contract group) or GSWINGF (for multiple contract groups). These keywords must not be used in the slave runs.

2 Each slave must use the GASYEAR or GASPERIO keyword in synchronization with the master. Each contract period must start on the same date and be of the same length in all the coupled runs. In the slave runs, all GASYEAR keyword items except item 1, or all GASPERIO keyword items except items 1 and 2, are ignored and should be defaulted, as there are no contract groups that require their DCQ to be calculated. In the master run, the time steps will be limited to one month, regardless of the setting of GASYEAR item 4 or GASPERIO item 5.

3 If the PRO option is selected in item 3 of keyword GASYEAR or item 4 of GASPERIO, the master run cannot activate the drilling queue in a slave reservoir to keep the delivery capacity above DCQ x swing factor. This is because the master has no information about individual wells in the slave reservoirs.

4 All automatic compressors (see keyword GASFCOMP) must be situated in the master reservoir. Any in the slave reservoirs will be ignored.

5 In the master reservoir, there must be no gas consumption or import rate specified for the master groups (keyword GCONSUMP). This is because their consumption or import rates are set equal to those of their corresponding slave groups in the slave reservoirs. This also applies to consumption by automatic compressors. An automatic compressor (keyword GASFCOMP) located at a master group should not be given a gas consumption rate.

6 Reservoir Coupling files should only be used with care. They store the master groups' flows during the second pass of the contract year, when producing at DCQ x profile factor. But the flows during the first pass (when producing at DCQ x swing factor) are **not** recorded. Thus if you replace a slave reservoir by a reservoir coupling file, its production rate in the first pass will be the same as in the second pass. This will only be correct when the swing factors are the same as the profile factors. A Reservoir Coupling file cannot be used to drive a slave on its own when it is using the Gas Field Operations model.

Sharing licenses with the slaves

There was a major change in the licensing model used with Reservoir Coupling in 2005A. Prior to 2004A a reservoir coupling job would check out only one `eclipse` license and one feature license no matter how many SLAVES there were.

From 2005A this policy was modified so that each SLAVE requires an ECLIPSE license as well as the master. Each SLAVE also requires one feature license for every feature it uses.

The following example indicates how works:

Master

SLAVE 1 uses LGR and Multisegwells

SLAVE 2 uses LGR and networks

SLAVE 3 uses networks

Prior to 2004A the following licenses would have been required to run this model:

- 1 `eclipse`
- 1 `lgr`
- 1 `network`
- 1 `multisegwells`
- `rescoupling`

Since 2005A this is:

- 4 `eclipse`
- 2 `lgr`
- 2 `network`
- 1 `multisegwells`
- 1 `rescoupling`

Reservoir coupling file structure

The Reservoir Coupling file structure is similar to the structure of other files that ECLIPSE reads and writes. This is described in "File Handling in ECLIPSE" on page 303. The file consists of a series of data blocks, each headed by a descriptor record. But there is an additional type of block descriptor record that is not mentioned in the File Handling chapter: the “message record”. The format of this type of record is:

- An 8-character keyword
- The integer zero
- The 4-character string 'MESS'

This type of record is **not** followed by a data block - the only information the record communicates is the 8-character keyword itself.

The file begins with the descriptor record

```
'RESTART ' 9 'CHAR'
```

that is followed by the root name of the input `RESTART` file, divided into 9 8-character elements. If the run is a base run and not a restart, the 9 elements are all blank.

The next descriptor record is

```
'INTEHEAD' 5 'INTE'
```

that is followed by the integer header block containing:

1 = the internal version number of the program release that wrote the file (for example 9501). This is used in future releases to identify files from previous releases.

2-4 = the day (1-31), month number (1-12) and year (for example 1994) of the start of the master base run (as set in `RUNSPEC` section keyword `START`).

5 = the unit convention used (1 = metric, 2 = field, 3 = lab).

Next, for every time step of the master run, the following records are written:

```
'TIMESTEP' 2 'DOUB'
```

followed by the start and end time of the time step, in days from the master base run’s start date.

For each master group, the following records are written:

```
'GRUPNAME' 2 'CHAR'
```

followed by the name of the master group and the name of its associated slave group in the corresponding slave run.

```
'RATES ' 22 'DOUB'
```

followed by the group’s production and injection rates of each phase and other information to account for free and dissolved gas, gas consumption and lift gas:

1-4 = oil, water, gas and reservoir volume production rates

5-8 = oil, water, gas and reservoir volume injection rates

9-11 = separate reservoir volume rates of oil, water and gas injection

12-13 = total gas consumption and import rates at and below the group

14-15 = vaporized oil and dissolved gas production rates (used for `SUMMARY` section output generated by the keywords `GOPRF`, `GOPRS`, `GGPRF`, `GGPRS`, etc.)

16 = lift gas supply rate, reduced by well efficiency factors

17 = lift gas supply rate, **not** reduced by well efficiency factors (Items 16 and 17 are intended for future use in lift gas accounting.)

18-22 = oil, water, gas, resv rates and gas consumption at delivery capacity (Gas Field Operations model)

23-27= energy production and injection rate, total energy consumption rate at and below this group, energy production rate at delivery capacity and energy consumption rate at delivery capacity (Gas Calorific Value Control)

.

```
'POTNS ' 10 'DOUB'
```

followed by the group's potential production and injection rates of each phase (that is the sum of the well potentials):

1-4 = oil, water, gas and reservoir volume production potentials

5-6 = oil surface rate and reservoir volume rate injection potentials

7-8 = water surface rate and reservoir volume rate injection potentials

9-10 = gas surface rate and reservoir volume rate injection potentials.

11 = energy production rate potential (Gas Calorific Value Control)

12 = free gas production potential

.

```
'CTLMODE ' 4 'INTE'
```

followed by integers defining the group's acting production and injection control modes:

1 = production control mode

(0 = `NONE`, 1 = `ORAT`, 2 = `WRAT`, 3 = `GRAT`, 4 = `LRAT`, 5 = `RESV`, 6 = `PRBL`, 7 = `ERAT`)

2-4 = oil, water and gas injection control modes

(0 = `NONE`, 1 = `RATE`, 2 = `RESV`, 3 = `REIN`, 4 = `VREP`

)

```
'LIMITS ' 21 'DOUB'
```

followed by the limiting production and injection rates of each phase, and the group's nodal pressure if there is a network:

1 -5 = oil, water, gas, liquid and reservoir volume production rate limits (default = 1.0E20)

6 = limiting production balancing fraction (default = 1.0E20)

7-8 = oil surface rate and reservoir volume rate injection limits (default = 1.0E20)

9-10 = oil reinjection fraction and voidage replacement fraction limits (default = 1.0E20)

11-12 = water surface rate and reservoir volume rate injection limits (default = 1.0E20)

13-14 = water reinjection fraction and voidage replacement fraction limits (default = 1.0E20)

15-16 = gas surface rate and reservoir volume rate injection limits (default = 1.0E20)

17-18 = gas reinjection fraction and voidage replacement fraction limits (default = 1.0E20)

19 = nodal pressure in production network (default = 0.0)

20-21 = nodal pressure in water and gas injection networks (default = 0.0).

22 = Energy production rate limit (default = 1.0E20)

.

```
'ENDSTEP ' 0 'MESS'
```

to signify the end of the information for the current time step.

The `RATES` and `POTNS` keyword data are required when the Reservoir Coupling file is used in the master run to substitute for a slave reservoir. The potentials are used to calculate group guide rates if a superior group has been given a guide rate of 0.0 in `GCONPROD`.

The `CTLMODE` and `LIMITS` keyword data are required when the Reservoir Coupling file is used to drive a slave run on its own. These keywords are not written to the file for groups whose slave reservoirs are “dormant” (that is, whose start date has not yet been reached by the master run).

The structure and contents of the file are likely to evolve as the Reservoir Coupling facility is developed further, but back-compatibility is maintained wherever possible.

Reservoir Coupling files can be written either formatted (as a text file) or unformatted (as a binary file). Formatted files occupy much more disk space than unformatted files, but this type should be chosen if the file is going to be used on a system that does not have binary compatibility with the one on which it was created. The Convert utility can convert Reservoir Coupling files from one format to the other (select the “General” file type).

The Reservoir Coupling file that is created by the master run has the same root name as the master run’s data file. The filename’s extension is `.CPL` for an unformatted file, and `.FCP` for a formatted file.

Example section of a reservoir coupling file

```
'RESTART '           9 'CHAR'
'        ' '        ' '        ' '        ' '        ' '        ' '
       '
'        ' '        '
'INTEHEAD'           5 'INTE'
           9402           1           1        1994           2
'TIMESTEP'           2 'DOUB'
 0.00000000000000D+00   0.10000000000000D+01
'GRUPNAME'           2 'CHAR'
'R3-A    ' 'R3-A    '
'RATES   '          27 'DOUB'
 0.99995623191871D+04   0.43768081294321D+00   0.14973184139827D+05
 0.11192063433604D+05   0.00000000000000D+00   0.00000000000000D+00
 0.15000000000000D+05   0.87697823644459D+04   0.00000000000000D+00
 0.00000000000000D+00   0.87697823644459D+04   0.00000000000000D+00
 0.00000000000000D+00   0.00000000000000D+00   0.14973184125069D+05
 0.00000000000000D+00   0.00000000000000D+00   0.00000000000000D+00
 0.00000000000000D+00   0.00000000000000D+00   0.00000000000000D+00
 0.00000000000000D+00   0.00000000000000D+00   0.00000000000000D+00
 0.00000000000000D+00   0.00000000000000D+00   0.00000000000000D+00

'POTNS   '          11 'DOUB'
 0.68819213068446D+05   0.00000000000000D+00   0.10322881960267D+06
 0.77047083871076D+05   0.00000000000000D+00   0.00000000000000D+00
 0.00000000000000D+00   0.00000000000000D+00   0.67192104926539D+05
 0.39284009120984D+05   0.00000000000000D+00

'CTLMODE '           4 'INTE'
           4           0           2           1
'LIMITS  '          22 'DOUB'
 0.10000000200409D+21   0.10000000200409D+21   0.55036185820205D+05
 0.10000000000000D+05   0.10000000200409D+21   0.10000000200409D+21
 0.10000000200409D+21   0.10000000200409D+21   0.10000000200409D+21
 0.10000000200409D+21   0.10000000200409D+21   0.87697823644459D+04
 0.10000000200409D+21   0.10000000200409D+21   0.15000000000000D+05
 0.10000000200409D+21   0.10000000200409D+21   0.10000000200409D+21
 0.00000000000000D+00   0.00000000000000D+00   0.00000000000000D+00
 0.00000000000000D+00
```

```
'GRUPNAME'           2 'CHAR'
'R3-B    ' 'R3-B    '
'RATES   '          27 'DOUB'
0.99989832425748D+04   0.10167574252128D+01   0.14963814179795D+05
0.11190869653077D+05   0.00000000000000D+00   0.91405301592612D+04
0.00000000000000D+00   0.91431416015625D+04   0.00000000000000D+00
0.91431416015625D+04   0.00000000000000D+00   0.00000000000000D+00
0.00000000000000D+00   0.00000000000000D+00   0.14963814179716D+05
0.00000000000000D+00   0.00000000000000D+00   0.00000000000000D+00
0.00000000000000D+00   0.00000000000000D+00   0.00000000000000D+00
0.00000000000000D+00   0.00000000000000D+00   0.00000000000000D+00
0.00000000000000D+00   0.00000000000000D+00   0.00000000000000D+00

'POTNS   '          11 'DOUB'
0.35029663767571D+05   0.00000000000000D+00   0.52544495651356D+05
0.39217731821364D+05   0.00000000000000D+00   0.00000000000000D+00
0.36486043687679D+05   0.36496467721760D+05   0.00000000000000D+00
0.00000000000000D+00   0.00000000000000D+00

'CTLMODE '           4 'INTE'
          4           0           2           1
'LIMITS  '          22 'DOUB'
0.10000000200409D+21   0.10000000200409D+21   0.55026815860173D+05
0.10000000000000D+05   0.10000000200409D+21   0.10000000200409D+21
0.10000000200409D+21   0.10000000200409D+21   0.10000000200409D+21
0.10000000200409D+21   0.10000000200409D+21   0.91431420260235D+04
0.10000000200409D+21   0.10000000200409D+21   0.00000000000000D+00
0.10000000200409D+21   0.10000000200409D+21   0.10000000200409D+21
0.00000000000000D+00   0.00000000000000D+00   0.00000000000000D+00
0.00000000000000D+00

'GRUPNAME'           2 'CHAR'
'R2      ' 'FIELD   '
'RATES   '          27 'DOUB'
0.00000000000000D+00   0.00000000000000D+00   0.00000000000000D+00
0.00000000000000D+00   0.00000000000000D+00   0.00000000000000D+00
0.00000000000000D+00   0.00000000000000D+00   0.00000000000000D+00
0.00000000000000D+00   0.00000000000000D+00   0.00000000000000D+00
0.00000000000000D+00   0.00000000000000D+00   0.00000000000000D+00
0.00000000000000D+00   0.00000000000000D+00   0.00000000000000D+00
0.00000000000000D+00   0.00000000000000D+00   0.00000000000000D+00
0.00000000000000D+00   0.00000000000000D+00   0.00000000000000D+00
0.00000000000000D+00   0.00000000000000D+00   0.00000000000000D+00

'POTNS   '          11 'DOUB'
0.00000000000000D+00   0.00000000000000D+00   0.00000000000000D+00
0.00000000000000D+00   0.00000000000000D+00   0.00000000000000D+00
0.00000000000000D+00   0.00000000000000D+00   0.00000000000000D+00
0.00000000000000D+00   0.00000000000000D+00

'ENDSTEP '           0 'MESS
```

Summary of keywords

SUMMARY section

The following keywords control output of data relevant to the Reservoir Coupling Facility:

Table 55.1 Reservoir coupling SUMMARY output controls

Field	Group	Well	Description
	GOPRL		Group Oil Production Rate Limit
	GOIRL		Group Water Production Rate Limit
	GWPRL		Group Oil Injection Rate Limit
	GWIRL		Group Water Injection Rate Limit
	GGPRL		Group Gas Production Rate Limit
	GGIRL		Group Gas Injection Rate Limit
	GLPRL		Group Liquid Production Rate Limit
	GVPRL		Group reservoir Volume Production Rate Limit
	GVIRL		Group reservoir Volume Injection Rate Limit

These keywords are intended for use with slave groups, in a slave reservoir being run on its own, when the rate limits of its slave groups are read from a Reservoir Coupling File. The keywords allow you to examine the rate constraints in the file that are applied to the slave groups.

When no rate limit is being applied, the value of the limit is normally set to 1.0E20. However, the automatic axis scaling in GRAF would make the y-axis extend to this value. To avoid this, infinite rate limits are reset to zero before writing to the Summary File.

SCHEDULE section

The keywords are:

Table 55.2 SCHEDULE section

Keyword	Description
DEBUG	Switch 39 turns on debug from the Systems Annex for MPI calls. Switch 42 turns on debug from the Reservoir Coupling facility.
DUMPCUPL	Instructs the master run to write a Reservoir Coupling File, containing the flows and applied constraints of the master groups.
GRUPMAST	Must be used in the master run to identify the master groups and to indicate which slaves they are associated with.
GRUPSLAV	Must be used in each slave run to identify the slave groups and their associated master groups in the master run, and to set the filter flags that determining which rate constraints are to be imposed on them

Table 55.2 SCHEDULE section

Keyword	Description
RCMASTS	Sets a minimum length for time steps restricted by a group's limiting fractional change in flow rate.
SLAVES	Must be used in the master run to identify the slave run data files and to indicate where they are located; their simulation processes will be started when this keyword is read
USECUPL	Instructs ECLIPSE to read group flows or constraints from a Reservoir Coupling File

Restarts

Chapter 56

Introduction

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

Restarts in ECLIPSE are described separately for ECLIPSE 100 and ECLIPSE 300 below.

For ECLIPSE 100, see "Restarts in ECLIPSE 100" on page 836.

For ECLIPSE 300, see "Restarts in ECLIPSE 300" on page 841.

Restarts in ECLIPSE 100

Restarting an ECLIPSE 100 run

There are two methods of restarting an ECLIPSE 100 run:

Fast Restart

This reads the RUNSPEC, GRID, EDIT, PROPS and REGIONS section data from a SAVE file. The data is stored in a processed form, so that for example the transmissibilities do not have to be re-calculated. The SAVE keyword must be used in RUNSPEC section of the original run to tell ECLIPSE 100 to construct a SAVE file. An important restriction with Fast Restarts is that the SAVE file must have been produced by the **current** version of ECLIPSE 100; the SAVE files are not compatible across different versions of ECLIPSE.

Flexible Restart

This reads a complete data file. This method is slower than the other one, as the data must be processed again (for example the transmissibilities are re-calculated). But it allows more flexibility, as it is possible to change some of the data items from their values in the original run, and restarts from files produced by earlier versions of ECLIPSE 100 are possible.

Both methods require the time-dependent data (for example, grid block solution variables, status of wells etc.) to be stored at one or more times in the original simulation run. By default, the time-dependent data is stored on the RESTART file. But alternatively, the time-dependent restart data may be appended to the SAVE file, so that the RESTART file can be used purely for graphics output.

“Fast Restarts” from a SAVE file

Restarting a run from a SAVE file will usually be quicker to initialize than a restart from a complete data file. This method can be used for simple restarts where only the SCHEDULE section data is to be modified.

The procedure is as follows:

1 In the base run, use the SAVE keyword to instruct ECLIPSE 100 to create a SAVE file, and use the RPTSCHED or RPTRST keyword to control the frequency with which the solution data is to be written to the RESTART file.

2 Set up the data file for the restart run, consisting of a LOAD keyword, a RESTART keyword, the SUMMARY and SCHEDULE section data. For example,

```
LOAD
 'BASE' /
RESTART
 'BASE' 11 /
SUMMARY
INCLUDE
 'SUMMARY.DAT' /
SCHEDULE
 etc.
```

Note The LOAD keyword replaces the RUNSPEC, GRID, EDIT, PROPS and REGIONS sections, and opens the SOLUTION section where the RESTART keyword is expected. The SAVE file that it names must have been produced by the current version of ECLIPSE 100; fast restarts cannot be performed from SAVE files produced by earlier versions. Formatted SAVE files require the fourth item of the LOAD keyword to be set appropriately.

Note For a parallel fast restart run the PARALLEL keyword should not be present in the restart data file since the record is read from the SAVE file instead. However, a parallel fast restart run must be executed on the same number of processors as the base run.

In the SCHEDULE section, unless keyword SKIPREST has been entered, delete the well and time-stepping keywords up to and including the restart time, at report number 11 in the example above (but keep any VFP tables, as these are not passed over in the RESTART file). If keyword SKIPREST is present, you need not delete these keywords as ECLIPSE 100 automatically skips the unwanted keywords until the restart time has been reached in the SCHEDULE data. Modify the SCHEDULE data after the restart time as required.

3 Restarting from time-dependent data stored in the SAVE file:

If the time-dependent restart data has been stored in the SAVE file, the RESTART keyword requires up to two extra items of data, for example:

```
RESTART
 'BASE'  11  'SAVE'  'FORMATTED' /
```

The string 'SAVE' indicates that the restart data is present in the SAVE file, and the string 'FORMATTED' indicates that the SAVE file is formatted. By default the SAVE file is assumed to be unformatted.

“Flexible Restarts” from a data file

A run can be restarted by supplying a complete data file, and taking the initial solution from the RESTART file. This method has the advantages of not requiring the SAVE file, and of allowing restarts from older versions of ECLIPSE 100. It is also enables the restart to be run with modified data; for example the maximum number of wells can be increased (but not reduced). It is possible to restart a run with, for example, altered permeability data; but in general such changes are not recommended.

To do a restart from (say) the 11th report time of run BASE, the procedure is:

1 Think of a new file name root - say R1.

2 Copy the data file BASE.DATA into R1.DATA.

3 Edit R1.DATA

In the SOLUTION section, delete all equilibration or enumeration keywords, and any analytic aquifer keywords. Insert the RESTART keyword to specify the RESTART file and report number,

```
RESTART
 'BASE' 11 /
```

In the SCHEDULE section, unless keyword SKIPREST has been entered, delete the well and time-stepping keywords up to and including the restart time, at report number 11 in the example (but keep any VFP tables, as these are not passed over in the RESTART file). If keyword SKIPREST is present, you need not delete these keywords as ECLIPSE automatically skips the unwanted keywords until the restart time has been reached in the SCHEDULE data. Modify the SCHEDULE data after report 11 as required.

4 Submit the run with R1 as the root filename.

The restart run is slower to initialize than a restart from a SAVE file, because the geometry data has to be processed to calculate the pore volumes, transmissibilities etc.

It is possible also to do a flexible restart from time-dependent restart information stored on a SAVE file, even though the SAVE file has not been used to provide the non-recurrent data as it would in a fast restart. In this case the RESTART keyword needs up to two extra items of data, for example:

```
RESTART
 'BASE'  11  'SAVE'  'UNFORMATTED' /
```

This keyword will request that the time dependent data for the restart is taken from the unformatted SAVE file generated by the base run.

Requesting RESTART files in ECLIPSE 100

By default, the restart data written at each report time will be stored in separate RESTART files. The suffix of the files indicates the report number at which the data was written. Optionally, the time-dependent restart data written at each report can be stored in a single “Unified” RESTART file instead. If you wish to produce a Unified RESTART file, enter the keyword UNIFOUT in the RUNSPEC section. To enable ECLIPSE 100 to read a Unified RESTART file, the keyword UNIFIN must be present in the RUNSPEC section - otherwise ECLIPSE 100 looks for non-unified RESTART files.

By default, any RESTART files created are unformatted files. If you wish to write formatted RESTART files instead, enter the keyword FMTOUT in the RUNSPEC section. To enable ECLIPSE to read formatted RESTART files, the keyword FMTIN must be present in the RUNSPEC section - otherwise ECLIPSE 100 looks for unformatted RESTART files. Files may be converted from formatted to unformatted and vice-versa by means of the utility program Convert. See "File Handling in ECLIPSE" on page 303 for more details on file formats.

To create RESTART files from which a later run can restart, the procedure is:

1. Set either the mnemonic 'BASIC' in the RPTRST keyword (switch 1) > 0, or set the mnemonic 'RESTART' in the RPTSCHED keyword (switch 7) > 0.
2. If either switch is set to 1, RESTART files are created at every report time, but deleted when a later restart is available. This facility is intended to provide a fail-safe restart. (Note that this facility is not available if Unified RESTART files are being output).
3. If the switch is = 2 (or = 1 also for Unified files), RESTART files are created at every report time until the switch is reset. None are deleted.
4. Giving the mnemonic 'BASIC' in the RPTRST keyword a value > 2 results in the automatic generation of RESTART files at specific intervals of time.
5. A RESTART file can also be generated at time 0.0 days, containing the initial solution, either by setting the mnemonic 'RESTART' (switch 7) in the RPTSOL keyword to 2, or by setting the mnemonic 'BASIC' to 2 in keyword RPTRST in the SOLUTION section. There must be at least one TIME, TSTEP or DATES keyword present to generate the time zero RESTART file.
6. A SAVE file, containing the non-recurrent data of the base run, is generated by specifying the SAVE keyword in the RUNSPEC section. A SAVE file is required if a Fast Restart is to be performed.
7. If you want the time-dependent restart data to be appended to the SAVE file, use the 'SAVE' mnemonic (switch 17) in keyword RPTRST. This controls the frequency with which the restart data is written to the SAVE file, in an analogous manner to the 'BASIC' mnemonic. The SAVE keyword must also be present in the RUNSPEC section. If the RESTART file is to be used for graphics output only, unwanted restart arrays may be prevented from being written to this file by using the NORST mnemonic (switch 16) in the RPTRST keyword.

SCHEDULE section data not stored in the ECLIPSE 100 RESTART file

Apart from the exceptions detailed below, all the SCHEDULE section data from the base run is written to the RESTART file and applied in the restart run. The exceptions are the following keywords, whose data is not carried over on the RESTART file:

- VFP tables (keywords VFPPROD, VFPINJ)
- Reporting instructions (keywords RPTSCHED, RPTRST)
- The "global keywords" that can be entered in any section (COLUMNS, DEBUG, ECHO, EXTRAPMS, FORMFEED, INCLUDE, MESSAGES, NOECHO, NOWARN, OPTIONS)

Gas Field Operations Model only

- Annual scheduling files in the Gas Field Operations Model (keywords GASBEGIN, GASEND and any keywords in between). See "Operations and reports during the contract period" on page 362 for further information.

Reservoir Coupling Only

- The keywords SLAVES, DUMPCUPL and USECUPL in the Reservoir Coupling Facility. See "Reservoir Coupling" on page 811 for further information.
- Well PI Scaling tables (keyword PIMULTAB).
- Scale deposition and damage tables (keywords SCDPTAB and SCDATAB). See "Scale deposition model" on page 1209 for further information.

Notes on RESTARTS in ECLIPSE 100

1 The following data cannot be changed at a restart:

   a The grid dimensions NDIVIX, NDIVIY, NDIVIZ, and the number of active cells.

   b The radial/Cartesian geometry switch.

   c The phases present and the permitted mass transfers (OIL, WATER, GAS, dissolved GAS, vaporized OIL).

   d The unit convention.

Vertical Equilibrium Model only

   e The Vertical Equilibrium options, keyword VE in the RUNSPEC section.

   f The type of well model for existing wells.

2 Passive tracers can be added in a flexible restart, even if the base run had no tracers. They should be initialized as usual in the SOLUTION section with keyword TBLK or TVDP.

3 The maximum number of wells, groups, connections per well and wells per group may be increased at a flexible restart (keyword WELLDIMS in the RUNSPEC section), but cannot be decreased.

4 A warning message is issued if the RESTART file was not generated by the current version of ECLIPSE 100. In general, using flexible restarts, ECLIPSE 100 can usually be restarted from RESTART files written by the two previous versions of the program, unless options not available in the original run are to be used in the restarted run. A RESTART file produced by ECLIPSE 100 cannot be loaded into a previous version of the code. If it is not possible to restart the run from the RESTART file, an error message is issued.

Restarts in ECLIPSE 300

The method used for restart runs in ECLIPSE 300 is based upon the SAVE keyword and either the LOAD or RESTART keyword.

Whenever a SAVE keyword is encountered in the SCHEDULE section, the state of the simulation is written to the RESTART file. (This essentially involves writing the solution and well structures.) All the data that does not change during the simulation (pore volumes, depths, region numbers etc.) is written out once either to the start of a unified RESTART file, or to a separate file with extension .V0000 (.D0000 if formatted SAVE files are requested). Subsequently, a note is written to the main output after each step has been written to the RESTART file, giving the step number (simply the number of report steps to date).

To perform a restart run, the input file takes the following form:

```
- - Restart run
LOAD
'BASE' 1 /
SCHEDULE
RPTSCHED
'PRESSURE' 'SOIL' /
TSTEP
1 /
- - Continue with SCHEDULE section...
```

The first LOAD keyword replaces the RUNSPEC section, and all other sections before the SCHEDULE section. In the above example, the state of the simulation with root BASE after time-step 1 are loaded, and the next TSTEP command continues from that point, exactly as it would have in the base run. The wells, properties, grid description etc. are obtained from the LOAD operation. Unless a unified SAVE file is requested, to restart a run from step n both the initial file (.V0000) and the corresponding file (.V000n) are required.

In the SCHEDULE section, unless keyword SKIPREST has been entered, delete the well and time-stepping keywords up to and including the restart time, at report number 1 in the example above (but keep any VFP tables, as these are not passed over in the RESTART file). If keyword SKIPREST is present, you need not delete these keywords as ECLIPSE 300 automatically skips the unwanted keywords until the restart time has been reached in the SCHEDULE data

A further restart may be performed from a restart. The report step count is continued up through the sequence of runs. Note, however, that it is not possible to use the same root name for base and restart runs.

If the SAVE file is unified or formatted through the use of the UNIFSAVE and FMTSAVE keywords in the original run, the UNIFIN and FMTIN keywords should be placed before the LOAD keyword in the restart run.

Note The SAVE file that it names must have been produced by the current version of ECLIPSE 300; fast restarts cannot be performed from SAVE files produced by earlier versions.

Note The use of fast restarts is not compatible with the PARALLEL option.

Using the RESTART keyword, rather than the LOAD keyword, provides a flexible method of using a solution from a previous run to act as the starting point for a new one. A complete data file is provided, and sections up to SOLUTION are processed as usual. The initial solution is then taken from the RESTART file, rather than being obtained by equilibration. The existing production system solution is also loaded, so that all wells and groups present in the base run are carried over into the new run.

It is possible to change the data in sections before the SOLUTION section on a restart run. For example, new transmissibility multipliers might be entered in the EDIT section. This type of change must be performed with care, and changes to the GRID section which alter the pore volumes are not recommended.

To do a restart from (say) the 11th report time of run BASE, the procedure is:

1 Think of a new file name root
   - For example, R1.
2 Copy the data file BASE.DATA into R1.DATA.
3 Edit R1.DATA

   In the SOLUTION section:

   a Delete all equilibration or enumeration keywords, and any analytic aquifer keywords.

   b Insert the RESTART keyword to specify the RESTART file and report number.

   In the SCHEDULE section:

   a Unless keyword SKIPREST has been entered, delete the well and time-stepping keywords up to and including the restart time at report number 11 (but keep any VFP tables, as these are not passed over in the restart file). If keyword SKIPREST is present, you need not delete these keywords as ECLIPSE 300 automatically skips the unwanted keywords until the restart time has been reached in the SCHEDULE data.

   b Modify the schedule data after report 11 as required.
4 Submit run with R1 as the root filename.

The restart run is slower to initialize than a restart from a SAVE file, because the geometry data has to be processed to calculate the pore volumes, transmissibilities etc.

Requesting RESTART files in ECLIPSE 300

To create RESTART files from which a later run can restart, specify the item 'RESTART' in the list of mnemonics entered with RPTRST.

Note that if additional restart files are required to be output from a restart run, then the RPTRST keyword, together with the mnemonic 'RESTART' is required in the restart run. ECLIPSE 300 does not carry this information from the base run through to all subsequent restart runs.

The following data cannot be changed at a restart:

1 The grid dimensions N_x, N_y, N_z, and the number of active cells.

2 The radial/cartesian switch.

3 The number of components present.

4 The unit convention.

Passive tracers can be added in a restart, even if the base run had no tracers. They should be initialized as usual in the SOLUTION section with keyword TBLK or TVDP.

ECLIPSE 300 can be restarted from RESTART files written by the previous versions of the program, unless options not available in the original run are to be used in the restarted run.

In general, keyword data not specified in the SCHEDULE section of the restart run is carried over from the base run (apart from VFP tables, which are not written to the RESTART file).

Rivers

Chapter 57

Introduction

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL

This chapter describes how rivers are modeled in the ECLIPSE 100 simulator. Within ECLIPSE it is possible to model the complex interaction between surface river networks and subsurface aquifers for both saturated and unsaturated flows. This is achieved by computing a solution to the one-dimensional Saint-Venant equations, which model open channel flow, at each aquifer time-step. The Saint-Venant equations are coupled fully implicitly with the groundwater equations and in this manner ECLIPSE is able to provide an accurate solution.

Figure 57.1 Figure showing a simple river network illustrating the river reach structure

River keywords

RUNSPEC section keywords

List of RUNSPEC section keywords

Table 57.1 RUNSPEC keywords

Keyword	Description
RIVRDIMS	River dimension data and activates river option

PROPS section keywords

List of PROPS section keywords

Table 57.2 PROPS keywords

Keyword	Description
DEPTHTAB	Tables of river depth data for downstream boundary conditions
MASSFLOW	Tables of river massflow data for upstream boundary conditions
QHRATING	Tables containing relationships between massflow and depth for downstream boundary conditions
RBEDCONT	Tables of riverbed/grid-block contact area data
RIVRXSEC	Tables of river channel profile data

SOLUTION section keywords

List of SOLUTION section keywords

Table 57.3 SOLUTION keywords

Keyword	Description
RIVERSYS	Defines the branch structure and boundary conditions for a river

SUMMARY section keywords

List of SUMMARY section keywords

Table 57.4 SUMMARY keywords

Segment	Description	Metric	Field	Lab
SRRFLOW	Reach flux through cross-sectional area at current time	m^3/day	ft^3/day	cm^3/hr
SRRAREA	Reach area at current time	m^2	ft^2	cm^2
SRRDEPTH	Reach water depth at current time	m	ft	cm
SRREXCH	Reach exchange flow between river and grid at current time	m^3/day	ft^3/day	cm^3/hr
SRRHEAD	Reach hydraulic head at current time	m	ft	cm
SRRFRODE	Reach Froude number			
SRRVEDER	Reach Vedernikov number			
SRRQR	Reach exchange flow between river and grid over time step	m^3	ft^3	cc (cm^3)
SRRQT	Reach cumulative exchange flow between river and grid over all time	m^3	ft^3	cc (cm^3)
SRBQR	Branch exchange flow between river and grid over time step	m^3	ft^3	cc (cm^3)
SRBQT	Branch cumulative exchange flow between river and grid over all time	m^3	ft^3	cc (cm^3)
SRTQR	River total exchange flow between river and grid over time step	m^3	ft^3	cc (cm^3)
SRTQT	River total cumulative exchange flow between river and grid over all time	m^3	ft^3	cc (cm^3)
SRTFR	Reach tracer flow rate	Tracer unit/day	Tracer unit/day	Tracer unit/hr
SRTFC	Reach tracer concentration	Tracer unit/m^3	Tracer unit/ft^3	Tracer unit/cc
SRSFR	Reach brine flow rate	Kg/day	lb/day	g/hr
SRSFC	Reach brine concentration	Kg/m^3	lb/ft^3	g/cc

Table 57.5 Flow rate and total water keywords

Connection	Description	Metric	Field	Lab
CRREXCH	Exchange flow rate between river and grid	m^3/day	ft^3/day	cm^3/hr
CRRPROT	Total water production	m^3/day	ft^3/day	cm^3/hr
CRRINJT	Total water injection	m^3/day	ft^3/day	cm^3/hr

SCHEDULE section keywords

List of SCHEDULE section keywords

Table 57.6 SCHEDULE keywords

Keyword	Description
COMPRIV	Specifies the grid blocks which connect to a river
DEBUG	Item 76 controls the output of debug information for rivers. This keyword may also be placed in any other section
GRDREACH	Defines how the river reaches connect to the grid block connections
REACHES	Defines the reaches within a river
RIVDEBUG	Allow debug output for rivers
RIVERSYS	Defines the branch structure and boundary conditions of a river
RIVRPROP	Allow modification of river reach properties
RIVSALT	Allow salt injection in rivers
RIVTRACE	Allow tracers injection in rivers

List of WRFTPLT mnemonics

Table 57.7 RFT, PLT and river data output using WRFTPLT keyword

Mnemonic	Description
READEPTH	Reach depth
REALENST	Reach length
REALENEN	Reach cumulative distance from river mouth
REAXCORD	X-coordinate of reach node
REAYCORD	Y-coordinate of reach node
REACHFLX	Reach flux
REACHARE	Reach cross-sectional area
REACHDEP	Reach river water depth
REACHEXG	Reach exchange between river and grid
REAFRODE	Reach Froude number
REACHNBR	Reach number
REACHBRN	Reach branch number
REACHNXT	Outlet reach number
BRNST	First reach in river branch
BRNEN	Last reach in river branch
BRANNAME	River/branch name

List of RPTSCHED mnemonics:

Table 57.8 RPTSCHED keyword output

Mnemonic	Description
WELSPECS	Output the river reach and connection SCHEDULE section data to the print file.

Using the River option

The River option in ECLIPSE is enabled by specifying the keyword RIVERSYS together with other related keywords. The RIVERSYS keyword defines the name of the river, the names of the river branches, the solution method to be used and the upstream/downstream boundary conditions.

The structure of each river is specified by data input using the COMPRIV, REACHES and GRDREACH keywords in the SCHEDULE section. The keyword COMPRIV is used to input a list of grid blocks that are connected to the river. Note that rivers cannot be connected to cells in local grid refinements. (It is possible to connect a river reach to an inactive grid block, but in this case the exchange flow between the river and the aquifer is not computed.) The REACHES keyword is used to define the reach structure of the river and also the required parameters for each reach such as Manning's roughness coefficient. The GRDREACH keyword is used to define the location of grid blocks that connect to a given river reach. Other information about how the river connects to the grid, such as the contact area and the productivity index (PI), is also specified with this keyword.

The river cross-section and the upstream/downstream boundary conditions to be applied are defined using RIVRXSEC, MASSFLOW and DEPTHTAB keywords in the PROPS section.

The keyword RIVRDIMS, which is specified in the RUNSPEC section, is used to describe the dimensions of the river data to be used in the run.

The River Model

The Saint-Venant Equations

In this section the ECLIPSE river model is outlined. The ECLIPSE river is a tool to handle coupling of surface river networks and subsurface groundwater flows. This is achieved by computing a solution to the one-dimensional Saint-Venant equations at each simulation time-step. It is possible to simulate either transient or steady-state behavior in the river. The steady-state option partially addresses the difference in time scale between the fast river channel dynamics and the slow groundwater dynamics, and focuses on the exchange of flow between the river and groundwater.

The following form of the Saint-Venant equations, with the flux Q and cross-sectional area A as independent variables, are solved

$$\frac{\partial A}{\partial t} + \frac{\partial Q}{\partial x} = E$$

$$\frac{\partial Q}{\partial t} + \frac{\partial}{\partial x}\left(\frac{Q^2}{A} + gI_1\right) = gI_2 + gA(S_0 - S_f)$$

where

Q is the flux across the cross sectional area A

E is the river exchange term

x is the distance along the river

t is time

A is the cross sectional area

g is the acceleration due to gravity

I_1 is the hydrostatic pressure force term given by $I_1 = \int_0^{h(x,t)} (h-\eta)b(x,\eta)d\eta$ where

$b(x,\eta) = \frac{\partial A(x,\eta)}{\partial \eta}$ and $h(x,t)$ is the water depth.

I_2 is the pressure force term due to longitudinal width variations given by

$$I_2 = \int_0^{h(x,t)} (h-\eta)\frac{\partial b(x,\eta)}{\partial x}d\eta$$

S_0 is the natural river slope $S_0 = \frac{\partial z}{\partial x}$

S_f is the frictional slope $S_f = \frac{Q|Q|n^2}{A^2\Pi^2R^{4/3}}$, where n is the Manning friction coefficient, R is the hydraulic radius and Π is a constant depending on the unit system being used.

There are three simplified forms of the Saint-Venant equations which may be used instead of the full time-dependent Saint-Venant equations for the river model. The first simplified form is the steady-state Saint-Venant equations:

$$\frac{\partial Q}{\partial x} = E$$

$$\frac{\partial}{\partial x}\left(\frac{Q^2}{A} + gI_1\right) = gI_2 + gA(S_0 - S_f)$$

the second simplified form which can be used is the diffusive wave form:

$$\frac{\partial Q}{\partial x} = E$$

$$\frac{\partial I_1}{\partial x} = I_2 + A[S_0 - S_f]$$

and the third simplified form is the kinematic wave form:

$$\frac{\partial Q}{\partial x} = E$$

$$S_0 - S_f = 0$$

These three sets of equations are derived from the full Saint-Venant equations under certain simplifying assumptions. The kinematic wave equations are derived by making the assumption that the hydrostatic pressure force term and the net momentum flux are constant along the length of the river. The diffusive wave equations are derived from the St. Venant equations by making the assumption that the net momentum flux of the river is constant along the length of the river. The steady-state Saint-Venant equations are derived assuming the solution is independent of time.

ECLIPSE river solves the river equations in a number of stages. The first is the pre-solve stage, where a simplified form of the equation is solved individually for each river reach by Newton iteration. This provides an initial estimate of the river solution for each river reach. During a second solution stage, ECLIPSE solves the entire river system (that is all rivers in a given model) by computing a converged implicit solution coupled with the groundwater system. The user can request to solve each river by specifying one of the four solution methods, which are based on solving either the kinematic, diffusive, steady-state or the full Saint-Venant system of equations.

Coupling of River and Groundwater Flow

The river model computes the flow of water between a river and the underlying groundwater using an infiltration equation. In ECLIPSE there is a choice of two infiltration equations which may be used. The two models will now be described.

Model one

The first model, which is the default, computes the flow E between the river and the groundwater in the underlying aquifer using the equation

$$E = J\alpha[P_{aquifer} - P_{river} - H]$$

where

J is the reach productivity index (PI)

α is the contact area between the river channel and the river grid connection

$P_{aquifer}$ is the pressure in the aquifer

P_{river} is the pressure in the river

H is the river pressure head term between the aquifer and river elevation.

The pressure in the river is calculated using the expression

$$P_{river} = P_{ref} + \rho_{sw} g h$$

where ρ_{sw} is the water surface density in the river and P_{ref} is a given reference pressure.

The pressure in the aquifer takes into account capillary pressure P_{cow} and is given by

$$P_{aquifer} = P - P_{cow} .$$

The pressure head term H is calculated using the expression

$$H = \rho g (Z_{aquifer} - Z_{river})$$

where

ρ is the density of the fluid in the aquifer

g is the acceleration due to gravity

$Z_{aquifer}$ is the elevation of the aquifer

Z_{river} is the elevation of the river.

When the aquifer is not fully saturated with water (the aquifer is unconfined) the fluid density ρ in the aquifer is calculated using the expression

$$\rho = \frac{\sum_i k_{ri} \rho_i}{\sum_i k_{ri}} \quad i = o, g, w$$

where k_{ri} is the relative permeability and ρ_i is the reservoir density of phase i, respectively. When the aquifer is confined, the fluid density is simply taken to be the water phase density. It should be noted that the infiltration equation for the unconfined case reduces to that of the confined case when the aquifer is fully saturated.

The contact area α between the river channel and the aquifer may be a function of the water depth in the river h, however when the contact area α is constant the reach productivity index (PI) J is related to the streambed conductance C as follows

$$J = \frac{C}{\rho g \alpha} .$$

The use of a streambed conductance implies the presence of a leaky streambed layer with a given thickness.

Model 2

The second model of infiltration uses the equation

$$E = C(H_{aquifer} - H_{river})$$

where

C is the river bed conductance

$H_{aquifer}$ is the hydraulic head in the aquifer

H_{river} is the hydraulic head in the river.

The term riverbed conductance is defined to be

$$C = \frac{KwL}{m}$$

where

K is the hydraulic conductivity of the river

w width of the river reach

L is the length of the river

m is the thickness of the riverbed deposits.

In this model the hydraulic head in the river is simply defined to be

$$H_{river} = Z_{ref} - Z_{river} + m + h$$

where

Z_{ref} is the reference elevation

Z_{river} is the elevation of the river bottom

h is the water height in the river.

The hydraulic head in the aquifer is calculated using the expression

$$H_{aquifer} = Z_{ref} - Z_{aquifer} + \frac{(P_{aquifer} - P_{ref})}{\rho_w g}$$

where

$Z_{aquifer}$ is the elevation of the aquifer

$P_{aquifer}$ is the pressure in the aquifer

P_{ref} is a given reference pressure

ρ_w is the water density in the aquifer

g is the acceleration due to gravity.

The flow between the river and aquifer is no longer dependent on the hydraulic head in the aquifer when the hydraulic head in the aquifer is below the river bed bottom elevation. In this case the hydraulic head in the aquifer is taken to be the river bed elevation.

Examples

Example 1: A Simple River

The following dataset extract outlines the keywords required to simulate a simple river in ECLIPSE 100.

This example shows how to implement a simple single river channel with upstream and downstream boundary conditions prescribed through the use of mass flow and depth tables respectively. Note that all the data input is in METRIC units.

The RIVRDIMS keyword describes the dimensions of the river data to be used in the run and needs to be added to the RUNSPEC section. In this example RIVRDIMS specifies that the model will include one river consisting a single branch which can be made up of a maximum of ten reaches. The number of reaches specified in the table should always be one greater than the number input

The RIVRDIMS keyword also specifies the maximum number of RIVRXSEC, RBEDCONT, DEPTHTAB, MASSFLOW and QHRATING tables that can be included in the run and the maximum number of rows of data expected in the tables. For example, in the current instance a maximum of one massflow table consisting of a maximum of two rows of data can be specified. It should be noted that the number of reaches per river should always be set one greater than the number of reaches input in the REACHES keyword as ECLIPSE always reserves the first reach for internal use.

```
RIVRDIMS
--RIVER REACH BRANCH GRID RIVRXSEC RBEDCONT DEPTHTAB MASSFLOW QHRATING
      1    10      1  10        1    2  1    1  2    1  2    0  2 /
```

The keyword RIVRXSEC is used to describe the cross-section of the river as a function of the depth and is parameterised by the depth, wetted perimeter and the cross-sectional area of the river. In this example the keyword has been used to describe a river with a simple box channel cross-section which is 10m wide and 5m high. The RBEDCONT keyword is used to define the contact area of a river with a grid block as a function of time. The MASSFLOW and DEPTHTAB tables are used to define the upstream massflow and downstream depth boundary conditions as functions of time. The following keywords need to be added to the PROPS section for this simple example:

```
RIVRXSEC
      0        10         0
    5.0      20.0      50.0
/
```

```
RBEDCONT
0.0   100.0
5.0   100.0
/
```

```
MASSFLOW
1  JAN  2007  00:00  10.0
4  MAR  2007  00:00  10.0
/
```

```
DEPTHTAB
1  JAN  2007  00:00  1.0
4  MAR  2007  00:00  1.0
/
```

In the SCHEDULE section the keyword RIVERSYS is used to name the river, prescribe the type of equation to be solved and define which type of boundary conditions will be imposed on the river. This keyword may alternatively be specified in the SOLUTION section. In the dataset extract below the river has been called 'MYRIVER' and it has been specified that the kinematic wave equations will be solved. The upstream and downstream boundary conditions will be prescribed as a function of time through the use of the keywords MASSFLOW and DEPTHTAB set in the PROPS section.

The general structure of the river is defined with the use of the keywords COMPRIV, REACHES and GRDREACH. The keyword COMPRIV defines how the river is connected to the grid. In this example, the keyword is used to define a straight line of nine grid blocks which are connected to the river. The REACHES keyword is then used to specify the reaches structure of the river. The position of the river mouth, the length of each reach and the depth change along the river reach along with some other parameters such as Manning's roughness coefficient are also set here for each river reach. The keyword GRDREACH is then finally used to define how the river reaches defined in REACHES are connected to the grid blocks defined in COMPRIV. Other parameters such as the productivity index (PI) are set here.

The following keywords need to be added to the SCHEDULE section for this simple example:

```
RIVERSYS
 'MYRIVER' KINEMATI /
 1 MYRIVER DOWNSTREAM DEPTH TABLE=1 UPSTREAM MASSFLOW TABLE=1 /
/
```

```
COMPRIV
 'MYRIVER'    5 9 1   /
 'MYRIVER'    5 8 1   /
 'MYRIVER'    5 7 1   /
 'MYRIVER'    5 6 1   /
 'MYRIVER'    5 5 1   /
 'MYRIVER'    5 4 1   /
 'MYRIVER'    5 3 1   /
 'MYRIVER'    5 2 1   /
 'MYRIVER'    5 1 1   /
/
```

```
REACHES
'MYRIVER' 450.0 900.0 0.0 0.0 INC /
2   2   1   MYRIVER  100  0.0   1   0.22e-01    2*  /
3   3   2   MYRIVER  100  0.0   1   0.22e-01    2*  /
4   4   3   MYRIVER  100  0.0   1   0.22e-01    2*  /
5   5   4   MYRIVER  100  0.0   1   0.22e-01    2*  /
6   6   5   MYRIVER  100  0.0   1   0.22e-01    2*  /
7   7   6   MYRIVER  100  0.0   1   0.22e-01    2*  /
8   8   7   MYRIVER  100  0.0   1   0.22e-01    2*  /
9   9   8   MYRIVER  100  0.0   1   0.22e-01    2*  /
10  10  9   MYRIVER  100  0.0   1   0.22e-01    2*  /
/
```

```
GRDREACH
'MYRIVER' /
5 9 1   MYRIVER   2*         1*  'J'    1*   100 0  1.2e-2 3* /
5 8 1   MYRIVER   2*         1*  'J'    1*   100 0  1.2e-2 3* /
5 7 1   MYRIVER   2*         1*  'J'    1*   100 0  1.1e-2 3* /
5 6 1   MYRIVER   2*         1*  'J'    1*   100 0  1.2e-2 3* /
5 5 1   MYRIVER   2*         1*  'J'    1*   100 0  1.2e-2 3* /
5 4 1   MYRIVER   2*         1*  'J'    1*   100 0  1.1e-2 3* /
5 3 1   MYRIVER   2*         1*  'J'    1*   100 0  1.2e-2 3* /
5 2 1   MYRIVER   2*         1*  'J'    1*   100 0  1.2e-2 3* /
5 1 1   MYRIVER   2*         1*  'J'    1*   100 0  1.1e-2 3* /
/
```

In the summary section it is possible to output time-dependent information about the river. In this example the water depth (`SRRDEPTH`), exchange flow (`SRREXCH`) and Froude number (`SRRFRODE`) are output for the fifth reach of '`MYRIVER`'. The branch flow (`SRBQR`) is also output for the first branch in the river.

```
-- River reach water depth
SRRDEPTH
 MYRIVER 5 /
/
-- River reach exchange flow rate
SRREXCH
 MYRIVER 5 /
/
-- River reach Froude number
SRRFRODE
 MYRIVER 5 /
/
-- Branch exchange flow over time-step
SRBQR
 MYRIVER 1 /
/
```

Rock Compaction

Chapter 58

Introduction

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

By default ECLIPSE models rock compaction as a simple compressibility, which is entered using the ROCK keyword. Additional options are available to model more complex behavior:

1. A tabular description of the compaction as a function of pressure, which can be either reversible or irreversible ("Tabulated compaction curves" on page 860)
2. The ability to modify the transmissibility as a function of pressure
3. A hysteretic model to allow partial reflation

ECLIPSE 300

4. A cyclic hysteretic model to allow compaction and dilation

ECLIPSE 100

5. A water-induced compaction model ("Water induced compaction (ECLIPSE 100)" on page 864)
6. The Palmer-Mansoori rock model (Coal Bed Methane option only) ("Palmer-Mansoori rock model" on page 865)

ECLIPSE 300

7. The ability to modify the transmissibility as a function of porosity.

Tabulated compaction curves

The compaction of the rock can be modeled as a general function of pressure using the ROCKTAB keyword. In the case where a grid block pressure increases, the compaction can either be assumed to be reversible or irreversible.

In some cases these two reflation options (fully reversible or irreversible) may be too crude a description. A hysteretic option is available by using the ROCKTABH keyword. In this case a set of reflation curves can be input to describe the reversible reflation behavior from each starting pressure. For pressures below the starting pressure, the behavior is only partially reversible, given by the deflation curve.

Figure 58.1 Rock Compaction: Hysteresis option

A second hysteretic option can be used to model cyclic dilation and recompaction.This model is based on Beattie, Boberg and McNab (1991). The ROCKTABH keyword is again used to input curves that describe the reversible (or elastic) behavior. The reversible region is bounded at low pressures by the deflation (or compaction) curve and at high pressures by a dilation curve which describes partially reversible expansion. This option can be selected using the ROCKCOMP keyword.

Figure 58.2 Rock Compaction: Boberg Hysteresis option

Initially it is assumed the reservoir rock is fully compacted. The initial pore volume and transmissibility multipliers will be set to the deflation curve value at the cell pressure. In general these multipliers will not be equal to one, so the initial pore volume and transmissibility will differ from the values entered in the data file. Alternatively, the rock compaction multipliers can be tabulated as functions of pressure change (ΔP), and the ROCKOPTS keyword can be used to store the initial pressure (see below). The initial multipliers will then be set to the deflation curve values at $\Delta P = 0$, which can be set to 1.

If the initial reservoir rock is not fully compacted, pore volume and transmissibility values can be set by finding an elastic curve with pore volume multiplier equal to one at the initial cell pressure. In this case the pore volume will be the same as the value entered in the data file, although the transmissibility may still differ. This option can be selected using the ROCKOPTS keyword.

Figure 58.3 Initial multipliers

Transmissibility modification

In addition to the compaction (the reduction in the pore volume), it is possible to modify the transmissibility as a function of pressure.

At present the transmissibility modification is treated explicitly, that is the modification due to a change in pressure only affects the subsequent time-step.

Overburden pressure

A reservoir simulator typically works in terms of pressure, as pressure is the natural variable when considering the flow. However, it is often better to consider the compaction as a function of the effective stress on the rock, which is the overburden pressure minus the fluid pressure. The ROCKTAB, ROCKTABH or ROCK2D and ROCK2DTR keywords tabulate the compaction multipliers against the effective fluid pressure instead $P_{eff\,fl} = P_{fluid} - \text{Overburden}$.

The overburden pressure can be input into ECLIPSE as a function of depth. Note that the overburden is assumed to remain constant through time.

Typically the overburden pressure will be greater than the fluid pressure. In this case it is convenient to tabulate the compaction against the effective stress rather than the pressure. This can be achieved by using the ROCKOPTS keyword. In this case the pore volume and transmissibility multipliers are tabulated against the effective overburden pressure, $P_{eff\,ov} = \text{Overburden} - P_{fluid}$, which is $-P_{eff\,fl}$.

Reservoir pore volumes

There is scope for confusion when supplying porosities or pore volumes to ECLIPSE. These could either be at reference conditions (derived from core data); or at reservoir conditions (derived from log data).

ECLIPSE treats the input porosities (PORO) and pore volumes (PORV) as at reference conditions. With the rock compressibility model activated by the ROCK keyword, the pore volumes are then referenced at the reference pressure in the ROCK keyword. It should be borne in mind that other simulators will assume the geometric PORV represents reservoir pore volumes. The input PORV can be equivalenced to the initial reservoir pore volume in ECLIPSE by setting the second item of the ROCKOPTS keyword to 'STORE'.

With the rock compaction model (keywords ROCKTAB, ROCKTABH), setting the second item in ROCKOPTS to 'STORE' and tabulating the ROCKTAB and ROCKTABH tables against effective pressure (which is zero at equilibrated conditions) treats the initial PORO and PORV as reservoir conditions.

Transmissibility as a function of porosity

In order to define a model which can be used to specify the dependence of transmissibility on porosity, refer to items 4 and 5 of the ROCKCOMP keyword. There are two different options which can be used:

1 EXP Transmissibility is as exponential function of porosity:

$$\frac{\kappa}{\kappa_0} = \exp\left[\kappa_{mult}\left(\frac{\phi - \phi_0}{1 - \phi_0}\right)\right]$$

where κ_0 is the initial transmissibility, ϕ_0 is the initial porosity and κ_{mult} is the user-defined multiplier factor defined by keywords, ROCKTRMX, ROCKTRMY and ROCKTRMZ for a particular direction.

2 CZ Transmissibility is a function of porosity via the Carmen-Kozeny type formula:

$$\frac{\kappa}{\kappa_0} = \frac{\phi}{\phi_0}^{\tau\left[\left(\frac{1-\phi_0}{1-\phi}\right)\right]}$$

where τ is an integer defined in item 5 of the ROCKCOMP keyword.

Items 4 and 5 of the ROCKCOMP keyword are available for use with the ROCKTAB and ROCKTABH keywords. They are currently not available for use with the ROCK keyword.

Output

Both the modified pore volumes and the transmissibilities can be output the Print and RESTART files.

ECLIPSE 100

In ECLIPSE 100, the output is controlled by the mnemonic ROCKC in keywords RPTSCHED and RPTRST.

If we make the assumption that the compaction occurs in the vertical direction only, the compaction can be interpreted as a change in the height of the reservoir. When the hysteretic compaction option is used, the modified reservoir thickness can also be output.

The mnemonics used in the print and RESTART files are:

PRES-OVB Overburden pressure

PRES-EFF Effective fluid pressure

PEFF-MIN Effective minimum pressure for hysteretic compaction for ROCKTABH

PORV-MOD Multiplier on pore volumes for ROCKTAB

TRAN-MOD Multiplier on transmissibilities for ROCKTAB and ROCKTABH

DZ Modified DZ, assuming compaction is vertical for ROCKTAB and ROCKTABH

DZ-TOTAL Total modified DZ for each column of cells in the areal plane for ROCKTAB or ROCKTABH

DZ-COMP Total compacted DZ for each column of cells in the areal plane for ROCKTABH only

DZ-TOTM Total modified DZ for each column of cells in the areal plane for ROCKTABH only for the matrix cells in a dual porosity run only

DZ-COMPM Total compacted DZ for each column of cells in the areal plane for ROCKTABH only for the matrix cells in a dual porosity run only

DZ-TOTF Total modified DZ for each column of cells in the areal plane for ROCKTABH only for the matrix cells in a dual porosity run only

DZ-COMPF Total compacted DZ for each column of cells in the areal plane for ROCKTABH only for the fracture cells in a dual porosity run only

SWAT-EFF Effective water saturation for the water induced compaction model.

ECLIPSE 300

In ECLIPSE 300, the output is controlled by the RPTSOL, RPTSCHED and RPTRST keywords. The arrays are output with the name of the mnemonic used to request the output.

Available mnemonics are:

PRES_OVB Overburden pressure

PRES_EFF Effective fluid pressure for ROCKTAB or ROCKTABH

PORV_MOD Multiplier on pore volumes for ROCKTAB or ROCKTABH

PRESMIN Minimum pressure for rock compaction hysteresis with ROCKTABH.

Water induced compaction (ECLIPSE 100)

Most rocks will compact to some extent as the stress on the rock is increased, when the fluid pressure falls. Some rocks, typically chalks, will exhibit additional compaction when water contacts oil bearing rock, even at constant stress. This behavior could have a significant effect on the performance of a water flood as it adds significant energy to the reservoir.

The water induced compaction option allows you to model this effect by supplying tables making the pore volume a function of both pressure and water saturation. The option allows two methods of data input:

1 Two tables of pore volume multipliers, one as a function of pressure and the other as a function of water saturation. The pore volume modification is then the product of the two multipliers:

$$V = V_G \cdot M_p(P) \cdot M_s(Sw_{max} - Sw_{ini})$$

where

V is the pore volume

V_G is the geometric pore volume

$M_p(P)$ is the pore volume multiplier as a function of pressure

$M_s(S)$ is the pore volume multiplier as a function of water saturation

Sw_{max} is the maximum water saturation attained by the grid block

Sw_{ini} is the initial water saturation.

The pore volume multipliers are taken from the ROCKTAB/ROCKTABH and ROCKTABW keywords.

2 A full two-dimensional table of pore volume multipliers as a function of both pressure and water saturation:

$$V = V_G \cdot M(P, Sw_{max} - Sw_{ini})$$

Where $M(P, S)$ is a two-dimensional table entered using the ROCK2D and ROCKWNOD keywords.

The transmissibility modification is handled in a similar manner. If method 1 is used to input the compaction data, the transmissibility modification is the product of the pressure and saturation multipliers. If method 2 is used, transmissibility modifiers can be entered using an analogous two dimensional table with the ROCK2DTR keyword.

Palmer-Mansoori rock model

Note This is only available in the Coal Bed Methane option (see the COAL keyword).

The Palmer-Mansoori rock model([Ref. 109]) adjusts the pore volumes, $V(P)$, at the current pressure, P, using

$$V(P) = V(P_0)\left\{1 + \frac{c_m}{\phi_0}(P - P_0) + \frac{\varepsilon_l}{\phi_0}\left(\frac{K}{M} - 1\right)\left(\frac{\beta P}{1 + \beta P} - \frac{\beta P_0}{1 + \beta P_0}\right)\right\}$$

where:

ϕ_0 is the initial porosity

P_0 is the initial pressure

$V(P_0)$ is the pore volume at the initial pressure

K is the bulk modulus

M is the constrained axial modulus

γ is the grain compressibility

β, ε_l are parameters of the match between the Langmuir curve and the change in volumetric strain due to matrix shrinkage

f is a fraction between 0 and 1

and

$$c_m = \frac{1}{M} - \left[\frac{K}{M} + f - 1\right]\gamma$$

Following Palmer-Mansoori, assuming that permeability varies with porosity as

$$\frac{k}{k_0} = \left(\frac{\phi}{\phi_0}\right)^n$$

where the exponent n is typically 3, then the transmissibility multiplier is taken as

$$\left\{1 + \frac{c_m}{\phi_0}(P - P_0) + \frac{\varepsilon_l}{\phi_0}\left(\frac{K}{M} - 1\right)\left(\frac{\beta P}{1 + \beta P} - \frac{\beta P_0}{1 + \beta P_0}\right)\right\}^n$$

This model is activated using the 'PALM-MAN' option on item 1 of the RUNSPEC section keyword ROCKCOMP. The parameters for the model are set in the PROPS section keyword ROCKPAMA.

ECLIPSE 300

Cells not connected to coal cells only apply the compression term to the pore volume and permeability multipliers.

Compositional coal swelling/shrinkage model (ECLIPSE 300)

A modified form of the Palmer-Mansoori rock model is available by using the ROCKPAME keyword. A similar approach is described in [Ref. 119]. An important difference is that the model described here will not predict volume changes for undersaturated coal. Each component of the system can be assigned Langmuir curve parameters β_k, ε_k, making the swelling or shrinkage of the coal dependent on composition. The strain of one component is computed as if the coal gas is instantly in equilibrium following the extended Langmuir formula for adsorption capacity

$$e_k = \frac{\varepsilon_k \beta_k a_k P}{1 + \sum \beta_j a_j P}$$

where $a_k = L_k / \left(\sum L_j\right)$ represent the adsorbed mole fraction of component k, P the pressure and L_k are the adsorption capacities of each component (see keyword LANGMEXT).

The total strain is then computed as

$$E = \sum e_k$$

Using this in the Palmer-Mansoori equation the pore volume, $V(P, Y)$, at the current pressure, P, and gas phase composition $Y = (y_1, y_2, \ldots, y_{nc})$ are adjusted using

$$V(P, Y) = V_0 \left\{ 1 + \frac{c_m}{\phi_0}(P - P_0) + \frac{1}{\phi_0}\left(\frac{K}{M} - 1\right)(E - E_0) \right\}$$

where:

ϕ_0 is the initial porosity

P_0 is the initial pressure

V_0 is the pore volume at the initial pressure and composition

K is the bulk modulus

M is the constrained axial modulus

γ is the grain compressibility

β_k, ε_k are parameters of the match between the Langmuir curve and the change in volumetric strain due to matrix shrinkage for component k.

f is a fraction between 0 and 1

The term c_m is given as

$$c_m = \frac{1}{M} - \left[\frac{K}{M} + f - 1\right]\gamma .$$

For undersaturated coal where the adsorbed gas is not changing the formula is modified so the strain is constant. The strain used then corresponds to the state where the gas would start to desorb.

Note also that the ROCKPAME keyword only apply to regions with cells connected to coal cells as assigned by COALNUM.

Note The rock compaction models takes the input porosities as the porosities at reservoir conditions. In areas without coal different rock compaction models can be used but some care must be taken so the input porosities are treated consistently for the entire model.

Saturation Functions

Chapter 59

Introduction

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

Saturation functions such as relative permeabilities and capillary pressures are entered as tables (keywords SWFN, SGFN, SOF2 and SOF3 (or SWF3, SGF3 and SOF3) or the alternative family SWOF, SGOF and SLGOF). Saturation values need not be specified at equal intervals, and are used in the specified form, rather than being interpolated onto regular intervals.

All the saturation functions may be entered in multi-table form; tables are associated with cells using the SATNUM keyword and its hysteresis counterparts.

There are two families of keywords that define the saturation-dependent properties of the reservoir fluids. The difference between them is that the first family allows you to enter the oil relative permeabilities in the same tables as the water and gas relative permeabilities, while in the second family the oil relative permeabilities must be entered in a separate table versus oil saturation.

Either family can be used; however, keywords from different families must not be mixed in the same run.

ECLIPSE 100

Note If running the ECLIPSE 100 Miscible Flood or Solvent options
OR
In 2-phase runs with no oil (that is gas-water runs),
family (ii) must be used.

The keyword families are described below:

Family (i)

The keywords are:

Table 59.1 Family (i)

Keyword	Description
SWOF	Sets water relative permeability, oil relative permeability in water, and water-oil capillary pressure as a function of water saturation. This is required for 2 or 3 phase systems with both water and oil.
SGOF	Sets gas relative permeability, oil relative permeability in gas at connate water, and oil-gas capillary pressure as a function of gas saturation. Either SGOF or SLGOF is required for 2 or 3 phase systems with both gas and oil.
SLGOF	Sets gas relative permeability, oil relative permeability in gas at connate water, and oil-gas capillary pressure as a function of liquid saturation. Either SGOF or SLGOF is required for 2 or 3 phase systems with both gas and oil.

Family (ii)

The keywords are:

Table 59.2 Family (ii)

Simulator	Keyword	Description
	SWFN	Sets water relative permeability and capillary pressure as a function of water saturation. This is required for 2 or 3 phase systems with water. For 3 phase systems keywords SWF32D and PCW32D can be used as an alternative to enter two-dimensional tables for the water relative permeability and oil/water capillary pressure (if non-zero) respectively.
ECLIPSE 300 only	SWF3	Sets water relative permeability to oil and gas and capillary pressure as a function of water saturation. This is required for 3 phase systems which use the IKU3P (see IKU3P keyword) or the ODD3P (see the ODD3P keyword) relative permeability models.
	SGFN	Sets gas relative permeability and capillary pressure as a function of gas saturation. This is required for 2 or 3 phase systems with gas. For 3 phase systems keywords SGF32D and PCG32D can be used as an alternative to enter two-dimensional tables for the gas relative permeability and gas/oil capillary pressure (if non-zero) respectively.

Table 59.2 Family (ii)

Simulator	Keyword	Description
ECLIPSE 300 only	SGF3	Sets gas relative permeability to oil and water and capillary pressure as a function of gas saturation. This is required for 3 phase systems which use the IKU3P (see IKU3P keyword) or the ODD3P (see the ODD3P keyword) relative permeability models.
	SOF3	Sets relative permeability of oil in water, and oil in gas at the connate water saturation, as a function of oil saturation. Either SOF3 or SOF32D are required for 3 phase systems.
	SOF2	Sets oil relative permeability as a function of oil saturation. This is required for 2 phase systems with oil.
	SOF32D	Sets the oil relative permeability in a 3-phase system as a function of both water saturation and gas saturation, in a 2-dimensional table. This may be used as an alternative to SOF3 in 3-phase runs. (This keyword may also be used in combination with the Family (i) keywords. It will override the oil relative permeability data supplied in SWOF and SGOF or SLGOF.)
ECLIPSE 100 only	SGWFN	Sets the gas and water relative permeabilities and the gas-water capillary pressure for two-phase systems with only gas and water present. (This keyword is optional, as the gas and water saturation functions could alternatively be supplied to ECLIPSE through the SWFN and SGFN keywords, where the water capillary pressure supplied in SWFN represents the true gas-water capillary pressure and the capillary pressure is set to zero in the SGFN table.)
ECLIPSE 100 only	SWF32D	Sets the water relative permeability in a 3-phase system as a function of both **oil** saturation and **gas** saturation, in a 2-dimensional table. This may be used as an alternative to SWFN in 3-phase runs. If oil/water capillary pressure is required this keyword has to be used in conjunction with PCW32D.
ECLIPSE 100 only	SGF32D	Sets the gas relative permeability in a 3-phase system as a function of both **oil** saturation and **water** saturation, in a 2-dimensional table. This may be used as an alternative to SGFN in 3-phase runs. If oil/gas capillary pressure is required this keyword has to be used in conjunction with PCG32D.
ECLIPSE 100 only	PCW32D	Sets the water/oil capillary pressure in a 3-phase system as a function of both **oil** saturation and **gas** saturation, in a 2-dimensional table. Can be used in conjunction with SWF32D.
ECLIPSE 100 only	PCG32D	Sets the gas/oil capillary pressure in a 3-phase system as a function of both **oil** saturation and **water** saturation, in a 2-dimensional table. Can be used in conjunction with SGF32D.

Note Default values represented by 1* may be inserted as required in one or more rows for columns 2 onwards for the one-dimensional table keywords SWFN, SWF3, SWOF, SGFN, SGWFN, SGF3, SGOF, SLGOF SOF2 and SOF3. Where default values have been entered, the actual values will be calculated by linear interpolation using the nearest non-default neighbors and substituted into the table. However, where only one non-default value is entered in an entire column or where default values are entered at the start or end of the tables, the interpolation behavior for ECLIPSE 100 and 300 can differ. For example, for the SGOF keyword, if the capillary pressure column is defaulted throughout, the values are replaced with 0.0 throughout by both simulators. However, if only one value is not defaulted and this is not the first value in the table, in ECLIPSE 100, the first value is set to 0.0, linear interpolation is performed to replace any default values up to the defined value, and all subsequent values are assigned to the defined value. In contrast, in ECLIPSE 300, all defaulted values in the column are set equal to the single non-defaulted value. It is recommended, therefore, that when defaults are entered into a table, the table is reported using the RPTPROPS keyword and the resultant values inspected to verify the intended interpolation behavior.

Water saturation properties

An example data table for the SWFN keyword is given below. The data for the SWOF keyword is similar but with an additional column containing oil relative permeabilities.

```
-- SWAT KRW PCOW
--          (PSIA)
SWFN
  .22  .0   7.0
  .3   .07  4.0
  .4   .15  3.0
  .5   .24  2.5
  .6   .33  2.0
  .8   .65  1.0
  .9   .83   .5
 1.0  1.0    .0   /
```

The water saturations must be entered in ascending order.

The third column is the oil-water capillary pressure ($P_{cow} = P_o - P_w$). The values can become negative with increasing water saturation, if required. They should all be set to zero if there is no water-oil capillary pressure. In saturation tables intended for well completions only (that is, not referenced for any grid block in keyword SATNUM) the capillary pressure data is not used, and the values entered are disregarded. In gas-water problems, the water capillary pressure is interpreted as the gas-water capillary pressure ($P_{cgw} = P_g - P_w$).

Three saturation values in the table are of special interest:

S_{wcr} The **critical** water saturation.

This is the highest saturation value for which the relative permeability is zero. At saturations above this value, water is mobile. Note that a critical water saturation **must** be defined (the SWFN or SWOF table must contain a water saturation for which $k_{rw} = 0$). In the example above, $S_{wcr} = 0.22$.

S_{wco} The minimum water saturation value in the table.

The Equilibration calculation (see keyword EQUIL) sets the water saturation to this value in grid blocks that lie above the water contact (or water transition zone). In this respect S_{wco} is the **connate** water saturation. In the example shown above, $S_{wco} = S_{wcr}$. If for any reason the connate water saturation is less than the critical value, the table should begin with the connate value. For example for $S_{wco} = 0.2$ and $S_{wcr} = 0.22$, the table should begin with the lines

```
 .20  .0   7.0
 .22  .0   7.0
 .3   .07  4.0
etc.
```

(Note that ECLIPSE can handle zero gradients in the capillary pressure vs. saturation, as shown in the above example between the S_w values 0.2 & 0.22.)

$S_{w\ \max}$ The maximum water saturation value in the table.

The equilibration calculation sets the water saturation to this value in grid blocks that lie below the water transition zone.

In the example shown above, $S_{w\ \max} = 1.0$, which results in the water zone being fully saturated with water.

In an **oil-wet system**, in order to use correctly the Killough wetting phase model on the oil phase (option 7, Item 2 of `EHYSTR` keyword), $S_{w\ \max}$ should equal $1.0 - S_{oco}$, S_{oco} being the connate oil saturation.

Alternatively, a full 2-dimensional table of water relative permeabilities vs. oil and gas saturations can be entered in 3-phase runs using the `SWF32D` keyword. Keyword `PCW32D` can be used in conjunction to `SWF32D` to enter the water/oil capillary pressure if required.

When using `SWF32D` the connate water is defined as $S_{wco} = 1. - S_{o,max}$.

Gas saturation properties

An example data table for the SGFN keyword is given below. The data for the SGOF keyword is similar but with an additional column containing oil relative permeabilities. The data for the SLGOF keyword is similar to the SGOF keyword, except that the saturation column contains liquid saturation values in ascending order.

```
-- SGAS   KRG   PCOG
--              (PSIA)
SGFN
    .0     .0    0.0
    .04    .0     .2
    .1     .022   .5
    .2     .1    1.0
    .3     .24   1.5
    .4     .34   2.0
    .5     .42   2.5
    .6     .5    3.0
    .7     .81   3.5
    .78   1.0    3.9    /
```

The gas saturations must be entered in ascending order.

The third column is the oil-gas capillary pressure ($P_{cog} = P_g - P_o$). The values can start negative if required. They should all be set to zero if there is no gas-oil capillary pressure. In saturation tables intended for well completions only (that is not referenced for any grid block in keyword SATNUM) the capillary pressure data is not used, and the values entered are disregarded. In gas-water problems, the gas-water capillary pressure is defined in the water saturation table, so the capillary pressure values in the gas saturation table should be set to zero.

Alternatively, a full 2-dimensional table of water relative permeabilities vs. oil and gas saturations can be entered using the SGF32D keyword, in 3-phase runs. Keyword PCG32D can be used in conjunction with SWF32D to enter the gas/oil capillary pressure if required.

Three saturation values in the table are of special interest:

S_{gcr} The **critical** gas saturation.

This is the highest saturation value for which the relative permeability is zero. At saturations above this value, gas is mobile. Note that a critical gas saturation **must** be defined (the SGFN or SGOF {or SLGOF} table must contain a gas {or liquid} saturation for which $k_{rg} = 0$). In the example above, $S_{gcr} = 0.04$.

S_{gco} The minimum gas saturation value in the table.

The Equilibration calculation (see keyword EQUIL) sets the gas saturation to this value in grid blocks that lie below the gas contact (or water transition zone). In this respect S_{gco} is the **connate** gas saturation. Normally $S_{gco} = 0.0$, as in the example above.

$S_{g\,\max}$ The maximum gas saturation value in the table.

The equilibration calculation sets the gas saturation to this value in grid blocks that lie above the gas transition zone. Normally $S_{g\ \max} = 1.0 - S_{wco}$, as in the example above, with $S_{wco} = 0.22$ as in the first SWFN table example in this chapter.

In an **oil-wet system**, in order to use correctly the Killough wetting phase model on the oil phase (option 7, Item 2 of EHYSTR keyword), $S_{g\ \max}$ should equal $1.0 - S_{oco}$, S_{oco} being the connate oil saturation.

Oil saturation properties

The relative permeabilities of oil in water, and oil in gas and connate water, are entered either in keywords SWOF and SGOF (or SLGOF), or with keyword SOF3 (or SOF2 in a 2-phase run).

Alternatively, a full 2-dimensional table of oil relative permeabilities vs. water and gas saturations can be input using the SOF32D keyword, in 3-phase runs.

The relative permeability tables are not used in the calculation of the initial saturations. The ECLIPSE equilibration calculation sets oil saturations using the formula $S_o = 1 - S_w - S_g$.

An example data table for keyword SOF3 is given below. If Family (i) were being used, the k_{row} data would be included in keyword SWOF and the k_{rog} data would be included in either keyword SGOF or SLGOF.

```
-- SOIL   KROW     KROG
SOF3
  .0000    .0000    .0000
  .2000    .0000    .0000
  .3800   1*        .0000
  .4000    .0050   1*
  .4800   1*       .0200
  .5000    .0649   1*
  .5800   1*        .1000
  .6000    .1250   1*
  .6800   1*        .3300
  .7000    .4000   1*
  .7400   1*        .6000
  .7800   1.0000   1.0000     /
```

The oil saturations must be entered in ascending order in column 1. Columns 2 and 3 contain the corresponding oil relative permeabilities for oil-water systems, and for oil-gas-connate water systems respectively. Columns 2 and 3 may contain default specifications (represented by 1*). When the table is read in, defaults are replaced by values obtained by linear interpolation (in the above example, the first default in column 2 is replaced by the value 0.0045).

Five saturation values in the table are of special interest:

S_{oco} The minimum oil saturation value in the table.

This is the **connate** oil saturation. Normally, $S_{oco} = 0.0$, as in the example above. This value is particularly used in **oil-wet system** modeling, applying the Killough wetting model to the oil phase (option 7, Item 2 of EHYSTR keyword), to calculate both $S_{g\,\max}$ and $S_{w\,\max}$. In these cases, normally S_{oco} is a non-zero value.

S_{ocr} The **critical** oil saturation.

This is the highest saturation value for which **both** the oil-water and oil-gas permeabilities are zero. At saturations above this value, oil is mobile in the three-phase region. Note that a critical oil saturation **must** be defined (the SOF3 table must contain an oil saturation for which both k_{row} and k_{rog} are 0). In the example above, $S_{ocr} = 0.20$.

S_{orw} The Residual Oil Saturation in the oil-water system.

This is the saturation at which the oil relative permeability in the oil-water system becomes zero (S_o at which $k_{row} = 0$). In the example above, $S_{orw} = 0.20$.

S_{org} The Residual Oil Saturation in the gas-oil system.

This is the saturation at which the oil relative permeability in the gas-oil system becomes zero (S_o at which $k_{rog} = 0$). In the example above, $S_{org} = 0.38$.

$S_{o\ \max}$ The maximum oil saturation value in the table.

This should be equal to $1 - S_{wco}$ as determined from the SWFN table. The two oil relative permeabilities should be the same at $S_{o\ \max}$ (both represent a case with $S_o = S_{o\ \max}$, $S_w = S_{wco}$ and $S_g = 0$). ECLIPSE reports an error if this condition is not satisfied.

In an **oil-wet system**, normally $S_{o\ \max} = 1.0$.

If a two-dimensional table of oil relative permeability as a function of water saturation and gas saturation is required, this is entered using the SOF32D keyword. This keyword replaces SOF3, or overrides the oil relative permeability data supplied in the SWOF and SGOF or SLGOF keywords. An example of the SOF32D keyword is given below:

```
SOF32D
--      ------------------------------SWAT--------------------------------
        0.22  0.27  0.32  0.37  0.42  0.47  0.52  0.57  0.72  0.77  0.78 /
--SGAS
 0.00   1.000 0.625 0.345 0.207 0.113 0.083 0.053 0.023 0.002 0.001 0.000/
 0.05   0.555 0.337 0.210 0.110 0.078 0.047 0.021 0.004 0.001 0.000 /
 0.10   0.330 0.212 0.106 0.074 0.042 0.019 0.003 0.002 0.000 /
 0.15   0.215 0.103 0.069 0.036 0.017 0.003 0.002 0.001 0.000 /
 0.20   0.100 0.065 0.031 0.015 0.002 0.002 0.001 0.000 /
 0.25   0.060 0.025 0.014 0.002 0.001 0.001 0.000 /
 0.30   0.020 0.012 0.001 0.001 0.001 0.000 /
 0.35   0.010 0.001 0.001 0.001 0.000 /
 0.40   0.000 0.000 0.000 0.000 /
/
```

The first record contains the water saturation values, and the first column of the subsequent records contains the gas saturation values. At present the SOF32D keyword cannot be used with the End-point Scaling, Hysteresis and Miscible options.

Three phase oil relative permeability models

A choice of four different formulae are available for calculating the 3-phase oil relative permeability at particular water and gas saturations, from the input relative permeabilities of oil in water and oil in gas and connate water. The formulae are not needed, of course, if the SOF32D keyword has been used to input the 3-phase oil relative permeability as a 2-dimensional table vs. water and gas saturations (see above).

The default model for the 3-phase oil relative permeability is based on an assumption of complete segregation of the water and gas within each grid cell. The model provides a simple but effective formula which avoids the problems associated with other methods (poor conditioning, negative values etc.). ECLIPSE also contains options for using either of the modified three phase oil relative permeability models suggested by Stone. These models are summarized in [Ref. 18].

ECLIPSE 300

Further methods referred to as the IKU and the ODD3P methods are also available.

The default model assumed by ECLIPSE is shown in Figure 59.1. The oil saturation is assumed to be constant and equal to the block average value, S_o, throughout the cell. The gas and water are assumed to be completely segregated, except that the water saturation in the gas zone is equal to the connate saturation, S_{wco}. The full breakdown, assuming the block average saturations are S_o, S_w and S_g (with $S_o + S_w + S_g = 1$) is as follows:

In a fraction $S_g/(S_g + S_w - S_{wco})$ of the cell (the gas zone),

the oil saturation is S_o

the water saturation is S_{wco}

the gas saturation is $S_g + S_w - S_{wco}$

In a fraction $(S_w - S_{wco})/(S_g + S_w - S_{wco})$ of the cell (the water zone),

the oil saturation is S_o

the water saturation is $S_g + S_w$

the gas saturation is 0

The oil relative permeability is then given by

$$k_{ro} = \frac{S_g k_{rog} + (S_w - S_{wco}) k_{row}}{S_g + S_w - S_{wco}} \qquad \text{[EQ 59.1]}$$

where

k_{rog} is the oil relative permeability for a system with oil, gas and connate water (tabulated as a function of S_o)

k_{row} is the oil relative permeability for a system with oil and water only (also tabulated as a function of S_o)

Figure 59.1 The default 3-phase oil relative permeability model assumed by ECLIPSE

Stone's first model (Modified)

The second model available in ECLIPSE for calculating values of the three phase oil relative permeability is a modified version of the first model suggested by Stone [Ref. 42] and is available with keyword `STONE1` in the `PROPS` section. The formula is

$$k_{ro} = k_{rocw} SS_o F_w F_g \qquad \text{[EQ 59.2]}$$

where

k_{rocw} is the value of the oil relative permeability in the presence of connate water only

$SS_o = (S_o - S_{om})/(1 - S_{wco} - S_{om})$ when $S_o > S_{om}$

$F_w = k_{row}/(k_{rocw} \cdot (1 - SS_w))$

$F_g = k_{rog}/(k_{rocw} \cdot (1 - SS_g))$

where

$SS_w = (S_w - S_{wco})/(1 - S_{wco} - S_{om})$ when $S_w > S_{wco}$

$SS_g = S_g/(1 - S_{wco} - S_{om})$

In these formulae S_o, S_w and S_g denote block averaged values for the oil, water and gas saturations in a grid cell. k_{rog} denotes the oil relative permeability for a system with oil, gas and connate water, and k_{ro_w} denotes the oil relative permeability for a system with oil and water only. Both two phase oil relative permeability functions are tabulated as functions of oil saturation in the input data. k_{rocw} denotes the oil relative permeability in the presence of connate water only.

S_{om} is the minimum residual oil saturation. By default S_{om} is taken to be the minimum of the critical oil-to-water saturation and the critical oil-to-gas saturation, $min(S_{owcr}, S_{ogcr})$.

Alternatively, S_{om} can be input either as a function of water saturation (SOMWAT keyword) or as a function of gas saturation (SOMGAS keyword).

If the SOMWAT keyword is used, S_{om} is interpolated from the water saturation as shown in Figure 59.2:

Figure 59.2 Calculating the minimum residual oil saturation from the water saturation

If the SOMGAS keyword is used, S_{om} is interpolated from the gas saturation as shown in Figure 59.3:

Figure 59.3 Calculating the minimum residual oil saturation from the gas saturation

If the Hysteresis option is active, the drainage and imbibition S_{om} values are taken from the SATNUM and IMBNUM tables. For example in the SOMWAT case:

Figure 59.4 Calculating the minimum residual oil saturation for drainage and imbibition with hysteresis

The S_{om} value (point x in the picture above) is calculated as follows:

$$S_{om} = S_{om}^{(i)} + F \cdot (S_{om}^{(d)} - S_{om}^{(i)}) \qquad \text{[EQ 59.3]}$$

$$F = F_{sw} F_w + (1 - F_{sw}) F_g \qquad \text{[EQ 59.4]}$$

$$F_{sw} = \frac{S_w - S_{wco}}{1 - S_{wco} - S_{owcr}^{(h)}} \qquad \text{[EQ 59.5]}$$

$$F_w = \frac{S_{owcr}^{(i)} - S_{owcr}^{(h)}}{S_{owcr}^{(i)} - S_{owcr}^{(d)}}$$

$$F_g = \frac{S_{ogcr}^{(i)} - S_{ogcr}^{(h)}}{S_{ogcr}^{(i)} - S_{ogcr}^{(d)}} \qquad \text{[EQ 59.6]}$$

$S_{owcr}^{(i)}$ is the critical oil-to-water imbibition

$S_{owcr}^{(d)}$ is the critical oil-to-water drainage

$S_{owcr}^{(h)}$ is the critical oil-to-water actual

$S_{ogcr}^{(i)}$ is the critical oil-to-gas imbibition

$S_{ogcr}^{(d)}$ is the critical oil-to-gas drainage

$S_{ogcr}^{(h)}$ is the critical oil-to-gas actual

The calculation of three phase oil relative permeability using Stone's method 1 is enabled by means of the STONE1 keyword in the PROPS section of the input data. Tabulated output of the three phase oil relative permeability values is controlled by the mnemonic SOF in the RPTPROPS keyword.

ECLIPSE 300

If STONEPAR is used, this is obtained as a function of the A and B parameters:

$$S_{om} = S_{ogcr}X_g^A + S_{owcr}(1-X_g)^A + BS_{gm}^2X_g(1-X_g) \quad \text{[EQ 59.7]}$$

where:

$$S_{gm} = 1 - S_{wco} - S_{ogcr} \quad \text{[EQ 59.8]}$$

Stone's first model exponent

The formula for Stone's first model may be rearranged such that an exponent may be applied to the combination of saturation terms:

$$k_{ro} = \beta(k_{row}k_{rog})/k_{rocw} \quad \text{[EQ 59.9]}$$

where

$$\beta = \left(\frac{SS_o}{(1-SS_w)\cdot(1-SS_g)}\right)^{\eta} \quad \text{[EQ 59.10]}$$

This formulation is due to Hustad and Holt [Ref. 124].The β term may be interpreted as a variable that varies between zero and one for low and high oil saturations respectively. If the exponent η is equal to 1.0, the formula corresponds to Stone's first model. Values above 1.0 cause the oil isoperms to spread within the saturation space, whereas values below 1.0 have the opposite effect. The value of the exponent may be used, therefore, to match the predicted oil recovery to the observed data and may be specified using the STONE1EX keyword.

Stone's second model (Modified)

The third model provided in ECLIPSE for calculating values of the three phase oil relative permeability is a modified form of the second model suggested by Stone [Ref. 43]. The formula is

$$k_{ro} = k_{rocw}\left[\left(\frac{k_{row}}{k_{rocw}} + k_{rw}\right)\left(\frac{k_{rog}}{k_{rocw}} + k_{rg}\right) - k_{rw} - k_{rg}\right] \quad \text{[EQ 59.11]}$$

where k_{rog} denotes the oil relative permeability for a system with oil, gas and connate water; k_{row} denotes the oil relative permeability for a system with oil and water only. Both sets of two phase oil relative permeability functions are tabulated as functions of oil saturation in the input data. k_{rocw} is the oil relative permeability in the presence of connate water only.

Note the values of k_{ro} produced by this formula can be negative. ECLIPSE automatically changes any negative k_{ro} values produced to zero.

The calculation of three phase oil relative permeability using Stone's method 2 is enabled by means of STONE2 keyword in the PROPS section of the input data. Tabulated output of the three phase oil relative permeability values is controlled by the mnemonic SOF in the RPTPROPS keyword.

IKU method

ECLIPSE 300

The IKU model, which is due to Hustad [Ref. 120] and [Ref. 121], is an alternative three-phase relative permeability model to those of Stone and is selected by specifying the IKU3P keyword in the PROPS section. This model applies to all three phases symmetrically and data are entered for water and gas with respect to the other two phases using the SWF3 and SGF3 keywords. Normalized saturations S_{in} are used to look up, in normalized tables, the pairs of two-phase relative permeabilities $k_{rij}(S_{in})$ and $k_{rik}(S_{in})$ where $i, j, k = g, o, w$ subject to the constraint that $i \neq j \neq k$. These are then combined to calculate the three-phase relative permeability k_{ri}. The normalized saturation S_{in} is given by:

$$S_{in} = \frac{S_i - S_{imn}}{S_{imx} - S_{imn}} \qquad \text{[EQ 59.12]}$$

where S_i is the grid block saturation for phase i and S_{imn} and S_{imx} is given by:

$$S_{imn} = \frac{S_j S_{irj} + S_k S_{irk} + S_{irj} S_{irk}(S_i - 1)}{S_j(1 - S_{irk}) + S_k(1 - S_{irj})} \qquad \text{[EQ 59.13]}$$

$$S_{imx} = \frac{S_j S_{kri} + S_k S_{jri} + S_{jri} S_{kri}(S_i - 1)}{S_j S_{kri} + S_k S_{jri}} \qquad \text{[EQ 59.14]}$$

where

S_j and S_k are the grid block saturations for phases j and k respectively and

S_{irj}, S_{irk}, S_{jri}, S_{kri}, S_{krj} and S_{jrk} are the residual saturations of phases i to j, i to k, j to i, k to i, k to j and j to k respectively.

The normalized table look-up for $k_{rij}(S_{in})$ is accomplished by normalizing the tabulated data for $k_{rij}(S_i)$ between the lower limit S_{irj} and the upper limit $1 - S_{jri}$ such that $S_i = S_{irj}$ and $S_i = 1 - S_{jri}$ correspond to 0 and 1 respectively. The value of $k_{rij}(S_{in})$ is then determined for the supplied value of S_{in}. A similar process is employed to calculate $k_{rik}(S_{in})$, $k_{rji}(S_{jn})$, $k_{rjk}(S_{jn})$, $k_{rki}(S_{kn})$ and $k_{rkj}(S_{kn})$.

Each pair of two-phase relative permeabilities is then combined to calculate the three-phase relative permeability:

$$k_{ri} = \frac{S_j}{S_j + S_k} \cdot k_{rij} + \frac{S_k}{S_k + S_j} \cdot k_{rik} \qquad \text{[EQ 59.15]}$$

and similarly for k_{rj} and k_{rk}. This formulation is symmetrical and is applied to all three phases.

This calculation process may be illustrated for the case of relative permeability to oil k_{ro} with the aid of a ternary diagram (Figure 59.5):

Figure 59.5 Ternary diagram showing the linear interpolation used to calculate S_{omn} and S_{omx}

It is assumed that a three-phase saturation condition exists as depicted by the point denoted by (S_g, S_o, S_w) in the centre of the ternary diagram. A line is drawn through this point and the oil apex. This line meets the line constructed by joining the grid block points S_{orw} and S_{org} at the minimum saturation to oil flow S_{omn} and the line constructed by joining the grid block points S_{wro} and S_{gro} at the maximum saturation to oil flow S_{omx}. These oil saturation extrema are calculated according to:

$$S_{omn} = \frac{S_w S_{orw} + S_g S_{org} + S_{orw} S_{org}(S_o - 1)}{S_w(1 - S_{org}) + S_g(1 - S_{orw})} \qquad \text{[EQ 59.16]}$$

$$S_{omx} = \frac{S_w S_{gro} + S_g S_{wro} + S_{wro} S_{gro}(S_o - 1)}{S_w S_{gro} + S_g S_{wro}} \qquad \text{[EQ 59.17]}$$

using values pertaining to the grid block, and the normalized oil saturation S_{on} for the grid block is calculated according to:

$$S_{on} = \frac{S_o - S_{omn}}{S_{omx} - S_{omn}} \qquad \text{[EQ 59.18]}$$

The two-phase relative permeabilities $k_{row}(S_{on})$ and $k_{rog}(S_{on})$ are determined from table look-up where $k_{row}(S_o)$ has been normalized to unity extent between the table values for S_{orw} and $1-S_{wro}$ and $k_{rog}(S_o)$ has been normalized to unity extent between the table values for S_{org} and $1-S_{gro}$. Finally k_{ro} is calculated from:

$$k_{ro} = \frac{S_w}{S_w + S_g} \cdot k_{row} + \frac{S_g}{S_g + S_w} \cdot k_{rog} \qquad \text{[EQ 59.19]}$$

k_{rg} and k_{rw} are calculated in a similar fashion.

Baker method

ECLIPSE 300

A further relative permeability options are two Baker's interpolation methods, obtained with the BAKER1 and BAKER2 keyword in the PROPS section. This method is due to Baker [Ref. 123].

The keyword BAKER1 is used to perform linear interpolation for each of the oil (o), water (w), and gas (g) phases, respectively, based on the relation:

$$k_l = \frac{(S_i - S_{ic})K_{rli} + (S_j - S_{jc})K_{rlj}}{(S_i - S_{ic}) + (S_j - S_{jc})}, \text{ where } k \neq i \neq j, \qquad \text{[EQ 59.20]}$$

$k = w, g, o; i = w, g, o; j = w, g, o;$

The keyword BAKER2 is used to perform linear interpolation for each of the oil (o), water (w), and gas (g) phases, respectively, based on the relation:

$$k_l = \frac{S_i K_{rli} + S_j K_{rlj}}{S_i + S_j}, \text{ where } k \neq i \neq j, \qquad \text{[EQ 59.21]}$$

$k = w, g, o; i = w, g, o; j = w, g, o;$

The K_{rli} and K_{rlj} are then looked up with respect to considering phase, which is scaled into the corresponding two-phase mobile regions. This linear interpolation may be shown geometrically on a ternary diagram similar to Figure 59.5.

ODD3P method

Introduction

ECLIPSE 300

The ODD3P method provides a coupled hysteretic model for three-phase relative permeabilities and capillary pressures. This method is due to Hustad [Ref. 122] and extends the IKU method [Ref. 121] described above. A summary of the ODD3P method is presented below. A full description of this method may be found in [Ref. 122]. It is recommended that this reference is read as a precursor to using this model.

Whereas the Stone-type models are based upon water-wet systems, the ODD3P method is more general with respect to wettability in that it uses pairs of two-phase relative permeability data for each of the three phases, with the wettability property being implied via the supplied input data.

Motivation

The primary motivation for the development of the ODD3P method has been the difficulty in reproducing measured core flooding behavior using reservoir simulators with the traditional models of three-phase relative permeability [Ref. 122]. The ODD3P method also ensures consistency (continuous functions) of the three-phase properties at the two-phase and single-phase boundaries with the respective two-phase properties. The saturation profiles observed in core flooding simulations have been found to be strongly dependent upon the capillary forces and the boundary conditions at the core inlet and outlet. In addition, the relative permeability has been found to depend not only upon fluid saturation but also upon saturation history. The requirements for the ODD3P method, therefore, include relative permeability and capillary pressure values which are saturation history dependent. Hence the properties assigned to the relative permeability and capillary pressure for a particular set of three-phase saturations should not be unique but instead depend upon the saturation history leading up to the current saturation condition.

Problems have also been found to occur when modelling three-phase systems if a first-contact miscible injection process causes an oil-water system to become a gas-water system due to the labelling of the hydrocarbon phase. This can result in discontinuities in relative permeabilities and capillary pressures. A further motivation for the ODD3P method, therefore, has been to ensure that relative permeability and capillary pressure are consistent and continuous for the flow processes which transition from any two-phase state to another two-phase state via a three-phase state.

In this model an important distinction is made between the end-point saturations and residual saturations [Ref. 122]. The end-point saturation is defined as the lowest value of saturation for which the phase is continuous and also corresponds to the saturation where the relative permeability is zero. In contrast, the residual saturation is defined as the saturation at which the phase becomes immobile. This condition may arise for different values of capillary pressure according to the pressure gradient of the phase or how high or low a capillary pressure is achieved by the flow conditions. The relative permeability need not be zero at the residual saturation, but for a phase not to flow, the phase pressure gradients must be zero. On the basis of these definitions, therefore, the residual saturation is greater than or equal to the end-point saturation.

End-point and residual saturation data will typically incorporate information about the processes which have been subjected to upscaling. For this model, the end-point saturation applied to a simulation model grid block may be related to the upscaled processes by including a representation of the sweep efficiency of the grid block using a Land-type formulation [Ref. 21].

Overview

The ODD3P method provides a three-phase relative permeability model which is based upon the saturation functions of the contributing three pairs of two-phase relative permeability saturation functions. This method extends the normalized saturation approach employed by the IKU method [Ref. 121], which is described above. In the ODD3P method, relative permeability and capillary pressure hysteresis effects are incorporated together with grid-block end-point scaling which can optionally include miscible effects.

Instead of the traditional specification of drainage and imbibition saturation functions and the explicit selection of the wetting phase, three sets of saturation functions are employed which are based upon the following premises:

1 Originally, in terms of geological time, formations which currently contain hydrocarbon fluids contained only water. Over a period of millions of years, hydrocarbon fluids gradually penetrated these formations until a point of equilibrium was reached between water, oil and gas. A set of primary saturation functions is used to represent this initial state of the reservoir and the initial production of hydrocarbon fluids. These also define the initial departure of the reservoir from its equilibrium conditions.

2 When the formation is exploited commercially, an initial process of sometimes unassisted oil and gas production is usually followed by injection to maintain recovery. Typically the injection process will consist of alternating water and gas (WAG) injection which is one mechanism for improving oil recovery. At some point the saturation behavior will cease to follow the primary saturation functions and enter a hysteresis loop. Two sets of hysteresis saturation functions are used to model the hysteresis behavior experienced by the reservoir when fluid saturations increase and decrease as a function of the cyclical injection processes.

The hysteresis curves are specified in terms of increasing and decreasing saturation directions instead of the more usual notation of drainage and imbibition. For example, the relative permeability to water for the water-oil system is expressed both in terms of the increasing and decreasing water saturation. This corresponds to processes which result in the water saturation increasing and decreasing respectively with respect to the oil saturation. Conversely, the relative permeability to oil for the oil-water system is expressed in terms of increasing and decreasing oil saturation with respect to the water saturation.

The use of primary saturation functions which are distinct from the secondary and tertiary hysteresis saturation functions permits the modeling of the relative permeability behavior where, for example, the decreasing saturation characteristics of relative permeability are different for primary and tertiary processes. One complexity of this model is establishing the appropriate criteria for the relative permeabilities and capillary pressures to transition from the primary to the hysteresis saturation functions.

Three-Phase Formulation

The three-phase relative permeabilities and capillary pressures are based upon pairs of representative two-phase properties which are combined through a saturation weighting scheme. Each phase property is made dependent upon its own normalized saturation in order to ensure consistency [Ref. 121]. Zero relative permeabilities are ensured at and below the end-point saturations. The gas-water data is represented by the oil-water data in order to avoid the difficulties associated with representing the hydrocarbon-water system when gas and oil are miscible. This is especially important for first-contact miscible floods in which phase labeling changes between gas and oil require that the functions of relative permeability and capillary pressure remain continuous. This approach also avoids the potential interpolation complexities associated with the use of separate saturation functions tabulated against different sets of saturation values.

Unlike the choice of the different Stone methods for the selection of three-phase relative permeability models and the separate selection of the different methods for modelling hysteresis effects, for example, Jargon or Killough, the ODD3P method incorporates implicitly both three-phase relative permeability and capillary pressure hysteresis. In addition, no explicit specification of the wetting phase is required. Instead, the wetting phase is implicit in the characteristics of the tables of relative permeability and capillary pressure expressed as functions of gas, oil and water saturation for the primary, secondary hysteresis and tertiary hysteresis processes.

Saturation Process Modes (Directions)

The primary, secondary hysteresis and tertiary hysteresis processes refer to the direction of the phases' saturations. The directions of the gas, oil and water saturations are defined with respect to the change in their respective grid-block saturations. The relative permeability and capillary pressure data are based upon three pairs of two phase data such that, for each two-phase pair, $S_i = 1 - S_j$ and $S_k = 0$ where the subscripts i, j and k denote the three phases subject to the constraint that $i \neq j \neq k$. The saturation directions for each phase may be summarized as follows:

Primary: S_w decreasing

S_g increasing

$S_o = 1 - S_g$ or $S_o = 1 - S_w$

Secondary hysteresis: S_w increasing

S_g decreasing

$S_o = 1 - S_g$ increasing

$S_o = 1 - S_w$ decreasing

Tertiary hysteresis: S_w decreasing

S_g increasing

$S_o = 1 - S_g$ decreasing

$S_o = 1 - S_w$ increasing

For a given saturation process mode, the gas saturation direction will be in the opposing direction to that of the water saturation. For the increasing and decreasing hysteresis saturation directions, the oil saturation direction will be in the opposing direction to the water or gas saturation against which it has been measured. For example, for the secondary hysteresis process, gas saturation is decreasing; therefore, if the oil saturation is measured or modeled against gas saturation, the measurements for increasing oil saturation will be used for the secondary hysteresis process.

Availability and Limitations

The ODD3P model is currently only available for ECLIPSE 300 compositional models using an equation of state solution method.

ODD3P Model Definition

The ODD3P three-phase relative permeability and capillary pressure hysteresis model is selected by specifying the `ODD3P` keyword in the `PROPS` section.The use of the default three-phase relative permeability model and the other permeability models `STONE` (an alias for `STONE2`) and `STONE1`, `STONE2`, `IKU3P` and `ODD3P` are mutually exclusive.

ODD3P Saturation Functions

Saturation Function Keywords

The relative permeabilities and capillary pressures are expressed as a function of gas, oil and water saturation using the SGF3, SOF3 and SWF3 keywords respectively. In general, three sets of these tables will be required per saturation region corresponding to the primary, secondary hysteresis and tertiary hysteresis saturation processes.

In order to avoid discontinuities due to phase labelling changes in miscible models, the gas-water relative permeability data are replaced by the oil-water relative permeability data. Therefore, the relative permeability to water data for a water-oil system k_{rwo} (column 2 of the SWF3 keyword) are used to represent the relative permeability to water for a gas-water system k_{rwg} (column 3 of the SWF3 keyword). Similarly, the relative permeability to oil data for an oil-water system k_{row} (column 2 of the SOF3 keyword) are used to represent the relative permeability to gas for a gas-water system k_{rgw} (column 3 of the SGF3 keyword). It follows that the data in column 3 of the SWF3 and SGF3 keywords may be defaulted, subject to the caveat that sufficient information is specified in terms of initial values of 0.0 and final values of 1.0 in order to infer the residual and maximum saturations to be applied to the gas-water and water-gas systems. An example of the specification of these three keywords is shown below in abbreviated form.

...

```
SGF3
---      Sg        krgo(Sg)        krgw(Sg)        Pcog(Sg)
     0.00000   0.000000E+00   0.000000E+00   0.000000E+00
     0.00010   0.000000E+00   0.000000E+00   0.000000E+00
     0.00034   0.141600E-13          1*      0.197385E-02
     0.00049   0.610900E-13          1*      0.276339E-02
     ...
     0.83000   0.502886E+00          1*      0.404639E-01
     0.91000   0.726646E+00          1*      0.542808E-01
     0.95000   0.865329E+00   0.100000E+01   0.800395E+00
     0.96000   0.900000E+00   0.100000E+01   0.986923E+00
/
```

```
SOF3
--      So         krow(So)        krog(So)
     0.00000   0.000000E+00   0.000000E+00
     0.04000   0.140000E-02   0.000000E+00
     0.09000   0.315000E-02   0.383300E-06
     0.17000   0.595000E-02   0.455400E-04
     0.20000   0.700000E-02   0.333655E-03
     ...
     0.82000   0.670000E+00   0.360146E+00
     0.85000   0.700000E+00   0.433916E+00
     0.87000   0.700000E+00   0.483097E+00
     0.95000   0.700000E+00   0.765334E+00
     0.99767   0.700000E+00   0.987923E+00
     0.99922   0.700000E+00   0.995944E+00
     0.99951   0.700000E+00   0.997451E+00
     0.99966   0.700000E+00   0.998230E+00
     0.99990   0.700000E+00   0.100000E+01
/
```

```
SWF3
--      Sw         krwo(Sw)        krwg(Sw)        Pcwo(Sw)
     0.05000   0.000000E+00   0.000000E+00   0.986923E+00
     0.15000   0.000000E+00          1*      0.986923E+00
     0.18000   0.150000E-02          1*      0.493462E-01
     ...
     0.80000   0.500000E+00          1*      0.102640E-01
     0.99990   0.999750E+00   0.100000E+01   0.888300E-02
     1.00000   0.100000E+01   0.100000E+01   0.888231E-02
/
```

In practice it is envisaged that the relative permeability and capillary pressure data will be obtained either from laboratory measurements of cores or from network simulation models where grain size distributions and pore wettabilities will be representative of the reservoir being modeled. The data are obtained from three discrete sets of two phases measurements or simulations, that is, in the absence of the third phase. In particular it should be noted that, for the SOF3 keyword, the relative permeability to oil for an oil-gas system should be defined in the absence of connate water.

Saturation Function Region Numbers

The association between saturation tables and saturation region numbers follows the usual pattern. However, whereas in the traditional hysteresis models, the drainage and imbibition saturation tables are associated with saturation regions using the keywords SATNUM and IMBNUM respectively, for the ODD3P method, the keywords PSTNUM, ISTNUM and DSTNUM are used to define the tables associated with the primary, secondary hysteresis and tertiary hysteresis saturations respectively. Internally, the SATNUM and IMBNUM keywords are aliases of the PSTNUM and ISTNUM keywords respectively and may be used as alternatives. This may be useful when converting existing data sets to work with the ODD3P model. However, there is no equivalent internal alias for the DSTNUM keyword. It is recommended, therefore, that for the purposes of clarity the matching trio of PSTNUM, ISTNUM and DSTNUM keywords should be used wherever possible.

The saturation region keyword PSTNUM refers to the primary saturation tables, ISTNUM refers to the tables containing the secondary hysteresis saturation functions, and DSTNUM refers to the tables containing the tertiary hysteresis saturation functions. The selection of the appropriate combination of saturation functions for a given process direction is made internally according to the current saturations and the history of the saturation directions thus far in the simulation.

The PSTNUM, ISTNUM and DSTNUM keywords are optional but are required if hysteresis is to be modeled. These may be used to define the entire set of grid blocks or some subset via the usual set of array assignment and manipulation operators. For grid blocks for which one or more of these keywords have not been specified, the following defaulting process will be applied:

- PSTNUM: Primary saturation function region number defaults to 1.
- ISTNUM: Secondary hysteresis saturation function region number defaults to PSTNUM.
- DSTNUM: Tertiary hysteresis saturation function region number defaults to ISTNUM.

Although the ODD3P method is implicitly hysteretic, hysteresis may be switched off for a given grid block by defining the values of ISTNUM and DSTNUM to be equal. In addition, the modeling of separate primary, secondary and tertiary processes may be switched of by specifying the values of ISTNUM and DSTNUM not only to be equal to each other but also equal to the value of PSTNUM. The only constraint upon the values of PSTNUM, ISTNUM and DSTNUM for a given grid block, therefore, is that if PSTNUM is equal to ISTNUM or DSTNUM, then ISTNUM and DSTNUM must also be equal. Failure to comply with this constraint will cause an error to be reported during the checking of the input data.

Saturation Function Data Constraints

Because of the complexity of the ODD3P model, there is a number of specific input data constraints associated with the specification of the saturation function data for this model. These are summarized below.

1. For a specified pair of hysteresis curves, the hysteresis value of the end-point saturation S_{irj} is taken as the lower value of the two S_{irj} values for the secondary and tertiary hysteresis curves, where S_{irj} is defined to be the maximum tabulated saturation value in S_i for which k_{rij} is zero, with S_i measured in the increasing sense. In cases where the secondary and tertiary hysteresis values of S_{irj} are found to be different, which is permissible, a warning is issued. This is to identify possible mismatches between the pairs of curves specified for secondary and tertiary hysteresis operations.
2. Each pair of hysteresis curves are required to form a closed hysteresis loop in the region enclosed between their bounding residual saturations. This may be expressed as follows:
   a. For the increasing and decreasing relative permeability functions $k_{rij}(S_i(inc))$ and $k_{rij}(S_i(dec))$, the values should be equal at S_{irj} and $1 - S_{jri}$.
   b. Similarly for the increasing and decreasing relative permeability functions $k_{rji}(S_j(inc))$ and $k_{rji}(S_j(dec))$, the values should be equal at S_{jri} and $1 - S_{irj}$.
   c. Corresponding constraints apply to the increasing and decreasing hysteresis oil-water and gas-oil capillary pressures.

   If the above conditions are not satisfied, appropriate warning messages will be issued. Failure to comply with these conditions will lead to inconsistent or unpredictable results.

3 In general, it is expected that the relative permeability functions will be monotonically increasing over the range of the closed loop hysteresis curves. Similarly, it is expected that the capillary pressures will vary monotonically over this range. Although these conditions are not mandatory warning messages will be issued if they are not met.

4 The first saturation table entries for the secondary and tertiary hysteresis curves are checked for equal initial values of saturation and a warning is issued if this test is not satisfied. This is to identify possible mismatches between pairs of curves specified for secondary and tertiary hysteresis operations.

5 For the primary saturation SGF3 tables, the initial gas saturation value should be 0.0. Similarly for the primary SWF3 tables, the final water saturation value should be 1.0. In most cases, for the primary tables, negative values of capillary pressure are not required, and a warning is issued if negative values are encountered.

6 In general, the primary capillary pressure curve should not intersect the secondary hysteresis or tertiary hysteresis capillary pressure curves, although such intersections do not prevent correct operation of the model. A warning is issued if any such conditions are encountered.

7 If the secondary and tertiary hysteresis capillary pressure curves intersect within the hysteresis saturation extent, the hysteresis modelling will fail and this will be reported as an error condition during the input data checking.

8 There are no constraints on the intersection of the primary, secondary hysteresis and tertiary hysteresis relative permeability curves.

End-Point Scaling

For the ODD3P method, when the ENDSCALE keyword has been specified in the RUNSPEC Section, end-point scaling is accomplished by applying scaling via grid-block end-point saturations using the PROPS Section keywords PSORG, PSGRO, PSORW, PSWRO, PSGRW and PSWRG for the primary saturation functions and the corresponding keywords HSORG, HSGRO, HSORW, HSWRO, HSGRW and HSWRG for the hysteresis saturation functions. Only one set of hysteresis end-point scaling keywords is required because the hysteresis saturation functions form closed loop curves. For a given grid block, the following constraints apply to the specification of these data values:

Table 59.3 End-Point Scaling Constraints

Residual Saturation	Primary Constraint	Hysteresis Constraint	Primary-Hysteresis Constraint
SORW		$HSORW > 0.0$	$PSORW \leq HSORW$
SWRO	$PSWRO > 0.0$	$HSWRO > 0.0$	$PSWRO = HSWRO$
SORG	$PSORG > 0.0$	$HSORG > 0.0$	$PSORG = HSORG$
SGRO	$PSGRO \leq 0.001$	$HSGRO > 0.0$	$PSGRO < HSGRO$
SGRW		$HSGRW > 0.0$	$PSGRW < HSGRW$
SWRG	$PSWRG > 0.0$	$HSWRG > 0.0$	$PSWRG = HSWRG$

If ENDSCALE has not been specified, the grid-block end-point values will be taken from those of the appropriate saturation table. Similarly if ENDSCALE has been specified but not all end-point values have been defined for each grid block using the above keywords, the unspecified values will default to those of the appropriate saturation table. The above constraint checking is performed after any defaults have been applied. Failure to comply with the above input data constraints will lead to warning messages.

Capillary Pressure Calculation Control

The calculation of the capillary pressure is controlled by the PROPS Section table keywords PCODD3P, PCODD3PG and PCODD3PW which should be specified for each saturation region. In practice, only those tables which are associated with the primary saturation region numbers are used; those associated with the secondary and tertiary hysteresis region numbers may be defaulted. The PCODD3P keyword consists of a single row of seven columns of data per saturation region. These are defined as follows:

1 Capillary pressure calculation type. This provides thirteen functional relationships for the calculation of capillary pressure which are numbered 0 to 12.

2 The constant δ_g in the gas saturation dependent capillary pressure weighting function.

3 The first exponent α_g in the gas saturation dependent capillary pressure weighting function.

4 The second exponent β_g in the gas saturation dependent capillary pressure weighting function.

5 The constant δ_w in the water saturation dependent capillary pressure weighting function.

6 The first exponent α_w in the water saturation dependent capillary pressure weighting function.

7 The second exponent β_w in the water saturation dependent capillary pressure weighting function.

The data specified in columns 2 to 7 are only required if the functional relationship type is between 1 and 8. For these types, the capillary pressure weighting functions $H(S_g)$ and $F(S_w)$ are calculated as shown below:

$$H(S_g) = \frac{\delta_g S_g^{\alpha_g}}{S_g^{\alpha_g} + (1 - S_g)^{\beta_g}} \qquad \text{[EQ 59.22]}$$

and

$$F(S_w) = \frac{\delta_w S_w^{\alpha_w}}{S_w^{\alpha_w} + (1 - S_w)^{\beta_w}} \qquad \text{[EQ 59.23]}$$

The *representative* capillary pressures, that is, the values obtained from table look-up using the normalized saturations are corrected according to the following equations:

$$P_{cow} = \hat{P}_{cow} + H \cdot R \qquad \text{[EQ 59.24]}$$

$$P_{cgo} = \hat{P}_{cgo} + F \cdot R \qquad \text{[EQ 59.25]}$$

$$R = \hat{P}_{cgw} - \hat{P}_{cgo} - \hat{P}_{cow} \qquad \text{[EQ 59.26]}$$

where the circumflex denotes the **representative** (IFT scaled) capillary pressures. The corrected capillary pressures may be made functions of one of two pairs of saturations as shown below:

Table 59.4 Saturation functions

Saturation Dependence	Oil-Water System	Gas-Oil System	Gas-Water System
1	$\hat{P}_{cow}(S_o)$	$\hat{P}_{cgo}(S_g)$	$\hat{P}_{cgw}(S_g)$
2	$\hat{P}_{cow}(S_w)$	$\hat{P}_{cgo}(S_o)$	$\hat{P}_{cgw}(S_w)$

The calculation control parameter defined above in column 1 of the `PCODD3P` keyword selects the functional relationship type to calculate the capillary pressures as shown below:

0: $P_{cow} = 0.0\,, \qquad P_{cgo} = 0.0$ [EQ 59.27]

1 and 2: $P_{cow} = \hat{P}_{cow}(S_o) + H \cdot R\,, \qquad P_{cgo} = \hat{P}_{cgo}(S_g) + F \cdot R$ [EQ 59.28]

3 and 4: $P_{cow} = \hat{P}_{cow}(S_o) + H \cdot R\,, \qquad P_{cgo} = \hat{P}_{cgo}(S_o) + F \cdot R$ [EQ 59.29]

5 and 6: $P_{cow} = \hat{P}_{cow}(S_w) + H \cdot R\,, \qquad P_{cgo} = \hat{P}_{cgo}(S_g) + F \cdot R$ [EQ 59.30]

7 and 8: $P_{cow} = \hat{P}_{cow}(S_w) + H \cdot R \qquad P_{cgo} = \hat{P}_{cgo}(S_o) + F \cdot R$ [EQ 59.31]

9: $P_{cow} = \hat{P}_{cow}(S_o)\,, \qquad P_{cgo} = \hat{P}_{cgw}(S_w) - \hat{P}_{cow}(S_o)$ [EQ 59.32]

10:

$$P_{cow} = \frac{S_g}{S_g + S_o}\hat{P}_{cgw}(S_w) + \frac{S_o}{S_g + S_o}\hat{P}_{cow}(S_w)\,,$$

$$P_{cgo} = \frac{S_w}{S_o + S_w}\hat{P}_{cgw}(S_g) + \frac{S_o}{S_o + S_w}\hat{P}_{cgo}(S_g) \qquad \text{[EQ 59.33]}$$

11:

$$P_{cow} = \frac{S_g}{S_g + S_o}\hat{P}_{cgw}(S_w) + \frac{S_o}{S_g + S_o}\hat{P}_{cow}(S_w)\,,$$

$$P_{cgo} = \hat{P}_{cgo}(S_g) \qquad \text{[EQ 59.34]}$$

12:

$$P_{cow} = \frac{S_g(S_o[\hat{P}_{cgw}(\tilde{S}_w) - \hat{P}_{cgo}(\tilde{S}_g)] + S_w[\hat{P}_{cgw}(\tilde{S}_w) - \hat{P}_{cgw}(\tilde{S}_g)]) + S_o(S_o + S_w)\hat{P}_{cow}(\tilde{S}_w)}{S_o(S_g + S_o + S_w)}$$

$$P_{cgo} = \left[\frac{S_o}{(S_o + S_w)} + \frac{S_g S_w}{(S_o + S_w)(S_g + S_o + S_w)}\right] \cdot \hat{P}_{cgo}(\tilde{S}_g) + \qquad \text{[EQ 59.35]}$$

$$\left[\frac{S_w}{(S_o + S_w)} + \frac{S_g S_w^2}{S_o(S_o + S_w)(S_g + S_o + S_w)}\right] \cdot \hat{P}_{cgw}(\tilde{S}_g) -$$

$$\left[\frac{S_g S_w}{(S_g + S_o + S_w)}\right] \cdot \hat{P}_{cgw}(\tilde{S}_w) - \left[\frac{S_w}{(S_g + S_o + S_w)}\right] \cdot \hat{P}_{cow}(\tilde{S}_w)$$

For functional relationship types 1 to 8, R is calculated as shown below:

1: $R = \hat{P}_{cgw}(S_g) - \hat{P}_{cgo}(S_g) - \hat{P}_{cow}(S_o)$ [EQ 59.36]

2: $R = \hat{P}_{cgw}(S_w) - \hat{P}_{cgo}(S_g) - \hat{P}_{cow}(S_o)$ [EQ 59.37]

3: $R = \hat{P}_{cgw}(S_g) - \hat{P}_{cgo}(S_o) - \hat{P}_{cow}(S_o)$ [EQ 59.38]

4: $R = \hat{P}_{cgw}(S_w) - \hat{P}_{cgo}(S_o) - \hat{P}_{cow}(S_o)$ [EQ 59.39]

5: $R = \hat{P}_{cgw}(S_g) - \hat{P}_{cgo}(S_g) - \hat{P}_{cow}(S_w)$ [EQ 59.40]

6: $R = \hat{P}_{cgw}(S_w) - \hat{P}_{cgo}(S_g) - \hat{P}_{cow}(S_w)$ [EQ 59.41]

7: $R = \hat{P}_{cgw}(S_g) - \hat{P}_{cgo}(S_o) - \hat{P}_{cow}(S_w)$ [EQ 59.42]

8: $R = \hat{P}_{cgw}(S_w) - \hat{P}_{cgo}(S_o) - \hat{P}_{cow}(S_w)$ [EQ 59.43]

The functional relationship types 9, 10 and 11 correspond to cases where R is equal to 0.0.

Capillary pressure functional dependence calculation control parameter options 9, 10 and 11 correspond to cases where R is equal to 0.0. For options 10, 11 and 12 the saturation weightings are obtained from the grid block saturations S_i. Option 12 incorporates saturation weighting based upon all three grid block saturations (S-Weighted Model). This model has been developed by Statoil ASA see [Ref. 130] and results obtained from this model are described in [Ref. 131] and compared with alternative models above.

For ultimate flexibility the capillary pressure weighting functions $H(S_g)$ and $F(S_w)$ may be defined explicitly using the keywords `PCODD3PG` and `PCODD3PW` which tabulate these functions against gas and water saturation respectively. Either of these keywords may optionally be specified. If the `PCODD3PG` keyword is specified, the values defined in columns 2, 3 and 4 of the `PCODD3P` keyword will be ignored. Similarly, if the `PCODD3PW` keyword is specified, the values defined in columns 5, 6 and 7 of the `PCODD3P` keyword will be ignored.

End-Point Scaling Calculation Control

The end-point scaling calculation is controlled by the `PROPS` section table keyword `EPSODD3P`. This keyword provides control over the application of a sweep efficiency component to the process-dependent end-point saturations. The following Land formulation [Ref. 122] is used for this purpose and calculates a modified grid-block end-point saturation S_{irj} according to the expression:

$$S_{irj} = \frac{S_i^{max}}{1 + \dfrac{S_i^{max}}{1 + S_{irj}^p} - \dfrac{S_i^{max}}{1 - S_{jri}^p}} \qquad \text{[EQ 59.44]}$$

where S_i^{max} is the maximum grid block saturation and S_{irj}^p and S_{iri}^p are the process-dependent grid-block end-point saturations and $i \neq j$, and S_i^{max} is subject to the constraint $S_i^{max} \leq 1 - S_{jri}^p$. The Land formulation may be applied individually to any or all of the oil, gas and water saturations.

Saturation Direction Indicators

During the calculation process, six saturation process direction indicators are employed to monitor the switching of the process directions from primary to hysteresis and between increasing and decreasing saturations. These six saturation indicators are specified per grid block and may possess one of four values:

1 Using primary saturation functions with switching to hysteresis not permitted.

2 Using primary saturation functions with switching to hysteresis permitted.

3 Using increasing saturations.

4 Using decreasing saturations.

Internally, these saturation direction indicators are assigned an initial value of 1. When, during the course of the simulation, the grid-block saturations come within the hysteresis extent as defined by the hysteresis grid-block end-points, the indicators are assigned a value of 2. Subsequently when a saturation process direction change occurs these indicators are assigned a value of 3 or 4 corresponding to whether the saturation direction is increasing or decreasing respectively. Thereafter, the saturation direction indicators will switch between the values of 3 and 4 in response to changes in the phase's saturation direction.

In some special cases, it may be helpful to initialize these saturation direction indicators to values other than 1, for example, when the results of some experiment or some other simulation provide details for the initial saturation states. Under these circumstances, it is possible to initialize these saturation direction pointers to values in the range of 1 to 4 on a per grid-block basis using the REGION Section keywords `SDROW`, `SDRWO`, `SDROG`, `SDRGO`, `SDRGW` and `SDRWG`. When only partial array specifications are provided for these arrays, the unspecified portions of the arrays will be initialized to 1. Considerable care should be exercised when using this facility. In general, the initial is at ion of these arrays to non-default values will not be necessary and these arrays need not be specified in the input data.

Reporting Facilities

In view of the complexity of the ODD3P method, a considerable number of array reporting facilities are provided both as solution arrays and for output to the Summary file on a per grid-block basis. The arrays which are provided for output are listed below with the characters i and j signifying gas G oil, O and water W, noting that $i \neq j$. The following grid-block arrays are considered to be useful for the general monitoring of the calculation process.

- `KRijR`: Representative relative permeabilities.
- `SDRij`: Saturation process direction indicators.
- `SiNRM`: Normalized saturations.
- `SiRjB`: Residual saturations.

When specifying the required outputs in the input data file, the characters i and j should be replaced with the specific phase pairs which are required. For output on a per grid-block basis as Summary vectors, the array names above should be pre-appended with a B and defined in the `SUMMARY` section for the selected grid-block indices.

Table end points

In the equilibration calculation (see keyword EQUIL) the water and gas saturation values above and below the transition zones are determined from the maximum and minimum saturation values in the water and gas saturation tables. The oil saturation values are thus defined consequentially from these values. The initial saturations in each zone are shown in Figure 59.6.

Figure 59.6 Initial saturations for each zone

GAS ZONE	$Sg = Sg\ max$ $Sw = Swco$ $So = 1 - Sg\ max - Swco$
GAS - OIL TRANSITION ZONE	
OIL ZONE	$Sg = Sgco$ $Sw = Swco$ $So = 1 - Sgco - Swco$
OIL-WATER TRANSITION ZONE	
WATER ZONE	$Sg = Sgco$ $Sw = Sw\ max$ $So = 1 - Sgco - Sw\ max$

In the case where the two-dimensional alternatives of the saturation functions are used (SWF32D, SGF32D, PCW32D, PCG32D) the equilibration process is done as follows, based on the depth of the cell:

Figure 59.7 Equilibration process based on the depth of the cell

GAS	$Sw = Swco$ $So = 0$ *or from PCG32D (TZ)* $Sg = 1 - So - Swco$
OIL	$Sg = 0$ $So = 1\text{-}Swco$ *or from PCW32D (TZ* $Sw = 1 - So$
WATER	$Sg = 0$ $So = 0$ *or from PCW32D (TZ)* $Sw = 1 - So$

Note This equilibration process does not handle the situation where the water zone overlaps with the gas zone.

Consistency requirements

The oil, water and gas saturation tables, for each particular saturation table number region, must obey certain consistency requirements. These are detailed below.

1 $S_{g\ \max}$ must not exceed $1 - S_{wco}$

If this condition is violated, the gas saturation in the gas cap will be re-set to $1 - S_{wco}$, to prevent a negative oil saturation.

Normally, if there is no oil in the gas cap, $S_{g\ \max} = 1 - S_{wco}$.

2 S_{gco} must not exceed $1 - S_{w\ \max}$

If this condition is violated, the gas saturation in the water zone will be re-set to $1 - S_{w\ \max}$, to prevent a negative oil saturation.

Normally, there is no initial free gas below the gas cap and the water zone is fully saturated with water, thus $S_{gco} = 0$ and $1 - S_{w\ \max} = 0$.

3 $S_{o\ \max}$ must equal $1 - S_{wco} - S_{gco}$

4 $k_{row}(S_{o\ \max})$ must equal $k_{rog}(S_{o\ \max})$

5 $k_{rw}(S_w = 0) = k_{rg}(S_g = 0) = k_{row}(S_o = 0) = k_{rog}(S_o = 0) = 0$.

Otherwise, phases can be mobile even at zero saturation and there is nothing to stop saturations going negative.

In **oil-wet system** modeling, in order to apply correctly the Killough wetting phase model on the oil phase (option 7, Item 2 of `EHYSTR` keyword), the consistency requirements are slightly modified:

1' $S_{g\ \max}$ must equal $1 - S_{oco}$.

3' $S_{o\ \max}$ is normally equal to 1.0.

6 $S_{w\ \max}$ must equal $1 - S_{oco}$.

Mobile fluid requirements

ECLIPSE 100

In order to ensure that at least one fluid remains mobile in the three-phase ternary diagram, ECLIPSE 100 also checks for the following:

1 $S_{owcr} + S_{wcr} < 1$

2 $S_{ogcr} + S_{gcr} + S_{wco} < 1$

ECLIPSE issues a warning if these conditions are not satisfied. When `SCALECRS` is used, this would lead to an error because in an oil-water run, for example, K_{rw} would be scaled between S_{wcr} and $1 - S_{owcr}$ that is a null range if the first condition is not satisfied.

Figure 59.8 Ternary diagram showing mobile fluid end-points

Near critical oil and gas relative permeabilities

ECLIPSE 300

In compositional mode it is possible for a cell to make a direct transition from an oil to a gas, without passing through a series of intermediate saturations.

Consider a system with mobile water, initially regarded as an oil, but under gas injection. The effect of gas injection is to move the phase envelope to the left, and the system critical temperature can pass through the reservoir temperature.

Figure 59.9 The effect of gas injection

There must be continuity between the hydrocarbon relative permeability, K_{ro} when the system is an oil and K_{rg} when the system is a gas, for arbitrary S_w.

To do this we define a water-hydrocarbon relative permeability for single phase hydrocarbon states. Oil-gas and gas-oil relative permeabilities are only relevant if both oil and gas are present. This water-hydrocarbon K_{rh} must tend to the user entered K_{ro} when the composition is oil-like, and to K_{rg} when it is gas-like.

To set up a variable to interpolate between these two values, we define the pseudo-critical temperature T^{crit} which is given by the Li correlation as:

$$T^{\text{crit}} = \frac{\sum_j T_{cj} \cdot V_{cj} \cdot z_j}{\sum_j V_{cj} z_j} \qquad \text{[EQ 59.45]}$$

Then define f as:

$$f = \frac{T^{\text{crit}}}{T^{\text{res}}} \qquad \text{[EQ 59.46]}$$

f will clearly be 1 if the reservoir temperature T^{res} is the critical temperature, but becomes greater than 1 for an oil ($T^{\text{crit}} > T^{\text{res}}$) and less than 1 for a gas. If we choose values at which the system is to be regarded completely as an oil or a gas, such as $f_o = 1.25$ and $f_g = 0.75$ (these values may be user defined by the `LILIM` keyword), then an interpolating function may be defined as

$$E = \frac{f - f_g}{f_o - f_g} \qquad \text{[EQ 59.47]}$$

in the region $f_g < f < f_o$, and 0 or 1 outside this range. This goes from 0 when the system is gas-like (f below f_g) to 1 when the system is oil-like (f above f_o), with a linear interpolation in between. The hydrocarbon-water relative permeability may then be defined as

$$K_{rhw} = EK^u_{row} + (1 - E)K^u_{rgw} \qquad \text{[EQ 59.48]}$$

where K^u_{row} and K^u_{rgw} are the user-input oil and gas relative permeabilities in water, with their endpoints scaled, to force the endpoints of K_{rhw} to vary between those of K^u_{row} and K^u_{rgw} as E varies.

This is used for both oil and gas phases. A cell will make a transition from oil-filled to gas-filled when T^{crit} passes T^{res}, but the change from K_{row} to K_{rgw} will be continuous.

The oil-in-gas and gas-in-oil relative permeabilities are only used in the case when both oil and gas are present. To obtain the full oil and gas relative permeabilities in the case in which oil, gas and water are present either the default or the Stone method is used for oil, and the default method for gas. In the case in which the default method is used for oil, this yields:

$$K_{ro} = gK_{rhw}(S_w) + (1 - g)K_{rog}(S_o) \qquad \text{[EQ 59.49]}$$

$$g = \frac{S_w - S_{wc}}{S_w - S_{wc} + S_g} \qquad \text{[EQ 59.50]}$$

$$K_{rg} = hK_{rhw}(S_w) + (1 - h)K_{rgo}(S_g) \qquad \text{[EQ 59.51]}$$

$$h = \frac{S_w - S_{wc}}{S_w - S_{wc} + S_o} \qquad \text{[EQ 59.52]}$$

This interpolation is not used or required when `ISGAS` is specified. It will be disabled if `NOMIX` is used.

Saturation Table Scaling

Chapter 60

Introduction

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

The ECLIPSE saturation table End-point Scaling option allows you to redefine values for the connate, critical and maximum saturations in the saturation tables describing the flow of the reservoir fluids. The scaling facility is useful for modeling reservoirs which contain an initial depth variation of either the connate or critical saturations for one or more of the phases present. It has applications in the use of pseudo functions and in general cases where the saturation function data for the reservoir fluids depend on a normalized saturation variable.

The End-point Scaling option is enabled by the `ENDSCALE` keyword in the `RUNSPEC` section. Separate re-scaling is done on relative permeability and capillary pressure curves. Generally, each is a **two-point** rescaling, two saturation values in the tables being moved to new positions. There is also an option to apply a **three-point** rescaling to the relative permeability curves only.

In a three phase model there are eight saturation table end points which may be identified:

Table 60.1 Saturation table end points

Keyword	Description
`SWL`	The connate water saturation. This is the smallest water saturation entry in a water saturation table.
`SWCR`	The critical water saturation. This is the highest water saturation for which the water is immobile.
`SWU`	The maximum water saturation. This is the largest water saturation entry in a water saturation table.
`SGL`	The connate gas saturation. This is the smallest gas saturation entry in a gas saturation table.
`SGCR`	The critical gas saturation. This is the highest gas saturation for which the gas is immobile.

Table 60.1 Saturation table end points

Keyword	Description
SGU	The maximum gas saturation. This is the largest gas saturation entry in a gas saturation table.
SOWCR	The critical oil-in-water saturation. This is the highest oil saturation for which the oil is immobile in an oil-water system.
SOGCR	The critical oil-in-gas saturation. This is the highest oil saturation for which the oil is immobile in an oil-gas-connate water system

We shall refer to the saturation end points taken from the saturation tables in the input data file as the unscaled saturation end points.

The End-point Scaling option enables you to define new values for any of the above eight saturation end points for each grid cell, subject to maintaining a consistent set of saturation tables within each grid cell. The appropriate subset of the above eight end points is applicable in two phase runs. The new values defined using the scaling option will be referred to as the scaled end points.

When relative permeability and capillary pressure values need to be computed at a particular saturation, a linear transformation is used to determine the equivalent saturation to be used for look-up in the unscaled table.

For example, in a grid cell whose water saturation is SW, scaled connate water saturation is $SWCO$, scaled maximum water saturation is SWU, which belongs to a saturation region whose unscaled connate water is S_{wco} and whose unscaled maximum water saturation is S_{wmax}, the water capillary pressure P_{cow} is evaluated in the saturation table at the saturation, S_w', where

$$S_w' = S_{wco} + \frac{(SW - SWCO)(S_{wmax} - S_{wco})}{SWU - SWCO}$$

so that P_{cow} is evaluated by lookup in the input capillary pressure table using

$$P_{cow}(SW) = P_{cow}(S_w')(\text{table})$$

In addition it is possible to scale the relative permeability and capillary pressure values using the keywords KRW, PCW etc.

If the Hysteresis option is active, the above set of keywords is used to scale the drainage relative permeability curves. The imbibition curves can be scaled by using an analogous family of keywords:

- ISWL, ISWCR, ISWU
- ISGL, ISGCR, ISGU
- ISOWCR, ISOGCR.

Scaling of capillary pressure functions

The end points of the oil-water capillary pressure functions are taken to be the connate water and maximum water saturations. The values of P_{cow} at the saturation nodes SWL and SWU are preserved during the scaling process. Similarly, the gas-oil capillary pressure end points are SGL and SGU.

It is possible to modify the water (or gas) connate saturation for the capillary pressure scaling without modifying the connate saturation used for the oil relative permeability scaling.

ECLIPSE 100

The keywords SWLPC and SGLPC are used to input the connate saturation in this case. If both SWL and SWLPC keywords are present, the capillary pressure scaling will use SWLPC and the relative permeability scaling will use SWL.

Scaling of the capillary pressure values (vertical scaling)

It is possible to scale the maximum capillary pressure value on a grid block by grid block basis.

The maximum values are specified using the PCW and PCG keywords for the oil-water and gas-oil capillary pressures respectively.

In the water-oil case:

$$P_c = P_{ct}\frac{\mathrm{PCW}}{P_{cm}} \qquad \text{[EQ 60.1]}$$

where

P_{ct} capillary pressure from the table

P_{cm} maximum P_c in the table - at $S_w = S_{wco}$, connate water (this is the entry capillary pressure in the table)

PCW maximum P_c from the PCW data.

Scaling of relative permeability functions

There are two options available for the scaling of relative permeabilities. In the default case, the scaling process preserves relative permeabilities at two saturation nodes. The following end points are assumed for each phase relative permeability:

K_{rw}: SWCR and SWU

K_{rg}: SGCR and SGU

K_{row}: SOWCR and (1.0-SWL-SGL)

K_{rog}: SOGCR and (1.0-SWL-SGL)

An alternative scaling of relative permeabilities can be invoked by using the SCALECRS keyword in the PROPS section. The second form of scaling preserves the relative permeabilities at three saturation nodes. In 3-phase, oil-water or oil-gas runs, or runs which use the miscible flood option, the following end points are used:

K_{rw}: SWCR, (1.0-SOWCR-SGL) and SWU

K_{rg}: SGCR, (1.0-SOGCR-SWL) and SGU

K_{row}: SOWCR, (1.0-SWCR -SGL) and (1.0-SWL-SGL)

K_{rog}: SOGCR, (1.0-SGCR -SWL) and (1.0-SWL-SGL)

In gas-water runs, the following end points are used if the second form of relative permeability scaling is selected:

K_{rw}: SWCR, (1.0-SGCR) and SWU

K_{rg}: SGCR, (1.0-SWCR) and SGU

The second method for relative permeability scaling can be interpreted in two phase runs as preserving the values of relative permeabilities at both ends of the 2-phase mobile region. Convergence problems can arise in certain cases when using this form of scaling if the intermediate end point begins to approach the maximum saturation for one of the phases. For example, in an oil-water problem in which the residual oil saturation to water tends to zero at a certain depth, the saturation (1.0-SOWCR) begins to approach SWU and this can in turn lead to a discontinuity in the water relative permeability as $S_w \rightarrow S_{w\,\max}$.

ECLIPSE 100

You may limit this sharpness by using the SCALELIM keyword which sets an upper limit on the value of (1.0 - SOWCR) to be used in scaling the water relative permeability function. The SCALELIM keyword is optional and at present can only be applied to water relative permeabilities.

For example, we consider the scaling of the water relative permeability function $K_{rw}(SW)$ at a water saturation SW where the scaled and unscaled critical water saturations are $SWCR$ and S_{wcr}, scaled and unscaled oil-in-water critical saturations are $SOWCR$, and S_{owcr}, scaled and unscaled connate gas saturations are SGL and S_{gco}, and scaled and unscaled maximum water saturations are SWU and S_{wmax}.

With the two point scaling method, the water relative permeability $K_{rw}(SW)$ is then evaluated in the input saturation table at the new saturation S_w', where

$$S_w' = S_{wcr} + \frac{(SW - SWCR)(S_{wmax} - S_{wcr})}{SWU - SWCR} \qquad \text{[EQ 60.2]}$$

so that K_{rw} is evaluated by lookup in the input table using

$K_{rw}(SW) = K_{rw}(S_w')(table)$ for $SWCR \le SW \le SWU$. For $SW \le SWCR$ then $K_{rw}(SW) = 0$ and for $SW \ge SWU$ then $K_{rw}(SW) = K_{rwmax}(table)$.

With the three point scaling method, if SR is the displacing critical saturation of the associated phase saturation (SR),

$\mathrm{SR} = 1 - \mathrm{SOWCR\text{-}SGL}$ in water/oil or gas/oil/water runs

$\mathrm{SR} = 1 - \mathrm{SGCR}$ in gas/water runs,

and S_r is the value of the displacing critical saturation in the table

$S_r = 1 - S_{owcr} - S_{gco}$ in water/oil or gas/oil/water runs

$S_r = 1 - S_{gcr}$ in gas/water runs.

The two cases are:

$SWCR \le SW \le SR$

$$S_w' = S_{wcr} + \frac{(SW - SWCR)(S_r - S_{wcr})}{SR - SWCR} \qquad \text{[EQ 60.3]}$$

$SR \le SW \le SWU$

$$S_w' = S_r + \frac{(SW - SR)(S_{wmax} - S_r)}{SWU - SR} \qquad \text{[EQ 60.4]}$$

and also for $SW \le SWCR$ then $K_{rw}(SW) = 0$, and for $SW \ge SWU$ then

$K_{rw}(SW) = K_{rwmax}(table)$. [EQ 60.5]

The two- and three-point scaling equations are similar for K_{rg}, K_{row} and K_{rog} with alternate definitions of the critical and displacing critical saturations.

Scaling of the relative permeability values (vertical scaling)

It is possible to scale the relative permeability at the maximum phase saturation and at the critical/residual saturation of the associated phase.

The scaled relative permeabilities can be specified on a block by block basis, or alternatively the depth variation can be specified using the `ENKRVD` and `ENDNUM` keywords. The `KRW`, `KRG`, `KRO` keywords and their derivatives are used to set the relative permeabilities at the maximum phase saturation, while the `KRWR`, `KRGR`, `KRORW`, `KRORG` keywords and their derivatives are used to set the relative permeabilities at the critical/residual saturation of the associated phase.

The resultant scaling of the **water** relative permeability is shown below. **Gas** and **oil** relative permeabilities are scaled in an **equivalent** manner.

If the `KRWR` keyword has not been used, the `KRW` keyword has the effect of scaling the relative permeability value calculated from the appropriate saturation table after the scaled saturation end points have been accounted for. Hence:

$$K_{rw} = K_{rw}(\text{table})\left(\frac{\text{KRW(grid block)}}{K_{rw\,\max}(\text{table})}\right) \qquad \text{[EQ 60.6]}$$

The $K_{rw\ \max}(\text{table})$ is taken to be the value at either the maximum saturation of the saturation table or at SWU if this has been specified.

If the KRWR keyword has been used, then the scaling will honor the K_r at the critical saturation (SR) of the associated phase.

$\text{SR} = 1 - \text{SOWCR-SGL}$ in water/oil or gas/oil/water runs

$\text{SR} = 1 - \text{SGCR}$ in gas/water runs

Hence the two cases are:

1 $SWCR \le SW \le SR$

$$K_{rw}(S) = K_{rw}(S')\frac{\text{KRWR(grid block)}}{K_{rw}(S_r)(\text{table})} \qquad \text{[EQ 60.7]}$$

2 $SR \le SW \le SWU$

$$K_{rw}(S) = \text{KRWR} + \frac{(K_{rw}(S')(table) - K_{rw}(S_r)(table)))}{(K_{rw\max}(table) - K_{rw}(S_r)(table))}(KRW - KRWR) \qquad \text{[EQ 60.8]}$$

If the value of $K_{rwmax} = K_{rw}(SR)$, a linear function is assumed between KRWR and KRW. If the table end points are the same, that is $SR = SWU$ in the water case, then the action taken depends on whether the alternative scaling method is being used (see the SCALECRS keyword). If the alternative scaling is active, then both KRWR and KRW are honored with a linear function between the two points. If the default scaling is used, then the KRWR is ignored.

The two-point scaling is illustrated in Figure 60.1:

Figure 60.1 Two-point scaling

and the three-point scaling is exemplified in Figure 60.2:

Figure 60.2 Three-point scaling

Miscellaneous points

The End-point Scaling option is activated using the keyword ENDSCALE in the RUNSPEC section. The directional scaling options are activated by specifying the appropriate arguments of ENDSCALE.

The end point saturations may be supplied in one of two ways:

- by cell, using the keywords SWL, SWCR, SWU, SGL, SGCR, SGU, SOWCR, SOGCR; or,
- by depth, using the ENPTVD keyword.

The cell keywords need not specify data for the whole field - any values that are not set do not cause an error, (as for PORO or DX), but imply that they are to be obtained from the default values for the cell, or from ENPTVD, if present.

If SWL, SOGCR and ENPTVD are used, values taken from ENPTVD are used to fill in any data that is not set, as long as the endpoint table number is not 0. End point table numbers are set in the REGIONS section, using ENDNUM, and default to 1, but a value of 0 may be entered.

The overall process is thus:

1 If keywords SWL and SOGCR are present, use the value

2 If not, and ENPTVD is present with non-zero ENDNUM, use this

3 Otherwise, use defaults obtained from tables.

The values of the end-point arrays SWL and SOGCR may be printed out for each cell using RPTSOL with the appropriate mnemonic, and output for plotting with GRAF using RPTRST or OUTSOL.

ECLIPSE 100

If the Hysteresis model is active, the imbibition end points can be specified using the IMPTVD keyword.

When the End-point Scaling option is activated, the scaling keywords are entirely optional. If a particular end point keyword is not present in the input data file, the unscaled end points are substituted and used in the scaling calculations.

By default only scaled quantities are reported in any INIT file generated. To display the actual end points used in the simulation for each grid cell the keyword FILLEPS should be added.

The directional capability of the scaling option may be used to control the saturation functions which apply to the different faces of each grid cell. The facility is similar to the directional relative permeability option and allows saturation end points to be defined on the X,Y and Z faces of each cell or, in the highly directional case, on the +X,-X,+Y,-Y,+Z and -Z faces of each cell. Note that the saturation table look-ups required by the well model and equilibration algorithm are performed using the SATNUM table for each cell. If directional scaling is active, the isotropic end points are used to rescale the saturation tables within the well model and the equilibration algorithm. Note that if the directional form of an end point is not explicitly defined for a given cell, then it is taken from the relevant table, and is **not** defaulted to the isotropic form (except for pseudo-critical end points arising from the initial fluid mobility correction enabled by entering MOBILE in keyword EQLOPTS).

Data checking is performed on the rescaled saturation tables in each cell. The checks ensure that the connate, critical and maximum saturations are in increasing order and that the rescaled tables are consistent for each phase. In the case of directional scaling, the data checking is performed on the rescaled tables for each grid face.

The scaling calculation is performed on the saturation tables used for flow between grid cells and well connections. If, however, the saturation table for the well connection differs from that of the connecting cell then the scaling option is disabled. In this case, any scaling required in the connection saturation table should be done using the COMPRP keyword. If the table numbers are equal then the end points used are those of the connecting cell. These end points will then be over-ridden by any subsequent end points defined using the COMPRP keyword.

ECLIPSE 100 In the present version of ECLIPSE 100 the SWLPC and SGLPC values cannot be supplied using the ENPTVD depth type keywords.

ECLIPSE 100 The End-point Scaling option should not be used at present with the Vertical Equilibrium option.

ECLIPSE 300 The saturation table end-points can depend on temperature in a thermal simulation using the keywords ENKRVT, ENPCVT and ENPTVT.

ECLIPSE 300 The saturation table end-points can depend on composition using the keywords ENKRVC, ENPCVC and ENPTVC.

Output of scaled end-points

ECLIPSE 300 When the saturation end-points scaling depends on temperature or composition (ENPTVT and ENPTVC keywords), one can output the scaled values using the following keywords.

To the PRINT file:

```
RPTSOL
SWL SWCR SWU SGL SGCR SGU SOWCR SOGCR/
RPTSCHED
SWL SWCR SWU SGL SGCR SGU SOWCR SOGCR/
```

To the SUMMARY file

BSWL	Block minimum water saturation
BSWCR	Block critical water saturation
BSWU	Block maximum water saturation
BSGL	Block minimum gas saturation
BSGCR	Block critical gas saturation
BSGU	Block maximum gas saturation
BSOWC	Block critical oil-in-water saturation
BSOGC	Block critical oil-in-gas saturation

To the RESTART file:

```
RPTRST
SWL SWCR SWU SGL SGCR SGU SOWCR SOGCR/
```

The initial fluid mobility correction

In the production of a reservoir model it is important to ensure that the model is initialized with the correct mobile quantities of each fluid as well as the correct quantities of total fluid-in-place. If this is not done the recovery factors of each fluid are predicted erroneously.

The fine scale equilibration algorithm predicts too small an initial mobile oil-in-place even though the fluid volumes-in-place are predicted correctly. To see why the mobile fluids-in-place are underestimated consider the fine scale equilibration of an oil-water system. The variation of water saturation with depth for a typical grid cell that intersects the oil-water transition zone is shown in Figure 60.3.

Figure 60.3 Water saturation versus depth for a typical grid cell that intersects the oil-water transition zone

The water saturation at the value X is $1.0 - \text{SOWCR}$. V is the pore volume.

The initial volume of oil-in-place is $V(A+B)$

The correct volume of mobile oil-in-place is VA

The volume of mobile oil-in-place calculated by the fine scale equilibration procedure is $V(A-C)$, which leads to an underestimate of the total volume of mobile oil of VC.

The initial volume of mobile oil-in-place is underestimated in cases where there is a negligible transition zone. Note that the problem may also arise in the calculation of the initial mobile volumes of other phases.

A facility based on the End-point Scaling option is available in ECLIPSE to provide correct estimates of both the total and mobile fluids-in-place. The method uses the fine scale equilibration option and computes pseudo values for the critical saturation of each phase in each grid cell. The procedure adopted is as follows:

- For each of the fine scale integration layers (i=1,...,N) the saturation, S_{pi}, of each phase, p, and its critical saturation $S_{p,cr,i}$ is determined
- The volume of mobile phase within each layer, M_{pi}, is then found

  $M_{pi} = 0$ if $S_{pi} < S_{p,cr,i}$

$M_{pi} = V_i(S_{pi} - S_{p,cr,i})$ if $S_{pi} > S_{p,cr,i}$

where V_i is the pore volume of layer i

- The total mobile phase volume, M_p, in each cell is then determined using $M_p = \Sigma_i M_{p,i}$. The total phase volume, $T_p = \Sigma_i V_i S_{pi}$.
- The average phase saturation in the cell is $S_p' = T_p/(\Sigma_i V_i)$ and the average mobile phase saturation is $M_p' = M_p/(\Sigma_i V_i)$.
- For cells in a which a mobility-immobility transition occurs due to the saturation variation with depth, the pseudo-critical phase saturation is set to

$$S'_{p,cr,i} = S'_p - M'_p . \qquad \text{[EQ 60.9]}$$

- The remaining cells in the grid contain either mobile or immobile fluid throughout their depth. In this case

$$S'_{p,cr,i} = (\Sigma_i V_i S_{p,cr,i})/(\Sigma_i V_i) . \qquad \text{[EQ 60.10]}$$

This calculation of each pseudo critical phase saturation ensures that the critical saturations are only modified in the vicinity of the fluid contacts in the initial state. During the simulation phase, the relative permeability curves are rescaled using the pseudo critical saturations computed above. The pseudo critical phase saturations are assumed to be isotropic; if Directional Scaling is enabled, then the directional counterparts of the pseudo critical phase saturations are set equal to their isotropic form.

The mobile fluid-in-place calculation is activated using the `MOBILE` flag in the keyword `EQLOPTS` of the `RUNSPEC` section. In a three-phase run, the initial mobile fluid-in-place correction is applied to all the critical end points `SOWCR`, `SOGCR`, `SWCR` and `SGCR`. In an oil-water system it is applied only to `SOWCR` and `SWCR`. In a gas-oil system it is applied only to `SOGCR` and `SGCR`. In a gas-water system it is applied only to `SWCR` and `SGCR`. Output of the processed end points used in the simulation can be requested with the mnemonics `SOWCR`, `SOGCR`, `SWCR` and `SGCR` (switches 24-27) in the `RPTSOL` keyword (the output is headed by mnemonics `PSOWCR`, `PSOGCR` etc.).

End point variations through a transition zone

In some reservoirs it is necessary to model a depth variation of the critical saturation of one phase such that the critical saturation equals the phase saturation providing the phase saturation is less than a threshold (See Figure 60.4). Although the `ENPTVD` keyword could be used to model the profile, a more convenient method, which also gives the correct initial mobile fluids-in-place, is to initially specify a constant critical saturation with depth and set $S'_{p,\mathrm{cr},i} = S'_p - M'_p$ regardless of the phase mobility in each cell. This calculation can be invoked by using the `TZONE` keyword in the `PROPS` section.

Figure 60.4 Modeling a depth variation of critical saturation

To use the facility and achieve the result shown, specify the keyword ENDSCALE in the RUNSPEC section, turn on the second data item in the TZONE keyword, and set the critical water saturation to $S^*_{w,\,crit}$. Output of the processed end points used in the simulation can be requested with the mnemonics SOWCR, SOGCR, SWCR and SGCR (switches 24-27) in the RPTSOL keyword at all depths.

The TZONE keyword allows control of the calculation for each phase separately; if the water switch is set true then the critical water saturation will be corrected to the initial water saturation where this is below the input critical saturation.

Leverett J-function

The Leverett J-Function is a dimensionless group that allows the capillary pressure function to be correlated with the rock properties.

$$J(S_w) = \frac{P_c}{\mathrm{ST}}\left(\frac{K}{\phi}\right)^{\frac{1}{2}} \qquad \text{[EQ 60.11]}$$

where

P_c capillary pressure

ST surface tension

K rock permeability

ϕ rock porosity.

If the JFUNC or the JFUNCR keyword is used in the GRID section then the capillary pressure is calculated as follows:

The capillary pressure is then calculated as follows:

$$P_c = J(S) \cdot \mathrm{ST} \cdot \left(\frac{\phi}{K}\right)^{\frac{1}{2}} \cdot U_{\mathrm{const}} \qquad \text{[EQ 60.12]}$$

Where

$J(S)$

J-function. This is input as a function of saturation in the column for capillary pressure in the SWFN or SGFN table

ST Surface tension - specified in the JFUNC or the JFUNCR keyword

ϕ cell porosity

K cell permeability

U_{const} constant depending on the unit system employed:

= 0.318316 (METRIC units, pressure in BARS, perm in MD)

= 4.61678 (FIELD units, pressure in PSIA, perm in MD)

= 0.314153 (LAB units, pressure in ATMOS, perm in MD)

= 0.314153 (PVT-M units, pressure in ATMOS, perm in MD)

The permeability is taken as $K = (K_x + K_y)/2$, unless the model is a cross section, in which case K is taken as the appropriate horizontal permeability.

$$\text{Scaling Factor} = F = \text{ST} \cdot \left(\frac{\phi}{K}\right)^{\frac{1}{2}} \cdot U_{\text{const}} \qquad \text{[EQ 60.13]}$$

The JFUNC or the JFUNCR keyword in the GRID section causes the scaling factor (F) to be computed from the grid data and stored in the PCW or PCG array. The scaling factor can be output by using the mnemonic ENDPT in the RPTPROPS keyword.

ECLIPSE 100

Note that, if the surface tension is required to be a function of pressure, the pressure dependent capillary pressure option (STOW and STOG keywords) can be used to modify the surface tension. In this case:

$$P_c = J(S) \cdot \text{ST} \cdot \frac{\text{ST}(P)}{\text{ST}_{\text{ref}}} \cdot \left(\frac{\phi}{K}\right)^{\frac{1}{2}} \cdot U_{\text{const}} \qquad \text{[EQ 60.14]}$$

where ST(P) and ST_{ref} are input using the STOG or STOW keywords.

Matching initial water distribution

The water-oil capillary scaling facility can be used to honor an input water distribution map.

The required water distribution is entered using the SWATINIT keyword in the PROPS section. ECLIPSE automatically computes the PCW array such that:

$$P_{cw}(S_{w\ \text{init}}) = P_{\text{oil}} - P_{\text{wat}} \qquad \text{[EQ 60.15]}$$

for the appropriate equilibration region.

Where

P_{oil} is the oil phase pressure

P_{wat} is the water phase pressure

$S_{w\ \text{init}}$ is the required water saturation from SWATINIT

If $P_{\text{oil}} - P_{\text{wat}} < 0$, the PCW value is left unchanged and the equilibrated S_W value will be the maximum water saturation, SWU. The phase pressures P_{wat} and P_{gas} are determined by equilibration using EQUIL and the supplied PVT data. See OPTIONS switch 74 and OPTIONS3 switch 143 for a different scaling cases.

If the water distribution and the input capillary pressure curves are very inconsistent, the scaling required may be extreme. The scaled maximum capillary pressure can be output to the PRINT file using the mnemonic SWATINIT (30th switch) in the RPTSOL keyword, and is automatically output to the INIT file. These arrays should be checked to ensure that the scaling is physically reasonable. The PPCWMAX keyword could be used to limit the maximum capillary pressure scaling.

If the SWATINIT keyword is used with the J-function option (JFUNC or the JFUNCR keyword) then in regions of the field where SWATINIT has been set the J-function data will be ignored and the SWATINIT saturations honored instead. If the PCW keyword is used, then the SWATINIT distribution will override the input PCW where $P_{\text{oil}} - P_{\text{wat}} > 0$.

In a gas condensate study with no initial liquid phase, where the gas-oil contact coincides with the oil-water contact in the EQUIL keyword, then PCW is taken as the hydrocarbon capillary pressure and the SWATINIT distribution will override the input PCW where $P_{\text{gas}} - P_{\text{wat}} > 0$.

Consistency requirements

The oil, water and gas saturation tables, for each particular saturation table number region, must obey certain consistency requirements. Essentially these are described for unscaled tables in "Consistency requirements" on page 899. The same requirements apply to the scaled endpoints.

1 SGU must not exceed 1.0 - SWL.

If this condition is violated, the gas saturation in the gas cap is re-set to 1.0 -SWL, to prevent a negative oil saturation. Normally, if there is no oil in the gas cap, SGU = 1.0 - SWL.

2 SGL must not exceed 1.0 - SWU

If this condition is violated, the gas saturation in the water zone is re-set to 1.0-SWU, to prevent a negative oil saturation. Normally, there is no initial free gas below the gas cap and the water zone is fully saturated with water, thus SGL = 0.0 and 1.0 - SGL = 1.0

Otherwise, phases can be mobile even at zero saturation and there is nothing to stop saturations going negative.

The $K_{rw\ \max}(\text{table})$ is taken to be the value at either the maximum saturation of the saturation table or at SWU if this has been specified.

If the KRWR keyword has been used, then the scaling honors the K_r at the critical saturation (SR) of the associated (that is, displacing) phase. When the alternative three-point end point scaling method is invoked using keyword SCALECRS, the scaling uses the three nodes SWCR, SR and SWU where

$\text{SR} = 1 - \text{SOWCR(grid block)} - \text{SGL(grid block)}$ in water/oil or gas/oil/water runs

$\text{SR} = 1 - \text{SGCR(grid block)}$ in gas/water runs

ECLIPSE 100

In order to ensure that at least one fluid remains mobile in the three-phase ternary diagram, ECLIPSE also checks for the following:

3 $\text{SOWCR} + \text{SWCR} < 1.0$

4 $\text{SOGCR} + \text{SGCR} + \text{SWL} < 1.0$

ECLIPSE issues a warning if these conditions are not satisfied. When SCALECRS is used, this would lead to an error because in an oil-water run, for example, K_{rw} would be scaled between SWCR and 1.0 - SOWCR, which is a null range if the first condition is not satisfied.

Example of end point scaling

The following will produce the same results in a run:

With end point scaling

```
SGL
50*0.00 50*0.00 /
SGCR
50*0.10 50*0.20 /
SGU
50*0.99 50*0.98 /
SOWCR
50*0.05 50*0.22 /
SOGCR
50*0.08 50*0.21 /
SWL
50*0.01 50*0.02 /
SWCR
50*0.20 50*0.22 /
SWU
50*1.0 50*1.0 /
SGFN
0.0 0.0 0.0
1.0 1.0 0.5 /
SOF3
0.00 0.0 0.0
1.00 0.8 0.8 /
SWFN
0.0 0.0 1.0
1.0 1.0 0.0 /
```

With multiple tables

```
SGFN
0.0  0.0 0.0
0.1  0.0 1*
0.99 1.0 0.5 /
0.0  0.0 0.0
0.2  0.0 1*
0.98 1.0 0.5 /
SOF3
0.00 0.0 0.0
0.05 0.0 0.0
0.08 1*  0.0
0.99 0.8 0.8 /
0.00 0.0 0.0
0.21 0.0 0.0
0.22 0.0 1*
0.98 0.8 0.8 /
SWFN
0.01 0.0 1.0
0.2  0.0 1*
1.0  1.0 0.0 /
0.02 0.0 1.0
0.22 0.0 1*
1.0  1.0 0.0 /
REGIONS
SATNUM
50*1 50*2 /
```

Shale Gas

Chapter 61

Introduction

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL

Shale gas reservoirs often have a matrix porosity system where the transient behavior in the matrix becomes important. Some of the gas might be adsorbed on the surface of the shale and some exists as a free gas in the matrix pore structure. In order to model such reservoirs the dual/multi porosity option can be used together with the coal bed methane model for adsorbed gas on the rock formation. See "Multi porosity" on page 627 and "Adsorption model in ECLIPSE 300" on page 163.

Note The Coal Bed Methane License is required

There are two models for flow of adsorbed gas into a simulation cell.

- Instant sorption model. The amount of adsorbed gas is in instant equilibrium with the free gas composition and pressure.
- Time dependent sorption model.

To choose between the time dependent or instant sorption models the keyword `CBMOPTS` is used.

Grid cells with adsorption

In order to specify cells having an adsorbed gas the `COALNUM` keyword is used for both methods. It is possible to redefine the coal region numbers by the keyword `COALNUMR`.

Time dependent sorption

For the time dependent method a simulation cell either contains free gas in a pore space or adsorbed gas in the rock. The rock is represented by one simulation cell and the pore volume by a connecting simulation cell. Darcy flow through a rock cell is not permitted. The usual pore volume of a cell now represents the rock volume where the imaginary micro pore space flow is accounted for by a diffusive flow equation. A cell having a non-zero coal region number as set by COALNUM hence needs to specify a porosity value that corresponds to a rock fraction value. For cells having a zero coal region number the porosity value correspond to the pore volume fraction.

Note It is recommended that the ROCKFRAC keyword is used to set the coal volume and not the porosity keyword PORO.

If pore volume-pore volume connections exists a permeability value also need to be input in order to compute the transmissibility between the matrix subgrid cells. This is usually done by PERMX.

Note If the multi porosity option is used together with time-dependent sorption a coal cell needs to connect to a pore volume cell. Adjacent coal subcells are not permitted.

Matrix discretization (NMATOPTS)

Using the time dependent method the primary matrix subgrid cells are used to define both the total pore volume and the total coal volume that is partitioned. The primary matrix subgrid cells need to be defined as coal by COALNUM in order to tell that the matrix contains adsorbed gas. To redefine the matrix subgrid cells with a coal region number the keyword COALNUMR should be used.

The block volume of each simulation cell will be the same. This mean that the diffusivities between the matrix subgrid cells are not effected by the matrix discretization. To alter the diffusivities SIGMAV can be specified for the subcells or by DIFFCBM if several coal region numbers has been specified.

The subgrid cells transmissibilities are constructed as if the pore volume cells are connected, if the pore volume cells connects only to coal cells the transmissibilities will be reset to zero.

Instant sorption

For the instant desorption model the pore volume has the usual interpretation, and the adsorption desorption process is represented as a source sink term into the matrix pore system. It should be noted that the coal volume of each matrix subgrid cell is computed from the average grid cell volume of each coal cell, where the pore volume of all the connected matrix and fracture grid cells are subtracted. If ROCKFRAC is specified for a non-coal cell, this volume is also subtracted from the coal volume. The coal volume is then distributed evenly between the coal cells.

Matrix discretization (NMATOPTS)

In combination with the multi porosity option the matrix subgrid cells are assigned pore volumes and rock/coal volumes automatically if the NMATOPTS keyword is present. The primary (outer) matrix pore and rock volume is partitioned among the matrix subgrid cells. Note that the pore volume of the fractures are subtracted from the rock volume. The primary matrix subgrid cells need to be defined as coal by COALNUM in order to tell that the matrix contains adsorbed gas. To redefine the matrix subgrid cells with a coal region number the keyword COALNUMR should be used. Only the pore volume and coal volumes as defined by the primary matrix subgrid cells are used to generate the matrix subgrid cells coal and pore volumes.It should be noted that permeabilities should be supplied for the primary matrix subgrid cells only in order to define the transmissibilities between the matrix subgrid cells.

The partitioning of the rock follows the pore volume partitioning. If not all the matrix subcells are assigned a coal region number, the rock volume is maintained by lumping the volume onto the following cell with a non-zero coal region number, as illustrated below.

Note that having different adsorption properties for the different subcells also means that the partitioning of the volumes between the subcells becomes important.

It is possible to assign a rock density for each coal region number by the keyword CRNDENS in order to assign different density to the matrix subgrid cells.

Coal and pore volumes

If the ROCKFRAC keyword is used we have the following pore volume and coal volume definitions for the primary matrix subgrid cells:

Sorption model	Pore volume primary matrix subgrid, PV_m	Coal volume primary matrix subgrid, CV_m
Instant	$V_m \phi_m \times NTG_m$	$V_m r_m$
Time dependent	$V_m \phi_m \times NTG_m$	$V_m r_m$

The following table shows the pore volume and coal volume definitions for the primary matrix subgrid cells if ROCKFRAC is not used.

Sorption model	Pore volume primary matrix subgrid, PV_m	Coal volume primary matrix subgrid, CV_m
Instant	$V_m \phi_m \times NTG_m$	$V_m - PV_m - PV_f$
Time dependent	$V_m - CV_m - PV_f$	$V_m \phi_m \times NTG_m$

where

V_m , V_f Bulk volume, matrix or fracture

ϕ_m , ϕ_f Porosity, matrix or fracture

r_m Rock fraction value in matrix

NTG_m , NTG_f Net to gross, matrix or fracture

Examples

The following data sets come with the ECLIPSE installation:

Table 61.1 Shale gas example files

File	Description
MPORO.DATA	Gas water case (matrix discretization)
SHALEGAS1.DATA	Shale gas
SHALEGAS1T.DATA	Shale gas with time dependent sorption

Solid Phase

Chapter 62

Introduction

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

Solid components can be used with the Blackoil, Equation of State (EoS), K-value, or Thermal options. Within the reservoir, these components remain in the solid phase throughout the simulation. However, solid components can be converted to fluid components through a chemical reaction. A solid component could be used, for example, to simulate production and take up of coke during combustion.

Solids can be either suspended in the oil phase, or adsorbed on to the formation. Adsorption isotherms can be defined which determine the fraction of solids in suspension.

Suspended solids are transported with the flow.

Solids that adsorb to the formation reduce the pore volume available for flow. Permeability multipliers can be used to model this effect.

Solid keywords

RUNSPEC section keywords

The keywords are:

Table 62.1 RUNSPEC section

Keyword	Description
SOLID	Requests Solids option

GRID section keywords

The keywords are:

Table 62.2 GRID section

Keyword	Description
ROCKDEN	Rock density

PROPS section keywords

The keywords are:

Table 62.3 PROPS section

Simulator	Keyword	Description
	CVTYPE	Define solid phase components
	CVTYPES	Define solid phase components at surface conditions
Reactions only	REACSORD	Reaction rate solid order
	SCREF	Solid compressibility
	SDREF	Solid density at reference conditions
	SOLIDADS	Solid adsorption table
	SOLIDMMC	Solid mobility multiplier (as a function of adsorbed concentration)
	SOLIDMMS	Solid mobility multiplier (as a function of adsorbed saturation)
Thermal only	SPECHS	1st coefficient of solid specific heat
Thermal only	SPECHT	2nd coefficient of solid specific heat
	SPREF	Reference pressure for solid density calculation
	STREF	Reference temperature for solid density calculation
	STHERMX1	Solid thermal expansion coefficient

SOLUTION section keywords

The keywords are:

Table 62.4 SOLUTION section

Keyword	Description
SMF	initial solid molar fraction
SSOLID	initial solid saturation

SUMMARY section keywords

Solid properties and production data can be output to the summary file. Keywords are given in Table 62.5 and Table 62.6.

Table 62.5 Solid properties that can be output to the summary file

Keyword	Output
BBSOL	Solid reservoir molar density
BDENS	Solid Density
BHSOL	Solid enthalpy (Thermal option)
BSDEN	Solid DENsity (same as BDENS)
BSSAT	Solid SATuration
BSSOLID	Solid Saturation (same as BSSAT)
BSSFRAC	Suspended solids fraction
BSMMULT	Solid mobility multiplier

Table 62.6 Solid production data

Field	Group	Well	Output
FSMR	GSMR	WSMR	Solid mass rate
FSMT	GSMT	WSMT	Solid mass total
FSVR	GSVR	WSVR	Solid volume rate
FSVT	GSVT	WSVT	Solid volume total

SCHEDULE section keywords.

Table 62.7 Solid properties that can be output with RPTRST, RPTSCHED and RPTSOL

Keyword	Output
BSOL	Solid reservoir molar densities
DENS	Solid reservoir density
HSOL	Solid enthalpies (Thermal option)
SSOLID	Solid saturations
SSFRAC	Suspended solids fraction
SMMULT	Solid mobility multiplier

Solid properties

Solid phase

The SOLID keyword must be used if solid components are present.

Solid-only components

The CVTYPE keyword can be used to specify solid phase components. The CVTYPES keyword can be used to specify solid phase components at surface conditions.

These solid components are ignored by the flash - they cannot enter the liquid or gas phase. The only way they can change is through a chemical reaction, when they can become a different component.

Solid phase properties

Properties for solid-only components are defined using special keywords which are described in this section.

Solid phase density

The specific volume of solid phase components V_{sol} is calculated as

$$V_{sol} = m_S / b_s$$

where

m_S is the total number of moles in the solid phase per unit pore volume.

b_s is the solid molar density given by

$$1/b_s = \sum_{c=1}^{N_c} x_s^c \cdot V_{sol}^c$$

where x_s^c is the mole fraction of component c in the solid phase. The component molar solid phase volume V_{sol}^c is given by:

$$V_{sol}^c = \frac{MW^c}{\rho^c}$$

where MW^c is the molecular weight of component c given by the MW keyword, and the component solid phase density ρ^c is given by:

$$\rho^c = \frac{\rho_{ref}^c}{(1 + C_{T1}(T - T_{ref}))(1 - C_p(P - P_{ref}))}$$

The reference density ρ_{ref}^{c} is defined by the SDREF keyword; the standard temperature and pressure, T_{ref} and P_{ref} are defined by keywords STREF and SPREF (or by STCOND); the thermal expansion coefficient C_{T1} is defined by the STHERMX1 keyword; and the component isothermal compressibility C_P is defined by the SCREF keyword.

Trapped gas

Thermal only

When modeling foamy oil it may be necessary to model a component that has a gas-like compressibility in the solid phase (see "Foamy oil" on page 1068). This can be done using keywords CCTYPE. In this case the component molar solid volume is given by the gas law

$$V_{sol}^{c} = \frac{Z^{c} RT}{P} \qquad \text{[EQ 62.1]}$$

where R is the gas constant and the compressibility terms Z^c can be set for each component using the ZFACTOR keyword.

Keyword CCTYPES can be used to set the compressibility at surface conditions.

Specific heats

Thermal only

The specific heat C_{sol}^{c} of a component c in the solid phase is given by

$$C_{sol}^{c} = C_{sol_1}^{c} + C_{sol_2}^{c} \cdot (T - T_{st})$$

where the coefficients are defined by keywords SPECHS and SPECHT. T_{st} is the standard temperature defined by STCOND. The specific heat must always be positive.

Solid Saturations

When solids are present the pore volume V_p is taken up by both fluids and solids

$$V_p = V_f + V_s = V_p \cdot (V_{flu} + V_{sol})$$

where the specific fluid volume is given by

$$V_{flu} = V_{oil} + V_{gas} + V_{wat}$$

Normalized fluid saturations are defined as fractions of the fluid volume

$$S_o = \frac{V_{oil}}{V_{flu}},\ S_g = \frac{V_{gas}}{V_{flu}},\ S_w = \frac{V_{wat}}{V_{flu}}$$

The normalized saturations satisfy the equation:

$$S_o + S_g + S_w = 1$$

Saturations can be defined as fractions of the pore volume. These saturations are used in the permeability calculations and in the output reports.

$$\hat{S}_s = V_{sol}$$

$$\hat{S}_o = V_{oil} = V_{flu} \cdot S_o = (1 - \hat{S}_s) \cdot S_o$$

$$\hat{S}_g = V_{gas} = V_{flu} \cdot S_g = (1 - \hat{S}_s) \cdot S_g$$

$$\hat{S}_w = V_{wat} = V_{flu} \cdot S_w = (1 - \hat{S}_s) \cdot S_w$$

These modified saturations satisfy the equation:

$$\hat{S}_o + \hat{S}_g + \hat{S}_w + \hat{S}_s = 1$$

The solid saturation $\hat{S}$ and the multiplier $1 - \hat{S}$ are used to calculate permeability reduction and initial conditions.

Adsorption

Solid components can be modeled either as suspended in the oil phase, or as adsorbed by the formation rocks.

The suspended solid concentration C_s is defined as the weight of solids per unit pore volume

$$C_s = f_s \cdot W_s \qquad \text{[EQ 62.2]}$$

where f_s is the fraction of solids in suspension, W_s is the total weight of solids per unit pore volume. The adsorbed solid concentration C_a is defined as the weight of solids per weight of rock:

$$C_a = (1 - f_s) \cdot W_s / W_R \qquad \text{[EQ 62.3]}$$

where W_R is the weight of rock per unit pore volume:

$$W_R = \rho_R \cdot (1 - \theta) / \theta \qquad \text{[EQ 62.4]}$$

θ is the porosity and ρ_R is the rock density given by the `ROCKDEN` keyword.

Keyword `SOLIDADS` is used to define an adsorption function:

$$C_a = C_a(C_s) \qquad \text{[EQ 62.5]}$$

which allows [EQ 62.2], [EQ 62.3] and [EQ 62.4] to be solved for the suspended solids fraction f_s.

There are two special cases

- If the `SOLIDADS` keyword is not used, it is assumed that all solids are adsorbed, $f_s = 0$.
- If the `SOLIDADS` keyword is used, but the rock density is zero, then it is assumed that all solids are in suspension, $f_s = 1$.

Output of the suspended solids fraction

The suspended solids fraction f_s can be output to the print file or restart file by using the mnemonic `SSFRAC` with the `RPTRST`, `RPTSCHED` and `RPTSOL` keywords. Values of the suspended solids fraction in individual grid blocks can be output to the summary file by using the mnemonic `BSSFRAC`.

Adsorption example: the Langmuir isotherm

An example of an adsorption function is given by the Langmuir isotherm, which is used to describe the adsorption of gas at a fixed temperature. The same function can also be used to describe solid adsorption. The formula for the isotherm is derived by assuming the rate at which solids are adsorbed by the rocks is equal to the rate at which solids are desorbed.

If the rate at which solids are adsorbed per unit pore volume R_a is given by:

$$R_a = K_a \cdot C_s \cdot (C_{amax} - C_a),$$

where C_{amax} is the maximum adsorbed concentration and R_a is the adsorption rate, and if the rate at which solids are desorbed per unit pore volume is given by:

$$R_d = K_d \cdot C_a,$$

where K_d is the deposition rate;

then setting the adsorption rate equal to the desorption rate leads to an adsorption function given by the Langmuir isotherm:

$$C_a = C_{amax} \cdot \frac{C_s}{K + C_s}$$

where K is the ratio of the desorption rate to the adsorption rate:

$$K = K_d / K_a$$

```
--Langmuir isotherm, Camax = 0.01kg/kg; K = .05 kg/m3
SOLIDADS
--Cs       Ca
--kg/m3    kg/kg
  0.00     0.00000
  0.01     0.00167
  0.0125   0.00200
  0.0333   0.00400
  0.05     0.00500
  0.075    0.00600
  0.15     0.00750
  0.20     0.00800
  0.35     0.00875
  0.45     0.00900
  0.95     0.00950
/
```

Permeability reduction caused by plugging

When a solid phase is present the mobility M_p^c for component c in phase p is given by

$$M_p^c = x_p^c \cdot k_s \cdot k_{rp}(S_p) \cdot \frac{b_p}{\mu_p}$$

where

x_p^c is the mole fraction of component c in phase p,

k_s is the mobility multiplier due to the solids,

k_{rp} is the relative permeability of phase p,

S_p is the normalized saturation of phase p,

b_p is the molar density of phase p, and

μ_p is the viscosity of phase p.

The term k_s represents the reduction in mobility caused by adsorbed solids plugging the pores. There are two ways of entering k_s data. The SOLIDMMC keyword allows the mobility multiplier to be entered as a function of the adsorbed concentration:

$$k_s = k_{sc}(C_a).$$

Alternatively the SOLIDMMS keyword can be used to enter the mobility multiplier as a function of adsorbed saturation S_a:

$$k_s = k_{ss}(S_a),\ S_a = (1 - f_s) \cdot \hat{S}_s.$$

Output of the mobility multiplier

The mobility multiplier k_s can be output to the print file or restart file by using the mnemonic SMMULT with the RPTRST, RPTSCHED and RPTSOL keywords. Values of the mobility multiplier in individual grid blocks can be output to the summary file by using the mnemonic BSMMULT.

Transport of suspended solids

Solids adsorbed to the rock are immobile, but the suspended fraction will be transported with the oil phase. It is assumed that the concentration of suspended solids is small, so that they can be treated as a tracer - i.e. the oil flow is not modified by the presence of suspended solids, and the mobility of a solid component c is given by

$$M_o^{\ c} = x_{so}^{\ c} \cdot k_s \cdot k_{ro}(S_o) \cdot \frac{b_o}{\mu_o}$$

where $x_{so}^{\ c}$ is the number of moles of component c in the solid phase that are suspended in the oil phase, per mole of oil

$$x_{so}^{\ c} = \frac{f_s \cdot m^c}{S_o \cdot b_o}$$

Initial solids

Initial values for fluid components can be determined in three ways:

1 defined explicitly by the user

2 calculated by the simulator from equilibrium conditions

3 read from restart files

In restart runs, initial values of solid components are read from the restart file.

For solid components, explicit initialization is used in cases 1 and 2. This is because the solid components do not affect the flash and therefore do not affect the equilibration. The molar concentration of component c is given by

$$m^c = (1-\hat{S}_s)\cdot m_{fl}^c + \hat{S}_s \cdot b_s \cdot x_s^c \qquad \text{[EQ 62.6]}$$

where

$\hat{S}_s$ is the initial solid saturation, defined by keyword SSOLID

x_s^c is the initial solid molar fraction, defined by keyword SMF

and

m_{fl}^c is the concentration of the fluid components, set either explicitly or by equilibration.

For solid components the initial fluid concentration (which can be specified by ZI, ZMFVD, COMPVD, XMF, YMF, ZMF or NEI) should be zero. For fluid components the initial solid concentration should be zero.

If the initial solid saturation is not defined then a default value of zero is used and [EQ 62.6] becomes:

$$m^c = m_{fl}^c$$

Initial porosity

The terms $(1-\hat{S}_s)$ and $\hat{S}_s$ in [EQ 62.6] represent the fractions of the pore-volume initially containing fluid and solid. The initial pore volume, which is usually calculated from the porosity θ should therefore include both a fluid volume and a solid volume:

$$\theta = \frac{V_f + V_s}{V_f + V_s + V_r}$$

where V_r is the rock volume.

In some circumstances the measured porosity θ' may only take into account the initial fluid volume

$$\theta' = \frac{V_f}{V_f + V_s + V_r} = \theta \cdot (1-\hat{S}_s)$$

If these values are used to define the porosity (PORO) then the initial pore volumes should in theory be multiplied by a factor $1/(1-\hat{S}_s)$ using MULTPV. However if the initial solid saturations are small or zero this term can be neglected.

Initial energy

Thermal only

In thermal runs the initial energy density e is calculated from:

$$V_b \cdot e = V_p \cdot ((1-\hat{S}_s)\cdot e_{fl} + \hat{S}_s \cdot b_s \cdot e_s) + V_r \cdot e_r$$

where

V_b , is the bulk volume

$V_b = V_p + V_r$

e_{fl} is the fluid energy density

e_s is the solid energy density,

e_r is the rock energy density.

Chemical reactions

Solid components can be used in chemical reactions, see "Reactions involving solid components" on page 151. This provides a mechanism for modelling the appearance and disappearance of solid-only components.

When the reservoir contains solids, the reaction rates will depend on the solid saturation. This dependence can be modified by using the `REACSORD` keyword.

Solid production

If suspended solids are allowed, then solids will be produced by producer wells. Solid production data can be output to the summary file using the keywords in Table 62.6.

Solution of the Linear Equations

Chapter 63

Introduction

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

The linear equations are solved iteratively using Orthomin [Ref. 44] preconditioned by Nested Factorization. At each iteration, Orthomin minimizes the sum of the squares of the residuals by orthogonalities each new search direction to the existing search directions. To carry out these orthogonalizations, each search direction must be stored. Usually, only the last few search directions are significant and memory is saved if the number of stored search directions is restricted. This is done in ECLIPSE through the NSTACK keyword of the RUNSPEC data section.

The default NSTACK is 10 in ECLIPSE 100 and 40 in ECLIPSE 300. NSTACK should always be less than or equal to the maximum number of linear iterations (LITMAX), which may be altered using keyword TUNING (ECLIPSE 100) or CVCRIT (ECLIPSE 300) in the SCHEDULE section.

Each search direction consists of pressures, saturations and residuals for each cell in an ECLIPSE 100 black oil run, which is six double precision numbers per cell in a 3-phase black oil run. In ECLIPSE 300, each search direction consists of pressures for each cell and also molar densities for any implicit cells.

It should rarely be necessary to alter the default settings of NSTACK and LITMAX. However, for very difficult problems, they may have to be increased to ensure that the linear equations are solved correctly. Warning messages are printed by ECLIPSE when the linear equations are not fully converged. The total memory required for each run is printed out by ECLIPSE as soon as the required data has been read in and analyzed.

The progress of the linear solver may be monitored by setting the seventh argument of the DEBUG keyword to 1 in ECLIPSE 100 and by setting the ninth argument of the DEBUG3 keyword to 1 in ECLIPSE 300.

Material balance

At each Newton iteration we solve the linear equation

$$Ax = b$$

where A is the Jacobian matrix (dR/dX), and b is the non-linear residual of the prior Newton iteration, $b = R = (dM/dt) + F + Q$. If the elements of b are summed over all cells in the reservoir then the flow terms, F, will cancel and the sum corresponds to the rate of mass accumulation in the reservoir. ECLIPSE computes an initial approximate solution, y, by solving an approximate equation

$$By = b$$

The material balance error (that is the error in the mass accumulation rate) corresponding to y is obtained by summing the elements of the residual, r

$$r = b - Ay = (B - A)y \qquad \text{sum} = \sum_{i=1}^{\text{ncells}} r_i \qquad \text{[EQ 63.1]}$$

This sum is zero if B is chosen such that

$$\text{colsum}(B) = \text{colsum}(A)$$

where **colsum**(A) is the diagonal matrix formed by summing A in columns. The above colsum constraint is obeyed independently for each component by ECLIPSE. The **colsum** constraint is appropriate to obtain fast convergence for fully implicit black oil cases in ECLIPSE 100 because of the material balance constraint.

In adaptive implicit cases in ECLIPSE 300, it is more efficient to use a **rowsum** constraint where the preconditioner may be constructed such that

$$\text{rowsum}(B) = \text{rowsum}(A)$$

where **rowsum**(A) is the diagonal matrix formed by summing the pressure terms in A by rows. This technique is most effective for solving the pressure equation in an ECLIPSE 300 `AIM` run. This technique is described in [Ref. 46].

Nested factorization preconditioning

A typical matrix for a simple three-dimensional system is shown in Figure 63.1.

Figure 63.1 The structure of the Jacobian matrix for a simple 3*2*3 problem.

$$A = \begin{bmatrix}
\mathrm{D} & \mathrm{U1} & . & \mathrm{U2} & . & . & \mathrm{U3} & . & . & . & . & . & . & . & . & . & . & . \\
\mathrm{L1} & \mathrm{D} & \mathrm{U1} & . & \mathrm{U2} & . & . & \mathrm{U3} & . & . & . & . & . & . & . & . & . & . \\
. & \mathrm{L1} & \mathrm{D} & . & . & \mathrm{U2} & . & . & \mathrm{U3} & . & . & . & . & . & . & . & . & . \\
\mathrm{L2} & . & . & \mathrm{D} & \mathrm{U1} & . & . & . & . & \mathrm{U3} & . & . & . & . & . & . & . & . \\
. & \mathrm{L2} & . & \mathrm{L1} & \mathrm{D} & \mathrm{U1} & . & . & . & . & \mathrm{U3} & . & . & . & . & . & . & . \\
. & . & \mathrm{L2} & . & \mathrm{L1} & \mathrm{D} & . & . & . & . & . & \mathrm{U3} & . & . & . & . & . & . \\
\mathrm{L3} & . & . & . & . & . & \mathrm{D} & \mathrm{U1} & . & \mathrm{U2} & . & . & \mathrm{U3} & . & . & . & . & . \\
. & \mathrm{L3} & . & . & . & . & \mathrm{L1} & \mathrm{D} & \mathrm{U1} & . & \mathrm{U2} & . & . & \mathrm{U3} & . & . & . & . \\
. & . & \mathrm{L3} & . & . & . & . & \mathrm{L1} & \mathrm{D} & \mathrm{U1} & . & \mathrm{U2} & . & . & \mathrm{U3} & . & . & . \\
. & . & . & \mathrm{L3} & . & . & \mathrm{L2} & . & \mathrm{L1} & \mathrm{D} & \mathrm{U1} & . & . & . & . & \mathrm{U3} & . & . \\
. & . & . & . & \mathrm{L3} & . & . & \mathrm{L2} & . & \mathrm{L1} & \mathrm{D} & \mathrm{U1} & . & . & . & . & \mathrm{U3} & . \\
. & . & . & . & . & \mathrm{L3} & . & . & \mathrm{L2} & . & \mathrm{L1} & \mathrm{D} & . & . & . & . & . & \mathrm{U3} \\
. & . & . & . & . & . & \mathrm{L3} & . & . & . & . & . & \mathrm{D} & \mathrm{U1} & . & \mathrm{U2} & . & . \\
. & . & . & . & . & . & . & \mathrm{L3} & . & . & . & . & \mathrm{L1} & \mathrm{D} & \mathrm{U1} & . & \mathrm{U2} & . \\
. & . & . & . & . & . & . & . & \mathrm{L3} & . & . & . & . & \mathrm{L1} & \mathrm{D} & . & . & \mathrm{U2} \\
. & . & . & . & . & . & . & . & . & \mathrm{L3} & . & . & \mathrm{L2} & . & . & \mathrm{D} & \mathrm{U1} & . \\
. & . & . & . & . & . & . & . & . & . & \mathrm{L3} & . & . & \mathrm{L2} & . & \mathrm{L1} & \mathrm{D} & \mathrm{U1} \\
. & . & . & . & . & . & . & . & . & . & . & \mathrm{L3} & . & . & \mathrm{L2} & . & \mathrm{L1} & \mathrm{D}
\end{bmatrix} \qquad \text{[EQ 63.2]}$$

ECLIPSE chooses direction 1 automatically to correspond to the direction of highest transmissibility (usually the Z direction). Thus L1 and U1 usually correspond to derivatives of flows between cells in the Z direction. The inner solver direction may be overridden using the `SOLVDIRS` keyword. This may be useful, for example, if it is known in advance that the principal flow is in the X direction.

Three-dimensional finite difference systems give rise to sparse linear equations with banded matrices (Figure 63.1) such as

$$A = D + \mathrm{L1} + \mathrm{U1} + \mathrm{L2} + \mathrm{U2} + \mathrm{L3} + \mathrm{U3} \qquad \text{[EQ 63.3]}$$

where

D is a diagonal matrix

L1, L2 and L3 are lower bands

U1, U2 and U3 are upper bands

The elements of D, L1, L2, L3, U1, U2, U3 are 2×2 matrices for implicit 2-phase systems (for example oil/water, oil/gas, gas/water) and 3×3 matrices for implicit 3-phase systems. In the `IMPES` case these elements are scalars.

L2 and U2 connect cells in direction 2 etc. In Nested Factorization, an approximation, B, is constructed by the following nested sequence of factorizations

$$B = (P + \mathrm{L3})P^{-1}(P + \mathrm{U3}) = P + \mathrm{L3} + \mathrm{U3} + \mathrm{L3} \cdot P^{-1} \cdot \mathrm{U3}$$

$$P = (T + \mathrm{L2})T^{-1}(T + \mathrm{U2}) = T + \mathrm{L2} + \mathrm{U2} + \mathrm{L2} \cdot T^{-1} \cdot \mathrm{U2}$$

$$T = (G + \mathrm{L1})G^{-1}(G + \mathrm{U1}) = G + \mathrm{L1} + \mathrm{U1} + \mathrm{L1} \cdot G^{-1} \cdot \mathrm{U1} \qquad \text{[EQ 63.4]}$$

where G is a diagonal matrix and P^{-1} is the inverse of P etc. It follows that

$$\begin{aligned} B &= G + \mathrm{L1} + \mathrm{U1} + \mathrm{L2} + \mathrm{U2} + \mathrm{L3} + \mathrm{U3} + \mathrm{L1}G^{-1}\mathrm{U1} + \mathrm{L2}T^{-1}\mathrm{U2} + \mathrm{L3}P^{-1}\mathrm{U3} \\ &= A + G - D + \mathrm{L1}G^{-1}\mathrm{U1} + \mathrm{L2}T^{-1}\mathrm{U2} + \mathrm{L3}P^{-1}\mathrm{U3} \end{aligned} \qquad \text{[EQ 63.5]}$$

The diagonal matrix G is computed using

$$0 = G - D + \mathrm{L1}\,G^{-1}\mathrm{U1} + \mathrm{Colsum}(\mathrm{L2}\,T^{-1}\mathrm{U2} + \mathrm{L3}\,P^{-1}\mathrm{U3}) \qquad \text{[EQ 63.6]}$$

This choice of G ensures that **colsum**(B) = **colsum**(A) which, in turn, ensures that there are no material balance errors in the solution of the linear equations.

Alternatively, the preconditioner may be constructed such that **rowsum**(B) = **rowsum**(A), thus summing the pressure terms in A by rows to ensure accurate solution of the pressure equation in `IMPES` or `AIM` cases. This technique is described in [Ref. 46].

Off band elements, corresponding to non-neighbor connections, arising in the treatment of faults, local grid refinement, completing the circle in 3D radial studies etc., destroy the simple structure of the bands L1, U1..., but are easily incorporated in the Nested Factorization procedure by simply generalizing the definition of the bands.

Initialization for nested factorization preconditioner

Before iteration begins we must compute the diagonal matrix G. ECLIPSE computes and stores G^{-1} rather than G as this is more efficient in subsequent calculations. To conserve material, using the **colsum** variant on the preconditioner we have

$$G = D - \mathrm{L1}\,G^{-1}\mathrm{U1} - \mathrm{colsum}(\mathrm{L2}\,T^{-1}\mathrm{U2}) - \mathrm{colsum}(\mathrm{L3}\,P^{-1}\mathrm{U3})$$

The calculation proceeds one cell at a time; thus when G^{-1} is known for a cell, we can calculate the contribution $\mathrm{L1}\,G^{-1}\mathrm{U1}$ to the value of G in the next cell. When G^{-1} is known on a line, we can calculate the contribution of $\mathrm{colsum}(\mathrm{L2}\,T^{-1}\mathrm{U2})$ to G on the next line. When G is known on a plane, we can calculate the contribution of $\mathrm{colsum}(\mathrm{L3}\,P^{-1}\mathrm{U3})$ to G on the next plane. The cost of computing G (and G^{-1}) is reported under the mnemonic `GINV` in the ECLIPSE timing reports. Alternatively, the **rowsum** variant on the preconditioner leads to a similar expression for G, viz

$$G = D - \mathrm{L1}\,G^{-1}\mathrm{U1} - \mathrm{rowsum}(\mathrm{L2}\,T^{-1}\mathrm{U2}) - \mathrm{rowsum}(\mathrm{L3}\,P^{-1}\mathrm{U3})$$

Solution procedure for nested factorization

To compute a new search direction ECLIPSE solves the equation $By = r$ by the following hierarchical procedure. At the outermost level we solve

$$(P + \mathrm{L3})(I + P^{-1}\mathrm{U3})y = r \qquad \text{[EQ 63.7]}$$

using

$$y = P^{-1}(r - \mathrm{L3}y)$$

and

$$y = y - P^{-1}\mathrm{U3}y \qquad \text{[EQ 63.8]}$$

is solved one plane at a time, starting with the first plane and progressing forward until y is known on each plane. The equation is explicit because $\mathrm{L3}y$ involves the 'known' solution, y, on the previous plane. Similar considerations apply to the solution of [EQ 63.35] that is performed in reverse order starting with the last plane and sweeping backwards one plane at a time.

During the solution of and [EQ 63.35] we must compute vectors of the form $z = P^{-1}q$ on each plane. This involves solving equations like

$$(T + \mathrm{L2})(I + T^{-1}\mathrm{U2})z = q \qquad \text{[EQ 63.9]}$$

using

$$z = T^{-1}(q - \mathrm{L2}z)$$

and $z = z - T^{-1}\mathrm{U2}z$ [EQ 63.10]

[EQ 63.37] is solved in a forward sweep through the plane, one line at a time. Similarly equation [EQ 63.10] is solved by sweeping backwards one line at a time.

Finally, we note that during the solution of [EQ 63.37] and [EQ 63.10] we must compute vectors of the form $w = T^{-1}v$ on each plane. This involves solving the tri-diagonal equation

$$(G + \mathrm{L1})(I + G^{-1}\mathrm{U1})w = v \qquad \text{[EQ 63.11]}$$

using $w = G^{-1}(v - \mathrm{L1}w)$ [EQ 63.12]

and $w = w - G^{-1}\mathrm{U1}w$ [EQ 63.13]

[EQ 63.12] and [EQ 63.13] are solved by sweeping first forwards, then backwards through the cells in a line. This may be solved as a block tri-diagonal in the intermediate nesting if a method exists of inverting the line tri-diagonal equations. This may be done simply, using the Thomas algorithm, and so the entire system may be solved. Computing time required to calculate a new search direction is reported under the mnemonic SRCH in the ECLIPSE timing reports.

Orthomin

The Orthomin procedure used by ECLIPSE can be summarized in the steps below:

Compute the initial solution, $y = B^{-1}b$, $x \rightarrow y$ [EQ 63.14]

Compute the initial residual, $r = b - Ay$ [EQ 63.15]

- If the colsum constraint is used, the sum of the elements of r is now zero.
- If the rowsum constraint is used r is identically zero for the special case $b = Ap$ where p is the vector consisting of unity in each pressure location and zero elsewhere.

1 Test for convergence and escape if converged

Find a new search direction, $y = B^{-1}b$ [EQ 63.16]

2 Orthogonalize γ to the stack of previous search directions (z)

$$y = y - \frac{(y \cdot z)}{(z \cdot z)z} \quad \text{[EQ 63.17]}$$

3 Find the optimum step length, s, which minimizes the saturation normalized sum of the squares of the residuals $(r \cdot r)$:

$$s = \frac{r \cdot q}{q \cdot q} \quad \text{[EQ 63.18]}$$

$$q = A \cdot y \quad \text{[EQ 63.19]}$$

4 Update the solution $(x \rightarrow x + sy)$, the residual $r \rightarrow r - sq$ and the stack $z = (z, y)$. If not converged, return to step 3.

ECLIPSE 100

The progress of the linear solver may be monitored by setting the seventh argument of the `DEBUG` keyword to 1. The convergence criterion for the linear solver may be modified using the `TUNING` keyword.

ECLIPSE 300

The progress of the linear solver may be monitored by setting the ninth argument of the `DEBUG3` keyword to 1. The convergence criterion for the linear solver may be modified using the `CVCRIT` or `LSCRIT` keywords.

Convergence criteria

The convergence criterion in ECLIPSE 100 is based on the maximum residuals for each phase. These are normalized to represent effective saturation normalized residuals so that the convergence criteria are physically meaningful in terms of saturation changes.

In the ECLIPSE 300 formulation, because the equations are formulated in terms of pressure and molar densities, it is more appropriate to base the convergence criterion on a reduction in root mean square residual.

The convergence criteria for the linear equations in both ECLIPSE 100 and ECLIPSE 300 are described in more detail below.

ECLIPSE 100

Convergence of the linear solver can be monitored by setting the 7th parameter of keyword DEBUG greater than zero. If set to 1, a debug report is generated showing, at each iteration for each phase:

1 Saturation normalized root mean square residuals,

2 Residual sums (material balance errors) and

3 Absolute maximum saturation normalized residuals.

The Orthomin procedure is set to minimize item 1 above. Item 2 should be zero to within rounding error, while item 3 is used to test for convergence. The standard convergence criterion is that the maximum saturation normalized residual, $|r|_{max}$, be less than TRGLCV (= 0.0001). The value of TRGLCV may be reset using keyword TUNING, but this is seldom necessary and should not be done without good reason. Convergence is also assumed if $|r|_{max}$ is less than $\mathrm{MIN}(0.1|b|_{max}, 0.001 r_{top}, \mathrm{XXXLCV})$. The default setting of XXXLCV is 0.001 and this may also be reset in TUNING. $|b|_{max}$ is the largest saturation normalized input residual and r_{top} is the greatest saturation normalized residual to occur at any stage of the procedure. r_{top} may be much larger than $|b|_{max}$ because the colsum constraint makes the initial residuals quite large. Values of the step length, s, and the orthogonalizations coefficients (beta) are also printed in the debug output. The perfect step is s = 1.0, but this is seldom achieved. If the seventh DEBUG parameter is set to 2 then the sum of the absolute un-normalized residuals is printed corresponding to the input residual, b, and the final residual, $r = b - Ax$. This is the 'residual from solution check'. If the seventh DEBUG parameter is set to 3 then residual maps are printed for each phase at each iteration.

ECLIPSE 300

The convergence criterion on the linear solver is a reduction in RMS of the residual, so that convergence is achieved at the nth iteration if

$$\frac{\|R\|_2}{\|R^o\|_2} < TOL \qquad \text{[EQ 63.20]}$$

where

$$\|R\|_2 = \sqrt{\frac{\sum_{i=1}^{N} \sum_{c=1}^{N_{ci}} \left(\frac{R_{i,c}}{PV_i}\right)^2}{N}} \qquad \text{[EQ 63.21]}$$

with

R = linear residual

R^o = initial linear residual

$R_{i,c}$ = residual for equation c in grid cell i

PV_i = pore volume for cell i

N = total number of active cells

N_{ci} = number of equations for cell i

TOL = RMS convergence tolerance specified in `CVCRIT` (default 1.0E-5).

Treatment of wells

The strongly coupled fully-implicit treatment of wells in ECLIPSE 100 extends the linear equations to include the well variables, w, corresponding to the bottom hole flowing pressure, and well bore flowing fractions. Thus each well, like each cell, has three variables in the three phase case. Including the well terms, the linear equations now become

$$\begin{aligned} Ax + Cw &= r \\ Rx + Dw &= u \end{aligned} \qquad \text{[EQ 63.22]}$$

where

u is the well residuals,

and if there are N grid blocks and L wells:

A is a banded $\mathrm{N} \times \mathrm{N}$ matrix (Figure 63.1),

C is a sparse $\mathrm{N} \times \mathrm{L}$ matrix,

R is a sparse $\mathrm{L} \times \mathrm{N}$ matrix,

D is a sparse $\mathrm{L} \times \mathrm{L}$ matrix.

Usually D is lower triangular and easily inverted, allowing the well terms to be eliminated. Thus the equation for x becomes

$$(A - CD^{-1}R)x = r - CD^{-1}u \qquad \text{[EQ 63.23]}$$

which is easily solved by preconditioned Orthomin.

ECLIPSE 300

In ECLIPSE 300, there is a single well variable *w*, corresponding to well bottom-hole pressure. When an implicit treatment of the gravity head for each completion is used, the well constraint equations cannot be solved as in [EQ 63.22]. In the case of wells with multi-layer completions, extra coupling terms are introduced between all the cells on a given well. This is because the required well bottom hole pressure is a function of the solution values in each completion and each completion flow is a function of this well bottom-hole pressure. An additional contribution to the Jacobian is then defined:

$$J_w = \frac{dQ_w}{dX} \qquad \text{[EQ 63.24]}$$

which we include in the matrix A above. The column sum (or row sum) of J_w is included on the diagonal of the pre-conditioning matrix B. When using explicit head calculations the well equations do, however, reduce to the form of [EQ 63.23].

See [Ref. 47], and [Ref. 49] for further information.

Nested factorization for LGRs

The linear equations to be solved take the form

$$Ax = b \qquad \text{[EQ 63.25]}$$

where A is a banded matrix. The banded nature can be expressed as

$$A = D + L1 + U1 + L2 + U2 + L3 + U3 + L4 + U4 \qquad \text{[EQ 63.26]}$$

where

D is a diagonal matrix,

$L1$, $L2$, $L3$, $L4$ are lower bands, and

$U1$, $U2$, $U3$, $U4$ are upper bands.

The bands $L4$ and $U4$ represent couplings between sub-grids and are usually irregular.

In nested factorization, an approximate preconditioner M is constructed by the following nested sequence of factorizations:

$$M = (B + \mathrm{L4})B^{-1}(B + \mathrm{U4}) = B + \mathrm{L4} + \mathrm{U4} + \mathrm{L4} \cdot B^{-1} \cdot \mathrm{U4} \qquad \text{[EQ 63.27]}$$

$$B = (P + \mathrm{L3})P^{-1}(P + \mathrm{U3}) = P + \mathrm{L3} + \mathrm{U3} + \mathrm{L3} \cdot P^{-1} \cdot \mathrm{U3} \qquad \text{[EQ 63.28]}$$

$$P = (T + \mathrm{L2})T^{-1}(T + \mathrm{U2}) = T + \mathrm{L2} + \mathrm{U2} + \mathrm{L2} \cdot T^{-1} \cdot \mathrm{U2} \qquad \text{[EQ 63.29]}$$

$$T = (G + \mathrm{L1})G^{-1}(G + \mathrm{U1}) = G + \mathrm{L1} + \mathrm{U1} + \mathrm{L1} \cdot G^{-1} \cdot \mathrm{U1} \qquad \text{[EQ 63.30]}$$

It follows that

$$\begin{aligned} M = {} & G + \mathrm{L1} + \mathrm{U1} + \mathrm{L2} + \mathrm{U2} + \mathrm{L3} + \mathrm{U3} + \mathrm{L4} + \mathrm{U4} \\ & + \mathrm{L1}\,G^{-1}\mathrm{U1} + \mathrm{L2}\,T^{-1}\mathrm{U2} + \mathrm{L3}\,P^{-1}U3 + \quad L4M^{-1}U4 \end{aligned} \qquad \text{[EQ 63.31]}$$

that results in

$$M = A + G - D + \mathrm{L1}\,G^{-1}\mathrm{U1} + \mathrm{L2}\,T^{-1}\mathrm{U2} + \mathrm{L3}\,P^{-1}\mathrm{U3} + L4M^{-1}U4 \qquad \text{[EQ 63.32]}$$

The diagonal matrix G is defined as

$$G = D - L1\,G^{-1}U1 - rowsum(L2\,T^{-1}U2 + L3\,P^{-1}U3 + L4\,P^{-1}U4) \qquad \text{[EQ 63.33]}$$

This choice ensures that

$$rowsum(A) = rowsum(M) \qquad \text{[EQ 63.34]}$$

where rowsum (A) is the diagonal matrix formed by summing the pressure coefficients of A in rows. This ensures that M^{-1} is an exact inverse on the sub-space spanned by constant pressure within each sub-grid. This technique is described in [Ref. 50].

ECLIPSE 100

In-place LGRs (specified using keyword `LGRLOCK`) are solved using the above technique but with the rowsum constraint for black oil models.

ECLIPSE 300

For local grid refinements, the factorization can be enhanced by inner iterations on specific sub-grids. This technique is particularly effective when the inner iterations involve relatively small LGRs such as radial refinements. The sub-grid iterations are controlled by the `LSCRITS` keywords when using the WARP linear solver (see keyword `LSCRIT`).

The ECLIPSE 300 WARP linear solver

ECLIPSE 300

The WARP linear solver is the default solver for ECLIPSE 300. It is at present based on nested factorization only. Prior to 2002A the default solver was ECLSOL, which can be activated instead of WARP by using argument 66 of the `OPTIONS3` keyword. The WARP solver allows more flexibility for solutions of LGR problems than the ECLSOL solver.

Restrictions of the WARP solver

The 2003A WARP linear solver is incompatible with the following keywords:

- `MPFA`
- `NINEPOIN`
- `NONNC`

The 2002A WARP linear solver is also incompatible with

- `IMPSAT`

WARP sub-grids

The base entities that the WARP solver manipulates are sub-grids. These represent a portion of the total cells that are ordered to be a contiguous block of matrix.

- Every active cell in the grid is assigned to a sub-grid. In principle this association could be arbitrary; however, there are severe efficiency implications if there are very many sub-grids, or the sub-grids do not mainly contain cells which are only connected to other cells within the sub-grid.
- Sub-grid-to-sub-grid connections are implemented as NNC connections; the total generated is reported in the `PRT` file. Connections internal to sub-grids are, in general, stored as a mixture of band and NNC connections.
- Sub-grids may or may not be related to LGR grids. It is possible to have separate sub-grids for LGRs and the base grid. However, it is also possible to have sub-grids:
  - representing an amalgamation of portions of the base grid and portions of LGR grids (in this case the LGR planes are ordered first within any sub-grid to enhance the efficiency of this approach);
  - representing an amalgamation of portions of LGRs in a particular level of refinement.
- The form of the sub-grids is best observed by visualizing the output generated from the '`SUBG`' mnemonic in the `RPTRST` keyword.
- The preconditioner applied to each sub-grid is nested factorization (using ROWSUM on pressure only).
- The WARP linear solver performs nested factorization on the block matrix, where each block is a sub-grid. As with the sub-grid pre-conditioner, this block nesting preserves ROWSUM (on pressure only).
- When using the parallel option, sub-grids do not span parallel partitions; each sub-grid is completely contained with a processor.

WARP coloring

Each sub-grid is given a color. The WARP solver introduces extra nesting between colors. Sub-grids of the same color can be executed in parallel.

- NNCs between colors are preconditioned by extra nesting levels. If sub-grids only contain cells of a particular refinement level, then this is used by WARP to force a nesting between refinement levels. That is all colors on different levels of refinement are treated independently. Hence if two levels of refinement exist (because of the presence of LGRs), and each level has sub-grids containing two colors, then effectively four colors exits. The level of refinement is just used as an ordering tool for WARP to ensure that the highest level of refinement is the most heavily nested.
- Ideally there should be no NNC connections between sub-grids of the same color. Any NNC connections that do exist are not fully pre-conditioned, they are just added to the diagonal nesting matrix G to preserve rowsum.
- The coloring can be observed by visualizing the output from the `COLR` mnemonic in the `RPTRST` keyword.
- There is a limit of 4 levels of nesting between sub-grids, which correspond to a maximum of 16 colors.

WARP parallel example

The WARP solver can be used with the Parallel option. Each parallel partition is split into a number of sub-grids, which are connected through NNCs. Each sub-grid is colored (using the keyword `WCOL`), with an aim of having no NNCs connecting sub-grids of the same color. The WARP linear solver preforms nested factorization on a sub-grid level. Sub-grids of the same color can be computed in parallel. The pre-conditioner performed on a sub-grid is nested factorization. Note that:

- The generation of NNCs between sub-grids will cause a memory overhead.
- Increasing the number of colors will increase the expense of each linear iteration, as extra levels of nesting are required to solve the block sub-grid matrix.

The creation of sub-grids and coloring is controlled by the `PARALLEL` and `WCOL` keywords. The `PARALLEL` keyword controls the division of the original problem into partitions. Figure 63.2 shows a two-color scheme resulting from a 4-processor run. Note that all the LGRs have inherited the color from the base grid. The LGR coloring scheme is controlled by the `WCOL` keyword.

Figure 63.3 shows a possible set of 8 sub-grids that may arise from these colors. In this case portions of the LGR grids have been amalgamated with the base grid, and the WARP solver has no knowledge of refinement levels. This type of amalgamation is generated if the 3rd item of the `WCOL` keyword is set to 2. By amalgamation of LGR and base sub-grids, the load balancing of the parallel run is more easily facilitated. For a parallel run all colors must have equal work on all processors for a perfect load balance. The more colors that are introduced the more difficult this becomes. This amalgamation strategy ensures no extra levels of nesting are introduced by adding refinement levels. However, for cases where the nested factorization is required between refinement levels this strategy may prove to be far from ideal.

Figure 63.4 shows a different sub-grid configuration for the same coloring as in Figure 63.2. Here the LGRs are not amalgamated with the base grid; instead, all LGRs of the same refinement level and the same color are amalgamated together. Hence because the base grid has 8 sub-grids, and there is one level of LGR refinement spread fairly evenly over the grid, 16 sub-grids have been generated. Here WARP has knowledge of the refinement and will utilize this by adding nesting between refinement levels. This sub-grid configuration was generated by setting the 3rd item of `WCOL` to 1.

Figure 63.2 Two color (red-blue) split suitable for a 4-processors parallel run

Figure 63.3 The sub-grids from Figure 63.2 with amalgamation of LGRs and base grid

Figure 63.4 The sub-grids from Figure 63.2 with no amalgamation of LGRs and base grid.

WARP solution procedure for sub-grids

Firstly consider the two color case, where S_α and S_β represents all the sub-grids in colors α and β respectively, $L_{\alpha\beta}$ and $U_{\alpha\beta}$ are the lower and upper color-to-color connective terms (these are implemented using NNC connections). Let us assume that nested factorization can be performed on each sub-grid, and hence each color, as outlined in the previous sections. The matrix system A can be decomposed into a 2x2 block color matrix thus

$$A = \begin{bmatrix} 0 & 0 \\ L_{\alpha\beta} & 0 \end{bmatrix} + \begin{bmatrix} S_\alpha & 0 \\ 0 & S_\beta \end{bmatrix} + \begin{bmatrix} 0 & U_{\alpha\beta} \\ 0 & 0 \end{bmatrix} \qquad \text{[EQ 63.35]}$$

The procedure is to perform nested factorization on this block system, i.e. the pre-conditioner is

$$\tilde{M} = \begin{bmatrix} Q_\alpha & 0 \\ L_{\alpha\beta} & Q_\beta \end{bmatrix} \begin{bmatrix} Q_\alpha & 0 \\ 0 & Q_\beta \end{bmatrix}^{-1} \begin{bmatrix} Q_\alpha & U_{\alpha\beta} \\ 0 & Q_\beta \end{bmatrix} \qquad \text{[EQ 63.36]}$$

thus

$$\tilde{M} = \begin{bmatrix} Q_\alpha & U_{\alpha\beta} \\ L_{\alpha\beta} & (L_{\alpha\beta} Q_\beta^{-1} + Q_\beta) \end{bmatrix} \qquad \text{[EQ 63.37]}$$

The matrix Q is chosen to preserve rowsum (of pressure only) so

$$Q_\alpha = G_\alpha \qquad \text{[EQ 63.38]}$$

and

$$Q_\beta = G_\beta - rowsum(L_{\alpha\beta} Q_\beta{}^{-1} U_{\alpha\beta}) \qquad \text{[EQ 63.39]}$$

where G_α and G_β are the diagonal nested factorization matrices constructed from all the sub-grids associated with colors α . and β respectively. During this procedure Q^{-1} needs to be evaluated. If individual tolerances are imposed on sub-grids (through the `LSCRITS` keyword), then this inversion is approximated by multiple iterations of GMRES, rather than just one nested factorization inversion.

For multiple colors the above procedure is applied recursively. For example if four colors $\alpha, \beta, \gamma, \delta$ are present, the matrix A could be written

$$A = \begin{bmatrix} \begin{bmatrix} S_\alpha & U_{\alpha\beta} \\ L_{\alpha\beta} & S_\beta \end{bmatrix} & U_{\{\alpha\beta,\gamma\delta\}} \\ L_{\{\alpha\beta,\gamma\delta\}} & \begin{bmatrix} S_\gamma & U_{\gamma\delta} \\ L_{\gamma\delta} & S_\delta \end{bmatrix} \end{bmatrix} \qquad \text{[EQ 63.40]}$$

and then nested factorization can be performed on the 2x2 system, where the blocks are groups of colors. Up to four levels of nesting are currently available, which corresponds to a maximum of 16 colors.

Generalized minimum residual method (GMRES)

The preconditioned GMRES method is mathematically equivalent to the Orthomin method. However, it has better numerical properties and involves less computational work, so it is generally preferred over Orthomin and is used for the WARP solver. The GMRES method for solving $Ax = b$ can be summarized as follows:

Initialization

Set initial residual $r_0 = b$, $\beta = \|r_0\|_2$, $\upsilon_1 = r_0/\beta$ [EQ 63.41]

Iteration

For k = 1,2,... until convergence, do

Search direction $u_k = M^{-1}\upsilon_k$ [EQ 63.42]

Matrix multiply: $w = Au_k$ [EQ 63.43]

Orthogonalize: $h_{i,k} = (w \cdot \upsilon_i)$ [EQ 63.44]

$$w = w - \sum_{i=1}^{k} h_{i,k}\upsilon_i$$

Normalize: $h_{k+1,k} = \|w\|_2$ [EQ 63.45]

$$\upsilon_{k+1} = \frac{w}{h_{k+1,k}}$$

Solution: $$x_k = \sum_{i=1}^{k} \alpha_i u_i$$ [EQ 63.46]

where the α_i are chosen to minimize the residual $\|b - Ax_k\|_2$

For complete implementation details see [Ref. 51].

The Solvent Model

Chapter 64

Introduction

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL

This chapter describes a 4-component extension of the ECLIPSE Black-oil model. The aim of the extension to enable modeling of reservoir recovery mechanisms in which injected fluids are miscible with the hydro-carbons in the reservoir.

An injected fluid sets up a miscible displacement if there is no phase boundary or interface between the injected fluid and the reservoir oil. A miscible displacement has the advantage over immiscible displacements such as water flooding, of enabling very high recoveries. An area swept by a miscible fluid typically leaves a very small residual oil saturation.

The ECLIPSE solvent extension allows you to model gas injection projects without going to the complexity and expense of using a compositional model. The solvent extension implements the Todd and Longstaff empirical model for miscible floods and is flexible enough to model a wide range of gas injection schemes.

Applications

There is a wide variety of secondary and tertiary recovery schemes in which the aim is to enhance the reservoir sweep by using a miscible injection fluid.

Typically the miscible injection fluid is expensive compared with traditional flooding fluids, such as dry gas and water. Miscible fluids are often referred to as ‘solvents’; examples of solvent schemes are listed below:

- High pressure dry gas processes, in which mass transfer effects at the gas-oil contact generate miscible flow conditions between the gas and the oil.
- A solvent such as LPG or propane may be injected as a ‘slug’ to be followed by an extended period of lean gas injection. The slug fluid is miscible with both the gas and oil, and the success of the method depends upon maintenance of both the reservoir pressure and the solvent slug itself.
- Certain non-hydrocarbon gases such as carbon dioxide produce miscible displacement of oil at pressures above a threshold value.

- All-liquid miscible displacements by fluids such as alcohol, normally injected as a slug between the in-place oil and the injected chase water.

In many of the schemes, the cost of the solvent is such that the recovery scheme needs to be 'engineered' to allow enough solvent to be injected to increase the total oil recovery while keeping the solvent costs low enough to make the project economic.

In applications where the reservoir pressure falls below the bubble point, and in schemes where a solvent slug is chased by a lean gas flood, there is a requirement to model 4 components within the reservoir (water, oil, solvent and lean gas). The solvent model in ECLIPSE provides a simple method of modeling these recovery mechanisms.

The Todd and Longstaff mixing parameter model

In miscible gas injection processes, total displacement of the oil in the swept regions does not guarantee a high overall recovery efficiency. This is because the high adverse mobility ratio inherent in gas injection means that the flow is unstable and leads to the growth of viscous fingers that leave regions of by-passed oil in the reservoir at gas breakthrough. Gravitational fingering may also be present as a result of the large density differences between the injected gas and the in-place oil. The flow is unstable to finger formation at mobility ratios greater than unity, or in places where a dense fluid overlies a less dense fluid, and the fingers grow rapidly from small fluctuations in reservoir homogeneity. Reservoir simulation models must account for the growth of fingers whenever they might arise in the reservoir.

A further feature of miscible fluid displacement to be included in a simulation model is the mixing of the miscible components. Component mixing occurs on a microscopic scale and results from molecular diffusion and fluid velocity variations within the porous medium on the scale of the pore size. The mixing process is represented by a diffusion term in the component equations. Correct treatment of this mixing effect within the simulation model is important since the mixing term exerts a considerable damping effect on the growth of viscous and gravity fingers.

The simulation of incompressible miscible displacement was initially based on a direct solution of the convection-diffusion equations for the local concentration of each of the miscible components. However, such models are unreliable unless the fine scale structure of the fingers can be resolved otherwise mixing cell effects will dampen the finger growth rates and produce optimistic oil recovery forecasts. The numerical diffusion inherent in finite difference models also masks the true mixing of the miscible components unless the grid is sufficiently fine. The number of grid cells required for accurate field scale study of miscible displacement is unacceptably high and the cost of computing time prohibitive.

The ECLIPSE implementation

The model we shall consider is optionally a 4-component system consisting of water, oil, reservoir gas and an injected solvent, a 3-component system of water, oil and solvent gas or an oil/solvent system.

In regions of the reservoir containing only solvent and reservoir oil (possibly containing dissolved gas) the solvent and reservoir oil components are assumed to be miscible in all proportions and consequently only one hydrocarbon phase exists in the reservoir. The relative permeability requirements of the model are those for a two phase system (water/hydrocarbon).

In regions of the reservoir containing only oil and reservoir gas, the gas and oil components will be immiscible and will behave in a traditional black oil manner. In regions containing both dry gas and solvent, an intermediate behavior is assumed to occur. The modeling of the immiscible/miscible transition is described in subsequent sections.

The Todd-Longstaff model is an empirical treatment of the effects of physical dispersion between the miscible components in the hydrocarbon phase. The model introduces an empirical parameter, ω, whose value lies between 0 and 1, to represent the size of the dispersed zone in each grid cell. The value of ω thus controls the degree of fluid mixing within each grid cell. A value of $\omega = 1$ models the case when the size of the dispersed zone is much greater than a typical grid cell size and the hydrocarbon components can be considered to be fully mixed in each cell. In this case the miscible components have the same value for the viscosity and density,

as given by the appropriate mixing rule formulae. A value of $\omega = 0$ models the effect of a negligibly thin dispersed zone between the gas and oil components, and the miscible components should then have the viscosity and density values of the pure components. In practical applications an intermediate value of ω would be needed to model incomplete mixing of the miscible components.

By default the viscosity and density values will be calculated using the same mixing parameter, ω. However, if the mixing parameter is being used as a history matching parameter, it is sometimes useful to specify two separate mixing parameters, one for the viscosity and one for the density.

The relative permeability model

Immiscible gas phase relative permeabilities

In regions of the reservoir where the saturation of solvent is small, for example in the region swept by lean chase gas, the displacement is immiscible.

In the usual black-oil model the relative permeabilities for the 3 phases water, oil and gas are specified as follows:

$K_{rw} = K_{rw}(S_w)$ as a function of water saturation

$K_{rg} = K_{rg}(S_g)$ as a function of gas saturation

$K_{ro} = K_{ro}(S_w, S_g)$ as a function of both water and gas saturations.

When two gas components are present, the assumption is made that the total relative permeability of the gas phase is a function of the total gas saturation,

$$K_{rg} = K_{rg}(S_{gas} + S_{solvent}) \qquad \text{[EQ 64.1]}$$

Then the relative permeability of either gas component is taken as a function of the local solvent fraction within the gas phase,

$$\text{Fraction of solvent} = F_{sol} = \frac{S_{solvent}}{S_{gas} + S_{solvent}} \qquad \text{[EQ 64.2]}$$

$$\text{Fraction of reservoir gas} = F_{gas} = \frac{S_{gas}}{S_{gas} + S_{solvent}} \qquad \text{[EQ 64.3]}$$

$$\begin{aligned} K_{r\,\mathrm{sol}} &= K_{rg} \cdot K_{rfs}(F_{\mathrm{sol}}) \\ K_{r\,\mathrm{gas}} &= K_{rg} \cdot K_{rfg}(F_{\mathrm{gas}}) \end{aligned} \qquad \text{[EQ 64.4]}$$

Typically the functions $K_{rfs} = K_{r\,\mathrm{sol}}/K_{rg}$ and $K_{rfg} = K_{r\,\mathrm{gas}}/K_{rg}$ (Equation [EQ 64.4]) will be 'straight line' functions such that

$K_{rfs}(0.0) = 0.0 \quad K_{rfg}(0.0) = 0.0$

$K_{rfs}(1.0) = 1.0 \quad K_{rfg}(1.0) = 1.0$

However, ECLIPSE allows you to specify these gas-solvent relative permeabilities using the `SSFN` keyword.

Gas-oil relative permeabilities in miscible regions

In regions where solvent is displacing oil and the reservoir gas saturation is small, the hydrocarbon displacement is miscible. However, the 2-phase character of the water/hydrocarbon displacement needs to be taken into account. The relative permeabilities are given by:

$$K_{r\,\mathrm{oil}} = \frac{S_{\mathrm{oil}}}{S_n} K_{rn}(S_n) \qquad \text{[EQ 64.5]}$$

$$K_{rsg} = \frac{S_{\text{solvent}} + S_{\text{gas}}}{S_n} K_{rn}(S_n) \qquad \text{[EQ 64.6]}$$

where

K_{rsg} is the total relative permeability of the gas and solvent

$S_n = S_{\text{gas}} + S_{\text{solvent}} + S_{\text{oil}}$

$K_{rn}(S_n)$ is the relative permeability of hydrocarbon to water.

Then:

$$K_{r\ \text{sol}} = K_{rsg} \cdot K_{rfs}(F_{\text{sol}})$$

$$K_{r\ \text{gas}} = K_{rsg} \cdot K_{rfg}(F_{\text{gas}}) \qquad \text{[EQ 64.7]}$$

using the gas-solvent relative permeability curves (SSFN).

It is possible to modify the straight line miscible relative permeabilities, using the MSFN keyword to supply multipliers M_{Kro} and M_{Krsg}. The oil and total gas relative permeabilities will then be:

$$K_{r\ \text{oil}} = M_{Kro} \frac{S_{\text{oil}}}{S_n} K_{rn}(S_n) \qquad \text{[EQ 64.8]}$$

$$K_{rsg} = M_{Krsg} \frac{S_{\text{solvent}} + S_{\text{gas}}}{S_n} K_{rn}(S_n) \qquad \text{[EQ 64.9]}$$

Transition between miscible and immiscible regimes

The descriptions above are the treatments of the extremes when the displacement behavior is either fully miscible or fully immiscible. In practice there will be regions of transition where the displacement is changing from miscible to immiscible in character. Typically this occurs when a lean chase gas breaks through the solvent slug.

Todd and Longstaff recommend that the displacement will be miscible until the solvent saturation becomes very small (of the order of 0.01). However, this transition is likely vary from case to case.

The transition is handled in ECLIPSE by a user specified ‘Miscibility function’. This is a function of the Solvent fraction in the gas phase. The tabulated function is a parameter between 0 and 1, where 0 implies immiscible displacement and 1 implies miscible. Note that in an implicit simulator like ECLIPSE, the transition needs to occur over a range of solvent fraction rather than at a ‘cut-off’ point. The Miscibility function is supplied using the MISC keyword.

The transition algorithm has two steps:

1 Scale the relative permeability end points by the miscibility function. For example, the residual oil saturation is

$$S_{or} = S_{orm} M + S_{ori}(1 - M) \qquad \text{[EQ 64.10]}$$

where

S_{orm} is the miscible residual oil saturation (this is taken from SORWMIS, or defaulted to zero if SORWMIS is not present)

S_{ori} is the immiscible residual oil saturation (this is taken from S_{ogcr} in the SOF3 table, or SOGCR if entered)

M is the miscibility function

Similarly, for the solvent gas

$$S_{sgr} = S_{sgrm} M + S_{sgri}(1 - M) \quad \text{[EQ 64.11]}$$

where

S_{sgrm} is the miscible residual solvent+gas saturation (taken from SGCWMIS or defaulted to zero)

S_{sgri} is the immiscible residual solvent+gas saturation (taken from S_{gcr} in the SGFN table, or SGCR if entered)

2 Calculate the miscible and immiscible relative permeabilities, scaling for the new end points. Then the relative permeability is again an interpolation between the two using the miscibility function:

$$K_r = K_{rm} M + K_{ri}(1 - M) \quad \text{[EQ 64.12]}$$

where

K_{rm} the scaled miscible K_r

K_{ri} is the scaled immiscible K_r

3-phase cases

In 3-phase cases the solvent-oil displacement is always miscible, and hence the transition calculation is not used. The relative permeabilities are taken as:

$$K_{r\,\mathrm{sol}} = \frac{S_{\mathrm{solvent}}}{S_n} K_{rn}(S_n) \quad \text{[EQ 64.13]}$$

$$K_{r\,\mathrm{oil}} = \frac{S_{\mathrm{oil}}}{S_n} K_{rn}(S_n) \quad \text{[EQ 64.14]}$$

where

$$S_n = S_{\mathrm{solvent}} + S_{\mathrm{oil}}$$

Effect of water saturation

A feature of miscible gas injection processes that may also be modeled is the screening effect of high water saturation on the contact between the miscible gas and the in-place oil in each grid cell. The effective residual oil saturation to a miscible gas drive is found to increase with increasing water saturation, and correct modeling of the effect is important since it may reduce the efficiency of the miscible displacement. The process is modeled by introducing an effective residual oil saturation, S_{or}, which depends on the water saturation ($S_{or} = S_{or}(S_w)$).

A mobile oil saturation is then calculated by

$$S_o^* = \mathrm{MAX}(S_o - S_{or}, 0.0) \quad \text{[EQ 64.15]}$$

For completeness, a corresponding mobile gas saturation has been defined in which a critical gas saturation ($S_{gc} = S_{gc}(S_w)$) is specified for the miscible flood.

$$S_g^* = \mathrm{MAX}(S_g - S_{gc}, 0.0) \quad \text{[EQ 64.16]}$$

The mobile oil and gas saturations S_o^*, S_g^* are then used to determine the miscible component relative permeabilities and the effective viscosities and densities in each grid cell.

When End-point Scaling is used in conjunction with `SORWMIS`, then S_{orm} is replaced by the scaled value of the critical hydrocarbon to water saturation taken from the `SOF2` table (or keyword `SOGCR` if entered). When endpoint scaling is used in conjunction with `SGCWMIS`, then S_{sgrm} is replaced by the scaled value of `SGCR`.

The PVT model

4-phase viscosity model

The following form is suggested by Todd and Longstaff for the effective oil and solvent viscosities to be used in an immiscible simulator.

$$\mu_{o\ \mathrm{eff}} = \mu_o^{1-\omega} \cdot \mu_{mos}^{\omega}$$
$$\mu_{s\ \mathrm{eff}} = \mu_s^{1-\omega} \cdot \mu_m^{\omega} \qquad \text{[EQ 64.17]}$$
$$\mu_{g\ \mathrm{eff}} = \mu_g^{1-\omega} \cdot \mu_{msg}^{\omega}$$

where

μ_o , μ_s , μ_g are the unmixed viscosities of oil, solvent and gas.

μ_{mos} is the fully mixed viscosity of oil + solvent

μ_{msg} is the fully mixed viscosity of solvent + gas

μ_m is the fully mixed viscosity of oil + solvent + gas

ω is the Todd-Longstaff parameter.

The mixture viscosities μ_{mos} , μ_{msg} and μ_{um} are defined using the 1/4th-power fluid mixing rule, as follows:

$$\mu_{mos} = \frac{\mu_o \mu_s}{\left(\frac{S_o}{S_{os}}\mu_s^{1/4} + \frac{S_s}{S_{os}}\mu_o^{1/4}\right)^4} \qquad \text{[EQ 64.18]}$$

$$\mu_m = \frac{\mu_o \mu_s \mu_g}{\left(\frac{S_o}{S_n}\mu_s^{1/4}\mu_g^{1/4} + \frac{S_s}{S_n}\mu_o^{1/4}\mu_g^{1/4} + \frac{S_g}{S_n}\mu_o^{1/4}\mu_s^{1/4}\right)^4}$$

[EQ 64.19]

$$\mu_{msg} = \frac{\mu_s \mu_g}{\left(\frac{S_s}{S_{sg}}\mu_g^{1/4} + \frac{S_g}{S_{sg}}\mu_s^{1/4}\right)^4} \qquad \text{[EQ 64.20]}$$

where $S_n = S_{\mathrm{oil}} + S_{\mathrm{solvent}} + S_{\mathrm{gas}}$

The case $\omega = 1$ models a large dispersed oil-solvent zone. The Todd-Longstaff model treats this case as a local unit mobility ratio displacement.

If $\omega = 0$ then $\mu_{o\ \mathrm{eff}} = \mu_o$, $\mu_{s\ \mathrm{eff}} = \mu_s$ and each component has an effective viscosity equal to its ‘pure’ value. Such a case corresponds to a local high adverse mobility ratio displacement and models a negligibly thin oil-solvent dispersed zone.

The mixing parameter approach allows the case of a partial mixing zone to be modeled by choosing an intermediate value of ω .

The full 4-Component mixing calculation can be thought of as two separate miscible displacements: Gas/Solvent and Solvent/Oil.

4-component density model

The treatment of effective oil and solvent densities in the ECLIPSE model is based on the fourth power law for the effective viscosities. By default the density calculation can be based on the same mixing parameter as the viscosity, however a separate mixing parameter may be specified for the effective density calculation if required.

The densities are computed by the following procedure. First, the partially mixed or effective viscosities are calculated using equations [EQ 64.17] to [EQ 64.20] above. The value of each effective component viscosity is then used in turn in equations [EQ 64.18] to [EQ 64.20] to yield an effective saturation fraction to be used in oil and solvent density calculations. The effective saturation fractions used for density calculations are:

$$\left(\frac{S_o}{S_n}\right)_{oe} = \frac{(\mu_{o\,\mathrm{eff}}{}^{1/4}\mu_o{}^{1/4}) - (\mu_s{}^{1/4}\mu_o{}^{1/4})}{\mu_{o\,\mathrm{eff}}{}^{1/4}(\mu_o{}^{1/4} - \mu_s{}^{1/4})} \qquad \text{[EQ 64.21]}$$

$$\left(\frac{S_o}{S_n}\right)_{se} = \frac{(s_{of}\,\mu_o{}^{1/4}\mu_g{}^{1/4}) - \left(\dfrac{\mu_o{}^{1/4}\mu_s{}^{1/4}\mu_g{}^{1/4}}{\mu_{s\,\mathrm{eff}}{}^{1/4}}\right) + (s_{gf}\,\mu_o{}^{1/4}\mu_s{}^{1/4})}{(s_{of}\,\mu_o{}^{1/4}\mu_g{}^{1/4}) - (\mu_o{}^{1/4}\mu_g{}^{1/4}) + (s_{gf}\,\mu_o{}^{1/4}\mu_s{}^{1/4})} \qquad \text{[EQ 64.22]}$$

$$\left(\frac{S_o}{S_n}\right)_{ge} = \frac{(-\mu_{g\,\mathrm{eff}}{}^{1/4}\mu_s{}^{1/4}) + (\mu_g{}^{1/4}\mu_s{}^{1/4})}{\mu_{g\,\mathrm{eff}}{}^{1/4}(\mu_s{}^{1/4} - \mu_g{}^{1/4})} \qquad \text{[EQ 64.23]}$$

where

$$S_{of} = S_{\mathrm{oil}}/(S_{\mathrm{oil}} + S_{\mathrm{gas}})$$
$$S_{gf} = S_{\mathrm{gas}}/(S_{\mathrm{oil}} + S_{\mathrm{gas}})$$

The effective oil and solvent densities ($\rho_{o\,\mathrm{eff}}$, $\rho_{s\,\mathrm{eff}}$, $\rho_{g\,\mathrm{eff}}$) are now computed from the effective saturation fractions in equations [EQ 64.21] to [EQ 64.23], and the pure component densities (ρ_o, ρ_s, ρ_g) using the following formulae

$$\rho_{o\,\mathrm{eff}} = \rho_o\left(\frac{S_o}{S_n}\right)_{oe} + \rho_s\left[1 - \left(\frac{S_o}{S_n}\right)_{oe}\right] \qquad \text{[EQ 64.24]}$$

$$\rho_{s\,\mathrm{eff}} = \rho_s\left(\frac{S_s}{S_n}\right)_{se} + \rho_g S_{gf}\left[1 - \left(\frac{S_s}{S_n}\right)_{se}\right] + \rho_o S_{of}\left[1 - \left(\frac{S_s}{S_n}\right)_{se}\right] \qquad \text{[EQ 64.25]}$$

$$\rho_{g\,\mathrm{eff}} = \rho_g\left(\frac{S_g}{S_n}\right)_{ge} + \rho_s\left[1 - \left(\frac{S_g}{S_n}\right)_{ge}\right] \qquad \text{[EQ 64.26]}$$

3-component model

For the 3-component case the above viscosity model reduces to:

$$\mu_{o\,\mathrm{eff}} = \mu_o{}^{1-\omega}\mu_m{}^{\omega} \qquad \text{[EQ 64.27]}$$

$$\mu_{s\,\mathrm{eff}} = \mu_s{}^{1-\omega}\mu_m{}^{\omega} \qquad \text{[EQ 64.28]}$$

$$\mu_m = \frac{\mu_o \mu_s}{\left(\frac{S_s}{S_n}\mu_o^{1/4} + \frac{S_o}{S_n}\mu_s^{1/4}\right)^4} \quad \text{[EQ 64.29]}$$

$$S_n = S_o + S_s \quad \text{[EQ 64.30]}$$

The density model reduces to:

$$\left(\frac{S_o}{S_n}\right)_{oe} = \frac{(\mu_{o\,\mathrm{eff}}^{1/4}\mu_o^{1/4}) - (\mu_s^{1/4}\mu_o^{1/4})}{\mu_{o\,\mathrm{eff}}^{1/4}(\mu_o^{1/4} - \mu_s^{1/4})} \quad \text{[EQ 64.31]}$$

$$\left(\frac{S_o}{S_n}\right)_{se} = \frac{(\mu_{s\,\mathrm{eff}}^{1/4}\mu_o^{1/4}) - (\mu_s^{1/4}\mu_o^{1/4})}{\mu_{s\,\mathrm{eff}}^{1/4}(\mu_o^{1/4} - \mu_s^{1/4})} \quad \text{[EQ 64.32]}$$

$$\rho_{o\,\mathrm{eff}} = \rho_o\left(\frac{S_o}{S_n}\right)_{oe} + \rho_s\left[1 - \left(\frac{S_o}{S_n}\right)_{oe}\right] \quad \text{[EQ 64.33]}$$

$$\rho_{s\,\mathrm{eff}} = \rho_o\left(\frac{S_o}{S_n}\right)_{se} + \rho_s\left[1 - \left(\frac{S_o}{S_n}\right)_{se}\right] \quad \text{[EQ 64.34]}$$

In the 3-component case ECLIPSE allows you to specify a different surface density for the solvent gas to that for the solution gas. In equations [EQ 64.33] and [EQ 64.34], the pure solvent density at reservoir conditions, ρ_s, is computed from the solvent surface density. The oil density at reservoir conditions, ρ_o, is calculated from the stock tank oil and solution gas surface densities.

Choice of the mixing parameter

The mixing-parameter model would be of limited use unless the mixing parameter could itself be modeled over a wide range of operating conditions. A value of $\omega = 1$ results in a piston-like displacement of oil by the injected solvent. If $\omega = 0$ the displacement is similar to an immiscible displacement (except for the treatment of relative permeability). An intermediate value of ω results in a continuous solvent saturation increase behind the solvent front. Todd and Longstaff accounted for the effects of viscous fingering in 2D studies by setting $\omega = 2/3$ independently of mobility ratio. For field scale simulations they suggested setting $\omega = 1/3$. However, in general history matching applications, the mixing parameter may be regarded as a useful history matching variable to account for any reservoir process inadequately modeled.

The pressure dependence of miscibility

In many miscible displacements, the solvent is only miscible with the reservoir oil at high pressure. Typically the solvent/oil capillary pressure will reduce with increasing pressure, and only when it has reduced to zero can the two fluids be considered to be miscible.

The pressure dependence can be modeled in ECLIPSE by using the `PMISC` keyword to supply a pressure dependent miscibility function M_p. This is used to interpolate between the immiscible and miscible values of the PVT, relative permeability and capillary pressure data.

PVT properties are interpolated as follows (taking the gas case as an example):

$$\frac{1}{B_g} = \frac{1}{B_{gm}} M_p + \frac{1}{B_{gi}}(1 - M_p) \qquad \text{[EQ 64.35]}$$

$$\frac{1}{(B\mu)_g} = \frac{1}{(B\mu)_{gm}} M_p + \frac{1}{(B\mu)_{gi}}(1 - M_p) \qquad \text{[EQ 64.36]}$$

where

B_g is the gas formation volume factor

B_{gi} is the immiscible gas formation volume factor

B_{gm} is the miscible gas formation volume factor

$(B\mu)_g$ is the gas FVF × viscosity

$(B\mu)_{gi}$ is the gas FVF × viscosity (immiscible)

$(B\mu)_{gm}$ is the gas FVF × viscosity (miscible)

M_p is the pressure dependent miscibility function.

The relative permeability interpolation is treated in the same manner as the transition from immiscible displacement to miscibility as a function of the solvent concentration. (See "The relative permeability model" on page 959.)

The capillary pressure is treated as follows:

$$P_{cog} = P_{cog}(\text{misc}) M_p + P_{cog}(\text{immisc})(1 - M_p) \qquad \text{[EQ 64.37]}$$

$$\begin{aligned} P_{cog}(\text{misc}) &= P_{cog}(S_g) \\ P_{cog}(\text{immisc}) &= P_{cog}(S_g + S_s) \end{aligned} \qquad \text{[EQ 64.38]}$$

where

S_g is the lean gas saturation

S_s is the solvent saturation.

Using the solvent model

Initiating the model

Two keywords in RUNSPEC control the Solvent and Miscible models:

Table 64.1 RUNSPEC keywords for solvent and Miscible models

Keyword	Description
SOLVENT	Activates the Separate Solvent component
MISCIBLE	Initiates the mixing calculation.

If SOLVENT is specified, then the 4-component model is used. SOLVENT should only be used with all the other 3 phases (Water, Oil, Gas) present.

It is possible to run the Solvent model in immiscible mode. In this case the mixing is not performed, and the 4 phases take their pure PVT properties. The relative permeabilities are calculated as for a traditional black oil case, with the solvent and gas taking the 'gas' relative permeability, modified as a function of the local solvent fraction.

If MISCIBLE is specified, then the mixing calculations are used, employing the 3-component model if SOLVENT is **not** specified and the 4-component model when SOLVENT is specified.

PVT data

In the 4-component model, PVT data needs to be supplied for the Solvent. This consists of the PVDS and SDENSITY keywords:

Table 64.2 PVT data

Keyword	Description
SDENSITY	Surface density of the Solvent
PVDS	Pressure dependent data for each PVT region (analogous to PVDG).

The mixing parameter ω should be input in the PROPS section under the keyword TLMIXPAR. A value in the range 0 to 1 must be supplied for each miscible region. If a different mixing parameter is required for the effective densities, a second mixing parameter can be supplied for each miscible region. If the grid contains any cells whose MISCNUM region is zero, the mixing parameter is automatically set to zero in those cells. Use the mnemonic TLMIXPAR in the RPTPROPS keyword to echo the TLMIXPAR data.

The miscibility region numbers are specified using keyword MISCNUM in the REGIONS section. The parameter NTMISC set in the RUNSPEC section keyword MISCIBLE sets the maximum number of regions which can be defined. In any cell with a MISCNUM set to zero, the components are assumed to be immiscible and the mixing calculation is not used.

Relative permeability data

The relative permeability data needs to be specified as follows:

Table 64.3 Relative permeability data

Keyword	Description
SGFN	For K_{rg} in immiscible regions (either MISCNUM=0 or $S_{gas} \gg 0$)
SWFN	For any cases containing water.
SOF2	Either solvent/oil cases with MISCNUM=0 in at least one cell. SOF2 gives the K_{rog}. or 4-phase cases. SOF2 gives the K_{rhc} (the K_r of the miscible hydrocarbon phase to water).
SOF3	Either 3-phase cases with MISCNUM=0 in at least one cell. or 4-phase cases for the regions where the displacement is immiscible.
SSFN	In immiscible or miscible 4-component cases.

In 4-component cases the miscibility function needs to be defined to control the transition from miscible to immiscible relative permeabilities. This is done using the MISC keyword in the PROPS section.

The miscible solvent-oil relative permeability curves can modified using the MSFN keyword. If the MSFN keyword is omitted, straight line relative permeability curves will be used.

The transition between immiscibility and miscibility

The transition between an immiscible and a miscible displacement is controlled by the following keywords:

Table 64.4 Transition keywords

Keyword	Description
MISC	Tables of the miscibility function versus local solvent fraction
PMISC	Tables of the miscibility function versus oil pressure.

MISC is required for all 4-phase runs, but PMISC is optional. If PMISC is not specified, the solvent/oil displacement is assumed to be miscible at all pressures.

SUMMARY section data

New SUMMARY section keywords have been added to allow the output associated with the solvent component to be plotted using the GRAF program.

These keywords are the same set as for the gas phase; they are differentiated from the gas keywords by replacing the G by an N.

See "Summary of keywords" on page 972 for a list of these keywords.

Injecting solvent into the reservoir

Solvent can be injected by specifying a gas injection well. A separate keyword WSOLVENT can then be used to specify a fraction of the gas flow as solvent.

It is also possible to initialize the model by enumeration and to include a Solvent saturation. This is achieved using the SSOL keyword in the SOLUTION section.

Control of numerical diffusion in miscible flow simulation

The relative permeability of the oil and solvent components in the Todd- Longstaff model are specified by equation [EQ 64.4]. The relative permeability of each component is essentially proportional to the saturation of each component for low saturations. In such cases the influence of numerical diffusion in spreading the miscible components can be unacceptably high and some control of the numerical diffusion errors is required.

A two point upstream algorithm, operating only on the miscible component relative permeabilities, is available for use in the ECLIPSE Miscible Flood option to control the numerical diffusion of oil and solvent. In the present version of ECLIPSE the two-point upstreaming algorithm is only available for 3-phase cases.

The two-point upstream projection technique is illustrated in Figure 64.1.

Figure 64.1 The two point upstream projection technique

We need to compute the flow of component C between cells I and J which we take to be aligned along the X-axis. Suppose that cell I is the upstream cell for the flow of component C between I and J. Let cell K be the second upstream cell to cell J and K_{ck}, K_{ci} and K_{cj} denote the component relative permeabilities in cells K, I and J respectively. DX_k, DX_i and DX_j are the X-direction cell sizes.

For single point upstream weighting, the flow rate of component C across the I-J cell interface is $F_{ij,c}$ where

$$F_{ij,c} = T_{ij} \cdot \frac{K_{ci}}{B_{ci}\,\mu_{ci}} \cdot \delta\Phi_{ij,c} \qquad \text{[EQ 64.39]}$$

where

T_{ij} is the transmissibility between cells I and J

$\delta\Phi_{ij,c}$ is the potential difference between cells I and J for component C

B_c, μ_c are the formation volume factor and viscosity of component C

The two point upstreaming algorithm operating on component relative permeabilities replaces K_{ci} in the equation [EQ 64.39] by a linearly extrapolated relative permeability, $K_{ci}*$, obtained from cells I and K,

$$K_{ci}* = \frac{2\mathrm{DX}_i + \mathrm{DX}_k}{\mathrm{DX}_i + \mathrm{DX}_k} K_{ci} - \frac{\mathrm{DX}_i}{\mathrm{DX}_i + \mathrm{DX}_k} K_{ck} \qquad \text{[EQ 64.40]}$$

The above expression for $K_{ci}*$ can yield negative values or values greater than unity, and in such cases needs to be restricted to physical values. This is most simply achieved by constraining $K_{ci}*$ to lie between the values K_{ci} and K_{cj}. Thus,

$$\mathrm{MIN}(K_{ci}, K_{cj}) < K_{ci}* < \mathrm{MAX}(K_{ci}, K_{cj}) \qquad \text{[EQ 64.41]}$$

In some cases a single point upstream calculation of $K_{ci}*$ (that is, $K_{ci}* = K_{ci}$) should be used instead of the two point upstream value. Such cases are

1. Cell K lies outside the reservoir mesh (that is the K-I cell boundary is an edge of the reservoir)
2. Cell K is inactive
3. The K-I cell boundary contains a zero transmissibility multiplier
4. Cells I and J are coupled by a non-neighbor connection.

Summary of keywords

RUNSPEC section

The keywords are:

Table 64.5 RUNSPEC section

Keyword	Description
SOLVENT	Activates the separate Solvent component.
MISCIBLE	Initiates the mixing calculation.

PROPS section

The keywords are:

Table 64.6 PROPS section

Keyword	Description
MISC	Miscibility function tables.
MSFN	Miscible gas/oil saturation functions.
PMISC	Pressure dependent miscibility tables.
PVDS	Pressure dependent PVT properties for solvent gas.
RPTPROPS	Controls output from the PROPS section. Mnemonic MISC or PMISC outputs the miscibility function and the pressure miscibility function. Mnemonic PVDS outputs solvent PVT properties. Mnemonic SSFN outputs solvent/gas relative permeabilities.
SDENSITY	Solvent density at surface conditions.
SGCWMIS	Miscible critical gas saturation tables.
SORWMIS	Miscible residual oil saturation tables.
SSFN	Gas/solvent relative permeability functions.
TLMIXPAR	Todd-Longstaff mixing parameter.

REGIONS section

The keywords are:

Table 64.7 REGIONS section

Keyword	Description
MISCNUM	Miscibility region numbers.
RPTREGS	Controls output from the REGIONS section. Mnemonic MISCNUM outputs miscibility region numbers.

SOLUTION section

The keywords are:

Table 64.8 SOLUTION section

Keyword	Description
RPTSOL	Controls on output from SOLUTION section. Mnemonic FIPSOL outputs solvent fluid in place reports. Mnemonic FLOSOL outputs solvent inter-block flow rates. Mnemonic KRN outputs grid block solvent relative permeabilities. Mnemonic SSOL outputs initial solvent saturations.
SSOL	Initial solvent saturations.

SUMMARY section

The following keywords control output of data specific to the Solvent model.

Table 64.9 Solvent SUMMARY output controls

Field	Group	Well	Connection	Region	Block	Information
			CNFR			Solvent Flow Rate (+ or -)
FNPR	GNPR	WNPR				Solvent Production Rate
FNPT	GNPT	WNPT	CNPT			Solvent Production Total
FNIR	GNIR	WNIR				Solvent Injection Rate
FNIT	GNIT	WNIT	CNIT			Solvent Injection Total
					BNSAT	Solvent SATuration
FNIP				RNIP	BNIP	Solvent In Place
				RNFT		Solvent Flow
					BNKR	Solvent relative permeability

Note ECLIPSE reports the field, group and well gas rates and totals for the **total** gas phase, including the solvent gas. The connection gas flow rates and totals are reported solely for the component gas phase, excluding the solvent gas.

SCHEDULE section

The keywords are:

Table 64.10 SCHEDULE section

Keyword	Description
RPTSCHED	Controls on output from SCHEDULE section. Mnemonic FIPSOL outputs solvent fluid in place reports. Mnemonic FLOSOL outputs solvent inter-block flow rates. Mnemonic KRN outputs grid block solvent relative permeabilities. Mnemonic SSOL outputs current solvent saturations.
WSOLVENT	Sets solvent concentrations for gas injection wells.

Example problem

RUNSPEC

```
RUNSPEC
TITLE
SPE Fifth Comparison Test Problem - Scenario 2 - 4 Phase
DIMENS
 7    7    3  /
NONNC
OIL
WATER
GAS
DISGAS
SOLVENT
FIELD
MISCIBLE
 1   20    'NONE'   /
EQLDIMS
 1  100   10    1    1 /
WELLDIMS
 3    1    1    3 /
START
 1 'JAN' 1986  /
NSTACK
 24 /
FMTIN
UNIFOUT
UNIFIN
```

GRID

```
GRID  ============================================================
EQUALS
      'DX'    500       /
      'DY'    500       /
      'PORO'  0.3       /
      'DZ'    20        1  7  1  7  1  1  /
      'PERMX' 500       /
      'PERMZ' 50        /
      'TOPS'  8325      /
      'DZ'    30        1  7  1  7  2  2  /
      'PERMX' 50        /
      'PERMZ' 50        /
      'DZ'    50        1  7  1  7  3  3  /
      'PERMX' 200       /
      'PERMZ' 25        /
/

COPY
      'PERMX'    'PERMY'   1  7  1  7  1  3  /
/
RPTGRID
 'DX'  'DY'  'DZ'  'PERMX'  'PERMY'  'PERMZ'  'MULTZ'
 'PORO'  'TOPS'  'PORV'  'TRANX'  'TRANY'  'TRANZ'  /
```

PROPS

```
PROPS  ===========================================================
SWFN
    0.20    0.0      45.00
    0.2899  0.0022   19.03
    0.3778  0.0180   10.07
    0.4667  0.0607    4.90
    0.5556  0.1438    1.80
    0.6444  0.2809    0.50
    0.7000  0.4089    0.05
    0.7333  0.4855    0.01
    0.8222  0.7709    0.00
    0.9111  1.0000    0.00
    1.0000  1.0000    0.00  /
```

```
SGFN
    0.0000  0.0       0.0
    0.0500  0.0       0.0
    0.0889  0.001     0.0
    0.1778  0.01      0.0
    0.2667  0.03      0.001
    0.3556  0.05      0.001
    0.4444  0.1       0.03
    0.5333  0.2       0.8
    0.6222  0.35      3.0
    0.6500  0.39      4.0
    0.7111  0.56      8.0
    0.8000  1.0      30.0   /
SOF2
    0.0      0.0
    0.0889   0.0
    0.1778   0.0
    0.2667   0.0
    0.3000   0.0
    0.3556   0.0123
    0.4444   0.0835
    0.5333   0.2178
    0.6222   0.4153
    0.7101   0.6769
    0.8000   1.0000  /
SOF3
--           KROW       KROG
    0.0      0.0        0.0
    0.0889   0.0        0.0
    0.15     1*         0.0
    0.1778   0.0        0.0110
    0.2667   0.0        0.0370
    0.3000   0.0        1*
    0.3556   0.0123     0.0878
    0.4444   0.0835     0.1715
    0.5333   0.2178     0.2963
    0.6222   0.4153     0.4705
    0.7101   0.6769     0.7023
    0.75     1*         0.8800
    0.8000   1.0000     1.0000    /
SSFN
--          KRG*     KRS*
  0         0.0     0.0
  1.0       1.0     1.0
 /
MISC
  0.0 0.0
  0.1 1.0
  1.0 1.0 /
PVTW
       4000.0     1.000        3.3D-6           0.70            0 /
ROCK
        4000.0        5.0D-6          /
```

```
DENSITY
         38.53  62.40   0.06867  /
SDENSITY
                        0.06243  /
TLMIXPAR
 0.7 /
PVDG
     14.7 211.4160   0.0107
    500.0   5.9242   0.0127
   1000.0   2.8506   0.0134
   1200.0   2.3441   0.0138
   1500.0   1.8457   0.0145
   1800.0   1.5202   0.0153
   2000.0   1.3602   0.0159
   2302.3   1.1751   0.0170
   2500.0   1.1025   0.0177
   3000.0   0.9852   0.0195
   3500.0   0.9116   0.0214
   4000.0   0.8621   0.0232
   4500.0   0.8224   0.0250
   4800.0   0.8032   0.0261     /
PVDS
     14.7 223.2140   0.011
    500.0   5.6022   0.012
   1000.0   2.5310   0.013
   1200.0   2.0354   0.014
   1500.0   1.5593   0.016
   1800.0   1.2657   0.018
   2000.0   1.1296   0.019
   2302.3   0.9803   0.022
   2500.0   0.9085   0.023
   3000.0   0.7807   0.027
   3500.0   0.6994   0.031
   4000.0   0.6430   0.034
   4500.0   0.6017   0.037
   4800.0   0.5817   0.038      /
```

```
PVTO
    0.0000    14.7  1.0348  0.310   /
    0.1176   500.0  1.1017  0.295   /
    0.2226  1000.0  1.1478  0.274   /
    0.2677  1200.0  1.1677  0.264   /
    0.3414  1500.0  1.1997  0.249   /
    0.4215  1800.0  1.2350  0.234   /
    0.4790  2000.0  1.2600  0.224   /
    0.5728  2302.3  1.3010  0.208
            3302.3  1.2988  0.235
            4302.3  1.2966  0.260   /
    0.6341  2500.0  1.3278  0.200   /
    0.7893  3000.0  1.3956  0.187   /
    0.9444  3500.0  1.4634  0.175   /
    1.0995  4000.0  1.5312  0.167   /
    1.2547  4500.0  1.5991  0.159   /
    1.3478  4800.0  1.6398  0.155
            5500.0  1.6305  0.168   /
/
RPTPROPS
 'SOF2'   'SWFN'   'SGFN'   'PVTW'
 'PVTG'   'DENSITY'  'ROCK'  /
```

SOLUTION

```
SOLUTION ============================================================
=
EQUIL
       8400   4000   9900    0     1000    0      1      0       0
/

RSVD
       8200  0.5728
       8500  0.5728 /
RPTSOL
 'PRES'  'SOIL'  'SWAT'  'SGAS'  'FIP'  /
```

SUMMARY

```
SUMMARY =============================================================
FOPR
FNPR
FGPR
BOSAT
 4 4 1 /
/
BNSAT
 4 4 1 /
/
```

SCHEDULE

```
SCHEDULE ============================================================
=
RPTSCHED

 'PRES'  'SOIL'  'SWAT'  'SGAS'  'SUMMARY=2'
 'CPU=2'  /
WELSPECS
    'PRODUCER' 'G'    7  7    8400 'OIL'  /
    'INJ-G   ' 'G'    1  1    8335 'GAS'  /
    'INJ-W   ' 'G'    1  1    8335 'WAT'  /
/
COMPDAT
    'PRODUCER'   7 7 3  3 'OPEN' 0   -1   0.5  10000.0  /
    'INJ-G   '   1 1 1  1 'OPEN' 0   -1   0.5  10000.0  /
    'INJ-W   '   1 1 1  1 'OPEN' 0   -1   0.5  10000.0  /
  /
WCONPROD
     'PRODUCER' 'OPEN' 'ORAT' 12000  4*  3000 /
  /
```

```
WECON
   'PRODUCER'  1*  1*  5.0  10.0  1*  'WELL'  'YES'  /
   /
WCONINJ
  'INJ-W' 'WAT' 'OPEN' 'RATE'  45000   1*   1*   1*   4500  /
  'INJ-G' 'GAS' 'OPEN' 'RATE'  20000   1*   1*   1*   4500  /
  /
WSOLVENT
  'INJ-G'  1.0 /
  /
-- YEAR 1 -------------------------------------------------------
INCLUDE
 'CASE2.ONEYEAR' /
-- YEAR 2 -------------------------------------------------------
INCLUDE
 'CASE2.ONEYEAR' /
END
```

Included file

```
-- INCLUDED FILE:  CASE2.ONEYEAR
-- FIRST 3 MONTHS  ------------------------------------------------
--*WATER*--
TUNING
2.0 /
/
/
WELOPEN
  'INJ-G' 'SHUT' /
  'INJ-W' 'OPEN'/
/
TSTEP
   91.25
/
-- SECOND 3 MONTHS ------------------------------------------------
--*GAS*--
TUNING
2.0 /
/
/
WELOPEN
  'INJ-W' 'SHUT' /
  'INJ-G' 'OPEN' /
/
TSTEP
   91.25
/
--THIRD 3 MONTHS ------------------------------------------------
--*WATER*--
```

```
TUNING
2.0  /
/
/
WELOPEN
 'INJ-G' 'SHUT' /
 'INJ-W' 'OPEN' /
/
TSTEP
   91.25
/
-- FOURTH 3 MONTHS ----------------------------------------------
--*GAS*--
TUNING
2.0 /
/
/
WELOPEN
 'INJ-W' 'SHUT' /
 'INJ-G' 'OPEN' /
 /
TSTEP
   91.25
/
-----------------------------------------------------------------
```

Surface Tension Effects

Chapter 65

Evaluation of surface tensions

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

The treatment of surface tension effects follows that of [Ref. 52].

Surface tensions are obtained using the Macleod-Sugden correlation, given by

$$\sigma = \left| \sum_{i=1}^{N_c} [P]_i (b_L^m x_i - b_V^m y_i) \right|^4 \qquad \text{[EQ 65.1]}$$

where σ is the surface tension (always in dynes/cm) and (x_i, y_i) are the liquid and vapor mole fractions. The liquid and vapor phase molar densities (b_L^m, b_V^m) are converted into g-moles/cc for this evaluation. The parachors $[P]_i$ are data provided by the user for this choice of units. For one-phase states the surface tension is zero.

Warnings of un-physically large values of surface tension can be requested using the second data item in the `MISCSTR` keyword.

Use of surface tensions

Relative permeabilities

The expression for σ will go sharply to zero as the phase compositions become equal, such as occurs near the critical point. This can therefore be used to control an interpolation between miscible (straight line) and immiscible (entered data curves). Define an interpolation factor as:

$$F = \left(\frac{\sigma}{\sigma_0}\right)^N \qquad \text{[EQ 65.2]}$$

where

σ_0 is a reference surface tension

N is the exponent set by keyword `MISCEXP`.

By default this is the average initial surface tension, but may be set directly using the `MISCSTR` keyword. The interpolation factor may then be used to obtain a weighted average of miscible and immiscible hydrocarbon relative permeabilities:

$$K_{ro} = FK_{ro}^{imm} + (1-F)K_{ro}^{mis} \qquad \text{[EQ 65.3]}$$

where the reservoir oil saturation is scaled such that both miscible and immiscible relative permeabilities have the same critical saturation given by

$$S_{cr} = FS_{cr}^{imm} + (1-F)S_{cr}^{mis}\,. \qquad \text{[EQ 65.4]}$$

where

- S_{cr}^{imm} is the critical immiscible saturation, generally obtained from a user-defined rock curve. In the case of hysteresis it may be the critical point of a scanning curve (see "Hysteresis" on page 517 for further information). Note that if no miscible saturation is specified, S_{cr} reduces to FS_{cr}^{imm}, as depicted in the schematic graphs below.
- S_{cr}^{mis} is the critical miscible saturation obtained from a miscible saturation table specified using the `MISCNUM` keyword. If S_{cr}^{imm} is zero, then S_{cr} still scales to $(1-F)S_{cr}^{mis}$.

Figure 65.1 Two graphs to illustrate the scaling of a rock curve

Figure 65.2 Two graphs to illustrate the scaling of a straight line

Both contributions thus have the same endpoint, which controls the critical saturation as a function of F. Ultimately, as $F \to 0$, K_{ro}^{mis} becomes the dominant contribution, and this contribution becomes a straight line. However, as the tension goes to zero the critical saturation will go only to the critical miscible value specified in the miscible table.

Note that if the surface tension exceeds the reference value, F will be taken as 1.0 and the relative permeability curves will be immiscible.

Note the similarity between this model and that provided by the Velocity Dependent Relative Permeability option: see "Velocity-dependent relative permeabilities" on page 716.

Note that the length of time stepping is of great importance in simulating surface tension effects. Also note that F will be taken as 1.0 for single-phase cells.

Capillary pressures

Capillary pressures must also go to zero as the surface tension vanishes, and here one would expect a linear dependence:

$$P_{cog}^{\text{corr}} = \left(\frac{\sigma}{\sigma_0}\right) P_{cog} \qquad \text{[EQ 65.5]}$$

where P_{cog} is the user supplied curve entered with `SGFN`. The effect of this choice of interpolation factor is that the equilibration is done using the supplied curves.

By default the capillary pressure will be taken as the input P_{cog} when the surface tension exceeds the reference value. The maximum surface tension at which the scaling is halted can be entered in the `MISCSTR` keyword.

Tabular input

Alternatives to the treatment of miscibility described above are provided by two tabular methods of control:

- `STVP`

  This provides the variation of surface tension with pressure. This should decrease with increasing pressure, falling to a low value near P_{sat}.

- FVST
  This provides the variation of the relative permeability miscibility factor with surface tension. F_K is the weight factor for the immiscible contribution, (see below), and should increase with increasing surface tension.

These two keywords are independent. If STVP is not entered, the Macleod-Sugden correlation will be used to obtain surface tensions. If FVST is not entered, the miscibility factor will be obtained using the Coats expression:

$$F_K = \left(\frac{\sigma}{\sigma_0}\right)^P \qquad \text{[EQ 65.6]}$$

where P is the exponent set using MISCEXP, and defaulted to 0.25.

In either case F_K is used to define the oil and gas relative permeability curves:

$$K_r = FK_r^{imm} + (1-F)K_r^{mis} \qquad \text{[EQ 65.7]}$$

The two contributions are scaled so that they both have the same critical saturation, given by [EQ 65.4].

FVST avoids the use of both P and σ_0 in relative permeability calculations. σ_0 is still used in modifying capillary pressures:

$$P_{cog}^{corr} = \left(\frac{\sigma}{\sigma_0}\right)P_{cog} \qquad \text{[EQ 65.8]}$$

We thus still need a value for σ_0 if STVP is used. This may be entered with MISCSTR. If not, the largest value in any of the STVP tables will be used.

STVP and FVST are pressure tables, that is, one is required for each pressure table (PVTNUM) region. If both STVP and FVST are entered the effect is to make F_k a function of pressure.

Use of the MISCNUM option

In the default treatment of the variation of relative permeabilities with surface tension a straight line relative permeability curve is used as the limit as $\sigma \rightarrow 0$. This is a natural limit, as the two phases become identical and flow at the same velocity. It also gives us continuity of flow as the system makes a transition to the one hydrocarbon phase regime.

However, there are cases where another limit may be used. When straight line relative permeabilities are used, there may be excessive numerical dispersion on coarse full-field grids, so that the smearing of the fronts offsets the gain in recovery expected from the approach to miscibility. There is also some experimental evidence [Ref. 53] that relative permeabilities may be significantly different from straight lines even at very low surface tensions.

To model such cases the `MISCNUM` keyword may be used. This allows a set of relative permeability tables to be nominated to be used rather than the straight line default as $\sigma \rightarrow 0$. As in the default case the tables are scaled to the same critical saturation. In the limit of zero surface tension only the `MISCNUM` curve will be used.

Care is need when using this model - in particular when a transition is made from a near miscible two phase hydrocarbon state to a one phase hydrocarbon state. If the `MISCNUM` curves are far from a straight line this may involve a flow discontinuity which may make it difficult for the simulator to converge the flow equations.

The Surfactant Model

Chapter 66

Introduction

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL

Most large oil fields are now produced with some type of secondary pressure maintenance scheme, such as water flooding. Water flooding can increase recovery from around 1% to the 20-40% range. The remaining oil can be divided into two classes, firstly residual oil to the water flood, and secondly oil bypassed by the water flood. A surfactant flood is a tertiary recovery mechanism aimed at reducing the residual oil saturation in water swept zones.

Typically a water flood that contacts 100% of a given oil zone will leave a residual oil saturation of, say, 30%. This is the saturation at which the oil phase relative permeability value is zero. The oil is immobile at this saturation because of the surface tension between oil and water; the water pressure alone is unable to overcome the high capillary pressure required to move oil out of very small pore volumes.

A surfactant offers a way of recovering the residual oil by reducing the surface tension between the oil and water phases. A very low oil-water surface tension reduces the capillary pressure and hence allows water to displace extra oil. If it is possible to reduce the surface tension to zero, then theoretically the residual oil can be reduced to zero. In practice the residual oil to even high concentrations is unlikely to lead to 100% recovery of swept zones.

One of the effects that will influence the success or failure of a surfactant flood is the tendency of the surfactant being used to be adsorbed by the rock. If the adsorption is too high, then large quantities of surfactant will be required to produce a small quantity of additional oil.

The ECLIPSE Surfactant model does not aim to model the detailed chemistry of a surfactant process, but rather to model the important features of a surfactant flood on a full field basis.

The simulation model

Surfactant conservation equation

The distribution of injected surfactant is modeled by solving a conservation equation for surfactant within the water phase. The surfactant concentrations are updated fully-implicitly at the end of each time-step after the oil, water and gas flows have been computed. The surfactant is assumed to exist only in the water phase, and the input to the reservoir is specified as a concentration at a water injector (see the `WSURFACT` keyword).

Calculation of the capillary number

The capillary number is a dimensionless group that measures the ratio of viscous forces to capillary forces. The capillary number is given by:

$$N_c = \frac{|K \cdot \mathrm{grad}P|}{ST} C_{\mathrm{unit}} \qquad \text{[EQ 66.1]}$$

where

K is the permeability

P is the potential

ST is the interfacial tension (see `SURFST` keyword)

C_{unit} is the conversion factor depending on the units used.

$|K \cdot \mathrm{grad}P|$ is calculated as

$$|K \cdot \mathrm{grad}P| = \sqrt{(K_x \cdot \mathrm{grad}P_x)^2 + (K_y \cdot \mathrm{grad}P_y)^2 + (K_z \cdot \mathrm{grad}P_z)^2} \qquad \text{[EQ 66.2]}$$

where for cell i

$$K_x \cdot \mathrm{grad}P_x = 0.5\left[\left(\frac{K_x}{D_x}\right)_{i-1,i} \cdot (P_i - P_{i-1}) + \left(\frac{K_x}{D_x}\right)_{i,i+1} \cdot (P_{i+1} - P_i)\right] \qquad \text{[EQ 66.3]}$$

and similarly for the y and z directions.

The K/D value is calculated in an analogous manner to the transmissibility and depends on how the geometry was specified.

The surface tension is a tabulated function of the surfactant concentration.

The unit conversion constant is as follows:

- `METRIC`

  Pressure bars

  Length m

  Permeability mD

  Surface Tension N/m

  C_{unit} = 9.869234E-11

- `FIELD`

Pressure psi

Length ft.

Permeability mD

Surface Tension lbs/in

c_{unit} = 1.274784E-13

- `LAB`

  Pressure atm

  Length cm

  Permeability mD

  Surface Tension dyne/cm

  c_{unit} = 1.0E-5

Relative permeability model

The Relative Permeability model is essentially a transition from immiscible relative permeability curves at low capillary number to miscible relative permeability curves at high capillary number. You supply a table that describes the transition as a function of $\log_{10}$(capillary number).

The relative permeability used at a value of the miscibility function between the two extremes is calculated in two steps. Firstly the end points of the curve are interpolated and both the immiscible and the miscible curves are scaled to honor these points. The relative permeability values are looked up on both curves, and the final relative permeability is taken as an interpolation between these two values. This procedure is illustrated below for the oil to water curve:

Figure 66.1 Calculation of the relative permeability

The treatment of the water relative permeability is analogous to the oil case. In the 3 phase case, the oil-to-water relative permeability is calculated using the above procedure prior to calculating the 3 phase relative permeability. The surfactant relative permeability model can be used with either of the two STONE's 3 phase relative permeability models or the default model.

The well relative permeability will be taken as the block relative permeability following the above procedure, if the connection relative permeability tables are defaulted. However, if the connection relative permeability table number is set in the `COMPDAT` keyword, or if the `COMPRP` keyword is used, then that relative permeability table will be used with no modification to account for the surfactant present in the grid block.

The interpolation of relative permeability, as described above, is only performed for blocks with a non-zero surfactant concentration. For blocks with zero surfactant concentration the immiscible curves are used. The pre-2010.1 behavior of interpolating relative permeability for all blocks, regardless of surfactant concentration, can be restored by setting item 147 of the `OPTIONS` keyword to a non-zero value.

The surfactant model is currently not compatible with the relative permeability hysteresis option.

Capillary pressure

The water oil capillary pressure will reduce as the concentration of surfactant increases; indeed it is the reduction in the oil water capillary pressure that gives rise to the reduction in the residual oil saturation. The oil water capillary pressure is taken as:

$$P_{cow} = P_{cow}(S_w)\frac{ST(C_{\text{surf}})}{ST(C_{\text{surf}} = 0)} \qquad \text{[EQ 66.4]}$$

where

$ST(C_{surf})$ is the surface tension at the present surfactant concentration.

$ST(C_{surf} = 0)$ is the surface tension at zero concentration.

$P_{cow}(S_w)$ is the capillary pressure from the immiscible curves initially scaled to the interpolated end-points calculated in the relative permeability model.

Note The scaling of the immiscible capillary pressure to honor the interpolated endpoints calculated in the relative permeability model, was not done in versions before 2009.1. Item 119 of the `OPTIONS` keyword can be used to restore the pre-2009.1 calculations.

Water PVT properties

The surfactant modifies the viscosity of the pure or salted water input using the `PVTW` or the `PVTWSALT` keyword respectively. The surfactant viscosity input as a function of surfactant concentration using the `SURFVISC` keyword is used to calculate the water-surfactant solution viscosity as follows:

$$\mu_{ws}(C_{surf}, P) = \mu_w(P)\frac{\mu_S(C_{surf})}{\mu_w(P_{ref})} \qquad \text{[EQ 66.5]}$$

If the Brine option is active (and the Polymer option is inactive), the previous equation becomes a function of salt concentration c_s as well:

$$\mu_{ws}(C_{surf}, P, c_s) = \mu_w(P, c_s) \cdot \frac{\mu_S(C_{surf})}{\mu_w(P_{ref}, c_{sref})} \qquad \text{[EQ 66.6]}$$

where

μ_S is the viscosity from the `SURFVISC` keyword

μ_w is the viscosity from the `PVTW` or `PVTWSALT` keywords

μ_{ws} is the viscosity of the water-surfactant mixture

P_{ref} is the reference pressure in the `PVTW` or `PVTWSALT` keywords

c_{sref} is the reference salt concentration in the `PVTWSALT` keywords.

Treatment of adsorption

The adsorption of surfactant is assumed to be instantaneous, and the quantity adsorbed is a function of the surrounding surfactant concentration. The user is required to supply an adsorption isotherm as a function of surfactant concentration (see the `SURFADS` keyword). As an alternative to `SURFADS`, the surfactant adsorption can be specified using keyword `ADSORP` with `SURFACT` as an argument. `ADSORP` specifies coefficients for a generic, analytical adsorption isotherm which allows for dependencies of adsorption on rock permeability and effective brine salinity. When used in conjunction with the Multi-Component Brine Model, an effective salinity for surfactant can be calculated which can then be used in the adsorption isotherm. See "Brine Tracking" for information on the Multi-Component Brine Model and the calculation of effective salinity for surfactant. See the `ADSORP` keyword for information on the functional dependence of adsorption on salinity.

The quantity of surfactant adsorbed on to the rock is given by:

$$\text{Mass of adsorbed surfactant} = \text{PORV} \cdot \frac{1-\phi}{\phi} \cdot \text{MD} \cdot \text{CA}(C_{\text{surf}}) \qquad \text{[EQ 66.7]}$$

where

PORV is the Pore volume of the cell

ϕ is the Porosity

MD is the mass density of the rock (see SURFROCK keyword)

$\text{CA}(C_{\text{surf}})$ is the adsorption isotherm as a function of local surfactant concentration in solution.

There are two adsorption models that can be selected. The first model ensures that each grid block retraces the adsorption isotherm as the surfactant concentration falls in the cell. The second model assumes that the adsorbed surfactant concentration on the rock may not decrease with time and hence does not allow for any de-adsorption.

In the present version of ECLIPSE the adsorption concentrations are updated explicitly for the surfactant concentration. In the future it may be necessary to implement the adsorption implicitly.

Modeling the change to wettability

This feature enables the modeling of changes to wettability of the rock due to the accumulation of surfactant. It is activated by using the SURFACTW keyword, which also activates the Surfactant model.

Assuming that the usual SATNUM keyword defines oil-wet immiscible saturation functions (input with keywords SWFN, SOF2, SOF3 or SWOF), the user defines additional immiscible saturation functions with these same keywords and these are then taken to model the water-wet situation. The keyword SURFWNUM in the REGIONS section **must** be used to define the region (water-wet immiscible saturation function table) number of each grid block.

In calculating the immiscible relative permeabilities and capillary pressure, the water-wet and oil-wet endpoints are firstly interpolated and the curves scaled to honor these points. Then a weighted average of F times the oil-wet value and (1-F) times the water-wet value is used. The fraction, F, is the same fraction used for interpolating the endpoints and is a function of the adsorbed surfactant concentration. It **must** be defined by tables input using the SURFADDW keyword in the PROPS section; the number of the table used is that given in the SURFWNUM keyword.

The formula used for the new immiscible relative permeability is

$$k_r = Fk_r^{ow} + (1-F)k_r^{ww}$$

where

F is given by SURFADDW

k_r^{ow} is the scaled oil-wet kr specified by SATNUM

k_r^{ww} is the scaled water-wet kr specified by SURFWNUM.

This defines new immiscible saturation functions that are then used in the Relative permeability and Capillary pressure models described previously.

Using the surfactant model

The model is activated by specifying the keyword SURFACT in the RUNSPEC section.

It is possible to output the geometric data used by the surfactant model by setting the 'KOVERD' mnemonic in RPTGRID. This outputs a map of the K/D values used in the calculation of the capillary number.

There are three obligatory keyword in the PROPS section, and two optional keywords for cases in which adsorption takes place:

Table 66.1 SURFACTANT keywords

Keyword	Description	Notes
SURFST	Water-oil surface tension in the presence of surfactant	(Obligatory)
SURFVISC	Modified water viscosity	(Obligatory)
SURFCAPD	Capillary de-saturation data	(Obligatory)
SURFADS	Adsorption isotherm	(Optional)
SURFROCK	Rock properties and adsorption model indicator	(If SURFADS is present)

The surfactant properties data can be written to the PRINT file by setting the 'SURFVISC' mnemonic in the RPTPROPS keyword.

The miscible relative permeability curves are specified using the SURFNUM keyword in conjunction with the usual saturation function keywords. If a non-zero capillary pressure is entered in the SWFN or SWOF table, this will be ignored and the capillary pressure will be taken as the appropriate fraction of the capillary pressure for the SATNUM table.

The injected surfactant concentration of a water injector is specified using the WSURFACT keyword. Output of the surfactant concentration, capillary number, adsorbed concentration and fluid in place data can be controlled using mnemonics 'SURFBLK', 'SURFADS' and 'FIPSURF' in the RPTSCHED keyword. In addition a field/group/well/connection summary can be obtained at each report time by adding the mnemonic 'WELLS' to RPTSCHED. An extra mode of controlling the time step is available, which sets a target surfactant change in a time step. The default is to have no control. This control can be set by specifying a target TRGSFT in record 2 of the TUNING keyword.

The list of SUMMARY keywords has been extended to output surfactant specific data. These keywords follow the passive tracer format with a keyword concatenated with the tracer name. In this case surfactant is referred to as SUR. For example the keyword for the field total surfactant production would be FTPRSUR.

Diffusion can be applied to a surfactant if the Environmental Tracer option is active (Item 4 of the TRACERS keyword in the RUNSPEC section). This is done by using the TRDIFSUR keyword (that is the TRDIF keyword for the tracer called 'SUR' which is the surfactant).

Using partitioned tracers

Partitioned tracers allow a given tracer to dissolve in a secondary phase, as a function of pressure. With surfactants the primary phase is water but the secondary phase can be oil, say, and this feature can model the partitioning of the surfactant into the oil. See "Tracer Tracking" on page 1117 for a more detailed discussion.

Restrictions

Changing the pore volume in the `EDIT` section using `PORV` is not allowed with the surfactant option.

Summary of keywords

RUNSPEC section

The keywords are:

Table 66.2 RUNSPEC section

Keyword	Description
PARTTRAC	Activates and dimensions the Partitioned Tracer option.
SURFACT	Activates the Surfactant model.
SURFACTW	Activates the Surfactant model and enables modeling of changes of wetability.

GRID section

The keywords are:

Table 66.3 GRID section

Keyword	Description
RPTGRID	Controls output from the GRID section. Mnemonic KOVERD outputs the K/D values used in the calculation of the capillary number.

PROPS section

The keywords are:

Table 66.4 PROPS section

Keyword	Description
RPTPROPS	Controls output from the PROPS section. Mnemonics SURFVISC and so on output surfactant properties.
SOCRS	Scaled critical oil-in-water saturations at miscible conditions.
SURFADDW	Defines weighting between oil-wet and water-wet relative permeabilities as a function of the adsorbed surfactant mass.
SURFADS	Surfactant adsorption isotherm.
ADSORP	Analytical adsorption isotherm with salinity dependence.
SURFCAPD	Surfactant capillary de-saturation data.
SURFROCK	Surfactant-rock properties and adsorption model indicator.
SURFST	Water-oil surface tension in the presence of surfactant.

Table 66.4 PROPS section

Keyword	Description
SURFVISC	Modified water viscosity.
TRACER	Defines the tracers (for the Partitioned Tracer option).
TRACERKP	Defines the functions K(P) for the partitioned tracers.

REGIONS section

The keywords are:

Table 66.5 REGIONS section

Keyword	Description
SURFNUM	Surfactant miscible region numbers.
SURFWNUM	Defines the region number for water-wet immiscible saturation function table; the number also defines the table describing the fraction of oil/water miscibility to use.
TRKPFxxx	Specifies the partitioning function for tracer xxx for each grid block ('xxx' = 'SUR' for surfactant).

SOLUTION section

The keywords are:

Table 66.6 SOLUTION section

Keyword	Description
RPTSOL	Controls output from the SOLUTION section. Mnemonic SOCRS outputs the critical oil saturations following a surfactant flood.

SUMMARY section

The following keywords control output of data specific to the Surfactant model:

Table 66.7 Surfactant SUMMARY output controls

Field	Group	Well	Connection	Region	Block	Information
			CTFRSUR			Flow Rate (+ or -)
FTPRSUR	GTPRSUR	WTPRSUR				Production Rate
FTPTSUR	GTPTSUR	WTPTSUR	CTPTSUR			Production Total
FTIRSUR	GTIRSUR	WTIRSUR				Injection Rate
FTITSUR	GTITSUR	WTITSUR	CTITSUR			Injection Total

Table 66.7 Surfactant SUMMARY output controls

Field	Group	Well	Connection	Region	Block	Information
					BTCNFSUR	Concentration
FTIPTSUR				RTIPTSUR	BTIPTSUR	In Solution
				RTFTTSUR		Flow Total
FTADSUR				RTADSUR	BTADSUR	Adsorption total
					BTCASUR	Log (Capillary Num.)
					BEWV_SUR	Effective water viscosity due to surfactant

SCHEDULE section

The keywords are:

Table 66.8 SCHEDULE section

Keyword	Description
RPTSCHED	Controls output from the SCHEDULE section. Mnemonic - FIPSURF outputs surfactant fluid in place. - SURFADS outputs the current adsorbed surfactant concentration on the formation rock. - SURFBLK outputs the surfactant concentrations and the capillary number.
RPTRST	Controls output to the restart file Mnemonic - VISC allows the output of the effective water viscosity due to surfactant as EWV_SUR
TUNING	Sets simulator control parameters. Item 11 of Record 2 (TRGSFT) sets the target surfactant change.
WSURFACT	Specifies the surfactant concentration of a water injector.

Example problem

RUNSPEC

```
RUNSPEC
TITLE
Example Surfactant Flood Case.
DIMENS
 5    5    1  /
NONNC
OIL
WATER
SURFACT
FIELD
TRACERS
 0    1    0    0 'DIFF  ' /
TABDIMS
 2    2   20    5    6   20 /
REGDIMS
 6    1    0    0    0    1 /
WELLDIMS
 2    1    1    2 /
START
 1 'JAN' 1983  /
NSTACK
 8 /
```

GRID

```
GRID    ============================================================
DXV
   5*75.0 /
PERMX
  25*50.0 /
PERMY
  25*50.0 /
DYV
   5*75.0 /
DZ
  25*30.0 /
TOPS
  25*4000.0 /
PERMY
  25*50.0 /
PORO
  25*0.2 /
```

```
RPTGRID
 'DX'  'DY'  'DZ'  'PERMX'  'PERMY'  'PERMZ'  'MULTX'
 'MULTY'  'MULTZ'  'PORO'  'NTG'  'TOPS'  'PORV'
 'DEPTH'  'TRANX'  'TRANY'  'TRANZ'  'COORDSYS'
 'COORD'  'ZCORN'   'NNC'  'AQNUM'  'AQCON'
 'ALLNNC'  'SIGMAV'  'DZMTRXV'  'DIFFMX'  'DIFFMY'
 'DIFFMZ'  'DIFFMMF'  'DIFFX'  'DIFFY'  'DIFFZ'
 'KOVERD'  /
```

PROPS

```
PROPS   ============================================================
TRACER
 'ONE' 'WAT' /
/
SWFN
-- Table 1 - Immiscible curves - SATNUM
    .25    .0    4.0
    .7    1.0     .0
/
-- Table 2 - Miscible curves   - SURFNUM
    .25    .0    4.0
    .99    1.0     .0
/
SOF2
-- Table 1 - Immiscible curves - SATNUM
    .3000  .0000
    .7600 1.0000
/
-- Table 2 - Miscible curves   - SURFNUM
    .0100  .0000
    .7600 1.0000
/
PVTW
 4000.0  1.0  3.03E-06  .5  0.0 /
 4000.0  1.0  3.03E-06  .5  0.0 /
PVDO
  .0     1.0     2.0
 8000.0   .92    2.0
/
  .0     1.0     2.0
 8000.0   .92    2.0
/
ROCK
 4000.0         .30E-05 /
 4000.0         .30E-05 /
DENSITY
 52.0000  64.0000   .04400 /
 52.0000  64.0000   .04400 /
```

```
-- Water Viscosity vs surfactant concentration tables.
SURFVISC
-- Surfactant      Water viscosity at 4000 psi
-- concentration
--   LB/STB             CENTIPOISE
      0.0                   1.0
      0.1                   1.1
      0.2                   1.2
      0.8                   1.3  /
      0.0                   0.5
      0.1                   0.6
      0.2                   0.7
      0.8                   0.8  /

-- Surfactant Adsorption tables
SURFADS
-- Surfactant       Adsorbed mass
-- concentration
--   LB/STB              LB/LB
     0.0                  0.00000
     0.1                  0.00005
     0.2                  0.00005
     1.0                  0.00005 /
     0.0                  0.00000
     0.1                  0.00005
     0.2                  0.00005
     1.0                  0.00005 /
-- Water/Oil surface tension vs Surfactant concentration
SURFST
-- Surfactant        Water/Oil Surface Tension
-- concentration
--   LB/STB              LB/IN
      0.0                100.0
      0.1                50.0
      0.2                10.0
      0.5                0.0  /
      0.0                100.0
      0.1                50.0
      0.2                10.0
      0.5                0.0  /
-- Capillary de-saturation curve
SURFCAPD
-- LOG10(Capillary Number)     Miscibility
--                             Function
        -9                       0.0
         2                       0.0
         5                       1.0
        10                       1.0 /
       -10                       0.0
         2                       0.0
         5                       1.0
        10                       1.0 /
```

```
-- Extra rock properties.
SURFROCK
-- De-adsorption model            Rock mass
-- indicator                      Density
--   1 - reversible
--   2 - irreversible              LB/RB
      1                            1000 /
      1                            1000 /
RPTPROPS
 'SURFVISC'  /
```

REGIONS

```
REGIONS    ==========================================================
==
FIPNUM
 1 2 5 6 6
 3 4 5 6 6
 5 5 5 6 6
 6 6 6 6 6
 6 6 6 6 6  /

-- Immiscible table numbers
SATNUM
 25*1 /
-- Miscible table numbers
SURFNUM
 25*2 /
RPTREGS
 'PVTNUM'  'SATNUM'  'EQLNUM'  'FIPNUM'  'KRNUMX'
 'KRNUMX-'  'KRNUMY'  'KRNUMY-'  'KRNUMZ'  'KRNUMZ-'
 'IMBNUM'  'IMBNUMX'  'IMBNUMX-'  'IMBNUMY'  'IMBNUMY-'
 'IMBNUMZ'  'IMBNUMZ-'  'ROCKNUM'  'ENDNUM'  'TNUM'
 'MISCNUM'  'KRNUMMF'  'IMBNUMMF'  'SURFNUM'  /
```

SOLUTION

```
SOLUTION  ==========================================================
TBLKFONE
  25*0 /
EQUIL
4000  4000  6000  0   0   0   0   0   0  /
RPTSOL
'PRES'  'SOIL'  'SWAT'  'RESTART'  'OILAPI'
'FIPTR=2'  'TBLK'  'FIPPLY'   'SURFBLK'
'FIPSURF=2'  /
```

SUMMARY

```
SUMMARY    ==========================================================
==
CTFRSUR
 'P' 5 5  1 /
/
FTPRSUR
WTPRSUR
'P' /
FTPTSUR
```

```
WTPTSUR
'P' /
FTITSUR
WTITSUR
'I' /
BTCNFSUR
1 1 1 /
2 2 1 /
5 5 1 /
/
FTIPTSUR
FTIPFSUR
RTIPTSUR
1 /
RTIPFSUR
1 /
BTIPTSUR
1 1 1 /
/
FTADSUR
RTADSUR
 1 /
BTADSUR
1 1 1 /
/
BTCASUR
1 1 1 /
/
```

SCHEDULE

```
SCHEDULE  ==========================================================
RPTSCHED
'PRES'  'SOIL'  'SWAT'  'RESTART=2'  'FIP=2'
'WELLS=2'  'SUMMARY=2'  'CPU=2'  'NEWTON=2'
'OILAPI'  'FIPTR=2'  'TBLK'  'FIPSALT=2'
'SURFBLK'  'SURFADS'  'FIPSURF=2'  /
WELSPECS
'I'  'G'   1  1  4000  'WAT'  /
'P'  'G'   5  5  4000  'OIL'  /
/
COMPDAT
'I       '   1   1   1   1 'OPEN'   0  .0   1.0 /
'P       '   5   5   1   1 'OPEN'   0  .0   1.0 /
/
WCONPROD
'P' 'OPEN' 'BHP' 5* 3999.0 /
/
```

```
WCONINJ
'I' 'WAT' 'OPEN' 'RATE' 200.0 /
/
WTRACER
 'I' 'ONE'  0.8 /
/
-- Start injecting the Surfactant Slug
WSURFACT
 'I'   0.8 /
 /
TUNING
 1  /
 10* 0.4 /
  /
TSTEP
  2*100
/
WTRACER
 'I' 'ONE'  0.0 /
/
-- Water chase
WSURFACT
 'I'   0.0 /
 /
TSTEP
 5*100 /
END
```

Temperature option

Chapter 67

Introduction

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

The Temperature option enables ECLIPSE to model temperature effects, for example the injection of cold water into a hot reservoir.

The major effect of temperature changes in the vicinity of the injection wells is to modify the fluid viscosities. In addition, changes in the reservoir temperature will induce additional stresses within the reservoir which may modify the rock properties. ECLIPSE can output the temperature distribution to the `RESTART` file; this may be used by a separate program to model the effect of the temperature change on the reservoir rock.

The Temperature option is initiated by the keyword `TEMP` in the `RUNSPEC` section.

Particularly for dual porosity calculations, the user should refer to the documentation on the thermal option in "Dual porosity" on page 1077 for information on how the rock volume is handled in the presence of more than one porosity.

Differences between the Temperature option in ECLIPSE 100 and ECLIPSE 300

The Temperature option is handled differently in the two simulators. In ECLIPSE 100 temperature is treated like a tracer, while in ECLIPSE 300 the `TEMP` keyword invokes the Thermal option, see "Thermal Option" on page 1023. This leads to a number of important differences:

- In ECLIPSE 100 an energy conservation equation is solved at the end of each converged time step, and the grid block temperatures are updated. The new temperatures are then used to calculate the oil and water viscosities for the subsequent time step. In ECLIPSE 300 the energy conservation equation is solved simultaneously with the flow equations. In both cases, the rock and all fluids in a grid block are assumed to be at the same temperature.
- In ECLIPSE 300 water is allowed in the vapor phase.
- In ECLIPSE 300 the equilibrium state depends on the temperature.

- In ECLIPSE 300 the reservoir temperature changes if the reservoir pressure changes. Similarly the injection temperature changes as the well BHP changes.
- The ECLIPSE 300 thermal option is designed for modeling steam injection - it should not be used for pressures or temperatures above the critical point of water (221.2 bar / 3208 psi, 374.15^{o}C / 705.5^{o}F)

These differences mean that there will be a difference in performance between the two simulators, and there may be some difference in the results.

Differences between the Temperature option in ECLIPSE 300 and the Thermal option

The Thermal option can be used to run Temperature option data sets by replacing the `TEMP` keyword with the `THERMAL` and `BLACKOIL` keywords. This uses exactly the same code as the ECLIPSE 300 Temperature option, but allows `THERMAL` keywords to be used as well. For example, heat loss, stream injection and heater keywords can all be added to the data set.

Thermal properties

The minimum thermal properties required by temperature option are the specific heat capacity of the rock and the fluids present in the reservoir. The rock specific heat is entered as a volume specific heat, tabulated against temperature, using the `SPECROCK` keyword. The fluid specific heats are mass specific heats, again tabulated against temperature, entered using the `SPECHEAT` keyword.

Viscosity modification

Both the oil and water viscosities can optionally be specified as functions of temperature using the `OILVISCT` and `WATVISCT` keywords. The viscosities are supplied at the reference conditions specified using the `VISCREF` keyword. The oil viscosity at the prevailing temperature, pressure and gas/oil ratio is calculated as follows:

$$\mu_o(T, P, R_s) = \mu_T(T) \cdot \frac{\mu_p(P, R_s)}{\mu_p(P_{ref}, R_{s\,ref})} \qquad \text{[EQ 67.1]}$$

where

μ_T is the viscosity from the `OILVISCT` keyword

μ_p is the viscosity from the `PVTO` (or `PVDO`) keyword

P_{ref} is the reference pressure defined by the `VISCREF` keyword

$R_{s\,ref}$ is the reference gas/oil ratio defined by the `VISCREF` keyword.

Similarly, the water viscosity at the prevailing temperature, pressure and optionally salt concentration is calculated as follows:

$$\mu_w(T, P) = \mu_T(T) \cdot \frac{\mu_p(P)}{\mu_p(P_{ref})} \qquad \text{[EQ 67.2]}$$

or, if the Brine option is active, as

$$\mu_w(T, P, c_s) = \mu_T(T) \cdot \frac{\mu_p(P, c_s)}{\mu_p(P_{ref}, c_{s\ ref})} \qquad \text{[EQ 67.3]}$$

where

μ_T is the viscosity from the WATVISCT keyword

μ_p is the viscosity from the PVTW or PVTWSALT keyword

P_{ref} is the reference pressure defined by the VISCREF keyword

$c_{s\ ref}$ is the reference salt concentration defined by item 2, record 1 of the PVTWSALT keyword.

Thermal conduction

The thermal conduction of heat in the rock is often a small effect compared with the convection of heat with the injected water. Thermal conduction is optional and is initiated by using either the THCONR keyword, or for ECLIPSE 300 only, keywords THCROCK, THCOIL, THCWATER, THCGAS and THCSOLID, in the grid section, to specify the cell rock and fluid thermal conductivity.

ECLIPSE 100

The thermal conductivity and thermal transmissibility can be printed out using the THCONR and TRTHERM mnemonics in the RPTGRID keyword. The thermal transmissibilities calculated by ECLIPSE can be modified using inter-region thermal transmissibility multipliers specified using keywords MULTREGT and MULTNUM.

ECLIPSE 300

The average cell thermal conductivity can be printed out using the mnemonic THCAVE in the RPTGRID keyword. The thermal transmissibility can be printed out using the HEATTX, HEATTY, and HEATTZ mnemonics in the RPTGRID keyword. The thermal transmissibilities calculated by ECLIPSE can be modified using keywords HEATTX, HEATTY and HEATTZ.

Heat losses/gains

No account is taken of heat loss to and from areas outside the reservoir model. If these heat losses are likely to be significant, the reservoir model must be extended to include large blocks to act a heat sinks. These additional blocks have to be active cells (pore volume > zero), but the permeability may be set to zero.

Equilibrium state

ECLIPSE 300

In ECLIPSE 300 a thermal flash is performed to determine the state. K-values are calculated from R_s and R_v values as functions of pressure and temperature:

$$K_{oil} = \frac{R_s + F}{1 + F \cdot R_v} \cdot R_v \cdot e^{D_{oil} \cdot (1/T_0 - 1/T)}$$

$$K_{gas} = \frac{R_s + F}{1 + F \cdot R_v} \cdot \frac{1}{R_s} \cdot e^{D_{gas} \cdot (1/T_0 - 1/T)}$$

The temperature T_0 is taken to be the initial reservoir temperature RTEMPA and the coefficients for the temperature dependence D_{oil} and D_{gas} are taken from [EQ 69.42] with default values of the acentric factors and critical temperatures given in Table 67.1.

The constant F is the ratio of the molar volumes of gas to oil at surface conditions:

$$F = \frac{V_{gas,s}}{V_{oil,s}} = \frac{MW^{gas}}{MW^{oil}} \cdot \frac{\rho_{oil,s}}{\rho_{gas,s}}$$

Values of $\rho_{oil,s}$ and $\rho_{gas,s}$ are taken from the DENSITY keyword, while default values for the molecular weights are given in Table 67.1.

Table 67.1 Default values used in calculating K-values

Property	Symbol	Units	Oil	Gas
Acentric factor	Ac		0.348	0.056
Critical temperature	Tc	K	603	200
Molecular weight	MW	kg/kgmol	120	25

Setting the initial conditions

The initial temperature is either assumed to be constant throughout the reservoir or can be specified as function of depth. If the initial temperature distribution is specified as a function of depth then the reservoir will not be in thermal equilibrium as there are no heat sources/sinks at the top and bottom of the model. The initial temperature is set using either the RTEMPA or RTEMPVD keywords in the SOLUTION section.

Wells

ECLIPSE 100 In ECLIPSE 100 the temperature of the injected fluid can be specified for each well in the SCHEDULE section, using the WTEMP keyword. If a well does not have an injection temperature specified, the fluid is taken to be at zero degrees.

ECLIPSE 300 In ECLIPSE 300 the enthalpy of the injected water must be specified. This can be done by specifying the temperature and pressure of the injected water for each well in the SCHEDULE section, using the WINJTEMP keyword.

Solution output

ECLIPSE 100 In ECLIPSE 100 the temperature can be output using the TEMP mnemonic in RPTSCHED, and the field and region energy balance reports can be output by using the FIPTEMP mnemonic.

ECLIPSE 300 In ECLIPSE 300 temperature data can be output using Thermal option keywords, see "Thermal keywords" on page 1025.

Summary output

ECLIPSE 100

Information about the temperatures in the reservoir can be output to the `SUMMARY` file in an analogous manner to the tracer output. For the heat equation the 3-character mnemonic that should be appended to summary quantities is ‘HEA’.

Table 67.2 Controlling summary output for temperature option

Field	Group	Well	Description
FTPCHEA	GTPCHEA	WTPCHEA	Production Temperature
FTICHEA	GTICHEA	WTICHEA	Injection Temperature
FTPRHEA	GTPRHEA	WTPRHEA	Heat flows (Production)
FTPTHEA	GTPTHEA	WTPTHEA	Heat Production Total
FTIRHEA	GTIRHEA	WTIRHEA	Heat flows (Injection)
FTITHEA	GTITHEA	WTITHEA	Heat Injection Total

Table 67.3 Controlling summary output for temperature option

Field	Region	Block	Description
		BTCNFHEA	Block Temperature
FTIPTHEA	RTIPTHEA	BTIPTHEA	Heat in place

Note that the heat in place is quoted relative to the initial conditions.

ECLIPSE 300

In ECLIPSE 300 temperature data can be output using Thermal option keywords, see "Thermal keywords" on page 1025.

Restrictions

Changing the pore volume in the `EDIT` section using `PORV` is not allowed with the temperature option.

Example data set

The example data sets HEAT100.DATA and HEAT300.DATA which are in the directory `e300\data\thermal` can be used with ECLIPSE 100 and ECLIPSE 300 respectively.

Tensor Permeability

Chapter 68

Introduction

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

Full tensor permeabilities are often required to describe complex reservoirs. Given a detailed heterogeneous reservoir description at the geological scale, upscaling is normally carried out to reduce the model size to one suitable for reservoir simulation. These upscaling procedures will generally create full tensor permeabilities, even if the permeabilities in the fine geological grid are not represented as tensors. Full tensor permeabilities also arise in complex crossbedded systems where the dip directions will not in general coincide with a local coordinate direction. A further case for full tensor permeabilities is in the modeling of fractured systems.

Conventionally, reservoir simulators ignore the full tensor nature of the absolute permeability and assume the permeability can be represented by a diagonal tensor whose principal axes coincide with the local coordinate axes. This allows the construction of the usual 5 point discretization in 2D or 7 point in 3D which can be solved efficiently in the simulator. Introduction of a full tensor permeability extends the stencil of the discretization scheme to 9 points in 2D and either 19 or 27 points in 3D.

ECLIPSE 300 allows a full tensor description of the absolute permeability. The permeability tensor may either be supplied by a pre-processor by upscaling the properties on the geological model, or else it may be entered by keywords `PERMXX`, `PERMYY`, `PERMZZ` (which describe the diagonal components) and `PERMXY`, `PERMYZ`, `PERMZX` to describe the off-diagonal components.

Only positive definite symmetric tensors are allowed in the simulator. The principal directions of the tensor must also be supplied. At present, the options are

1. the directions for the components of the tensor are aligned with the Cartesian axes;
2. aligned with the axes joining the midpoints of each pair of opposite faces in corner point cells; and
3. aligned with the two axes formed by joining the midpoints of pairs of opposite faces in the I and J directions and with the third axis normal to the bedding plane.

The frame representing these directions is specified either by the `TBASIS` keyword or by the `MPFA` keyword.

The permeability tensor can be expressed as

$$\underline{\boldsymbol{K}} = \begin{bmatrix} k_{xx} & k_{xy} & k_{xz} \\ k_{yx} & k_{yy} & k_{yz} \\ k_{zx} & k_{zy} & k_{zz} \end{bmatrix} \qquad \text{[EQ 68.1]}$$

where by symmetry

$$k_{yx} = k_{xy}\,,\; k_{zy} = k_{yz}\,,\; k_{zx} = k_{xz}\,. \qquad \text{[EQ 68.2]}$$

In ECLIPSE, the coefficients are entered as

k_{xx}=PERMXX, k_{yy}=PERMYY, k_{zz}=PERMZZ, k_{xy}=PERMXY, k_{yz} =PERMYZ and k_{zx}=PERMZX.

The flow equations arising from the permeability tensor description are constructed using a Multipoint Flux Approximation (MPFA). The transmissibilities are calculated by the control volume discretization due to Aavatsmark et al. - see [Ref. 76] for details. The MPFA can only be used in corner point geometry, not block centered geometry.

The discretization scheme is selected by the MPFA keyword. The MPFA is a rigorous treatment of both cell non-orthogonality and tensor permeability. Even for a diagonal isotropic tensor, the MPFA will generally lead to a 9 point discretization in 2D in order to remove the bias of grid non-orthogonality and grid orientation effects.

The MPFA discretization is applied by default across the entire grid. However, it can be applied in selected regions by using the MPFANUM keyword. The MPFA is only applied where the MPFANUM array equals 1. Further control may be achieved by applying the discretization only within nominated local grids using the MPFA keyword within a CARFIN and ENDFIN pair.

The MPFA can also be used on unstructured grids generated by FloGrid. The multipoint flux non-neighbor connections are exported to ECLIPSE through the MPFNNC keyword. Further details can be found in [Ref. 75].

The use of the MPFA option is not allowed with the WARP solver. Please use the pre-2002a solver (ECLSOL) instead by setting OPTIONS3 item 66 to a non-zero value.

Discretization

Figure 68.1 Stencil for MPFA flow in 2D between cells (i, j) and (i + 1, j)

Referring to "Formulation of the Equations" on page 341, the standard two-point flow between cells A = (i, j) and B = (i + 1, j) for the corner point grid in Figure 68.1 can be written for a component c embedded in a phase p (p = o, w, g) as

$$F^{c}_{pAB} = T_{AB}\, M^{c}_{pAB}\, dP_{pAB} \qquad \text{[EQ 68.3]}$$

where

T_{AB} is the two-point transmissibility between cells A and B

M^{c}_{pAB} is the generalized mobility of component c in phase p, and is given by [EQ 23.26]

and

dP_{pAB} is the potential difference of phase p between cells A and B, given by

$$dP_{pAB} = P_A - P_B + P_{cpA} - P_{cpB} - \rho_{pAB} G (D_A - D_B) \qquad \text{[EQ 68.4]}$$

and where

P_{cp} is the capillary pressure for phase p

ρ_p is the mass density of phase p

G is the acceleration due to gravity

D is the cell center depth.

Alternatively, the MPFA constructs the flow F^{c}_{pAB} which depends on steering terms from the neighboring cells C, D, E and F in Figure 68.1 as

$$F^c{}_{pAB} = M^c{}_{pAB}\left(T_{AB}\,dP_{pAB} + \sum_S T_{AS}\,dP_{AS}\right) \qquad \text{[EQ 68.5]}$$

where the summation is over *steering* transmissibilities S between cells AC, AD, AE and SF.

The mobility M^c_{pAB} is evaluated in the upstream cell (A or B) according to the direction of flow in [EQ 68.5].

The steering potential differences dP_{AS} are evaluated using pressures and capillary pressures for cells A and S but using the density term ρ_{AB} for the primary flow coefficient obtained from [EQ 23.36], so that

$$dP_{pAS} = P_A - P_S + P_{cpA} - P_{cpS} - \rho_{pAB}G(D_A - D_S) \qquad \text{[EQ 68.6]}$$

The transmissibilities T_{AB} and T_{AS} are calculated following the procedure described in [Ref. 76] when the permeability $\underline{\boldsymbol{K}}$ is a full tensor.

Note that the flows between adjacent cells depend on steering terms from adjacent cells, but there are no flows created between cells which are diagonal neighbors as in the `NINEPOIN` scheme (see "Nine-point Schemes" on page 707).

Similarly in 3D, the flow between any two cells may depend on up to 16 steering transmissibilities and this leads to a 19 or 27 point scheme.

In the present version of ECLIPSE 300, the MPFA transmissibilities are not calculated across faults, but the two-point connections given by the `NEWTRAN` transmissibility calculation are used instead (see "Transmissibility Calculations" on page 1103).

If the `MPFA` keyword is used without supplying off diagonal tensor permeabilities `PERMXY`, `PERMYZ` and `PERMZX`, these terms are taken to be zero and the MPFA scheme is applied with the diagonal tensor $\underline{\boldsymbol{K}}$=diag (`PERMX`, `PERMY`, `PERMZ`) in the specified basis.

Orthogonality error

The MPFA or nine point discretization reduces to a standard two point flux scheme (that is, a five-point scheme in 2D or a seven-point scheme in 3D) if the grid is sufficiently orthogonal.

For example, the MPFA discretization reduces to 5- or 7-points when both

1 the grid system is orthogonal and

2 the permeability tensor $\boldsymbol{K}$ is diagonal

In general, when the grid is either non-orthogonal or $\boldsymbol{K}$ is a full tensor then a 9-point (2D) or 27 point (3D) discretization results. It is possible to control the application of the MPFA scheme based on a cell orthogonality error indicator. Essentially for grids which are nearly orthogonal it is desirable to apply the MPFA discretization only in regions of high non-orthogonality to reduce the CPU cost. In the context of a full tensor permeability, an orthogonality indicator can be defined as follows:

Figure 68.2 K-orthogonality error indicator

Consider an interface PQ between two cells 1 and 2. The permeabilities in each cells are $\boldsymbol{K}_1$ and $\boldsymbol{K}_2$. The directed area normal on the interface is denoted by $\underline{A}$. Let $\underline{a}_1$ and $\underline{a}_2$ be the vectors joining the midpoints of opposite faces of the cells. We call $\underline{a}_1$ and $\underline{a}_2$ the *covariant* basis vectors. Define the *contravariant* basis vectors $\underline{a}^1$ and $\underline{a}^2$ with the condition that

$$\underline{a}_i \bullet \underline{a}^j = \delta_{ij}$$

where δ_{ij} is the Kronecker delta.

Now consider a potential gradient in cell 1,

$$\underline{\nabla}\phi = \sum_{j=1}^{2} \alpha_j \underline{a}^j$$

The flux through PQ from cell 1 is therefore

$$f = \lambda \cdot \underline{A} \bullet \underline{K} \bullet \nabla\phi = \lambda \sum_{j=1}^{2} \alpha_j \underline{A} \bullet \underline{K} \bullet \underline{a}^j$$

where λ accounts for the mobility.

Since this expression can be used for arbitrary values of α_j, it follows that if $\underline{A} \bullet \underline{K}$ is parallel to $\underline{a}_1$, such that $\underline{A} \bullet \underline{K} \bullet \underline{a}^2 = 0$, then the flux depends only on the gradient in the covariant direction $\underline{a}_1$ and implies that a two point discretization can be employed.

The grid is said to be ***K***-*orthogonal* (see [Ref. 75] and [Ref. 76]) when $\underline{A} \bullet \underline{K}$ is parallel to $\underline{a}_1$.

A convenient measure of orthogonality error for an interface is therefore

$$\varepsilon = \left\| \underline{A} \bullet \underline{K} - \frac{[(\underline{A} \bullet \underline{K}) \bullet \underline{a}_1]\underline{a}_1}{\|\underline{a}_1 \bullet \underline{a}_1\|} \right\| / \|\underline{A} \bullet \underline{K}\|$$

This is evaluated separately for each of the two cells adjoining the interface $\underline{A}$. The maximum of these two values provides a measure of the departure from orthogonality for the interface, that is, the departure from a two point flux approximation for this interface.

Keyword `ORTHERRO` may be used to specify two point fluxes for all interfaces with $\varepsilon < ORTHERRO$. The familiar 5-point (2D) or 7-point (3D) scheme then applies in regions when all interfaces have negligible orthogonality error. Alternatively using `ORTHERRX`, `ORTHERRY` and `ORTHERRZ` controls calculation of MPF fluxes for interfaces in the grid I, J and K directions.

Application of MPFA can also be controlled by a minimum pore volume limit using the `MINPORV9` keyword, so that cells with $PORV < MINPORV9$ use the familiar 5- point or 9-point discretization. The 5- or 9-point discretization (using `NEWTRAN` transmissibilities) is also used in regions for which `MPFANUM` is set equal to 0.

The keywords `ORTHERRO`, `ORTHERRX`, `ORTHERRZ` and `MINPORV9` can be individually applied to local grids if they are specified between the corresponding `CARFIN` and `ENDFIN` pairs.

Restrictions

The MPFA discretizations cannot at present be used with:

1. The parallel option (keyword `PARALLEL`)
2. Velocity dependent relative permeabilities (keyword `VELDEP`)
3. Cell coarsening (keyword `COARSEN`)
4. The Diffusion option (keyword `DIFFUSE`)
5. The Thermal option (keyword `THERMAL`)
6. Radial geometry (keyword `RADIAL`)
7. The WARP solver (keyword `WARP`)

Thermal Option

Chapter 69

Introduction

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL

Thermal recovery methods are typically used in heavy oil reservoirs, where the oil viscosity is high at reservoir temperatures, but reduces as the temperature increases. A number of thermal recovery processes can be simulated in ECLIPSE. These include:

- steam injection, such as cyclic steam injection (huff and puff), steam flood, or steam assisted gravity drainage (SAGD)
- hot fluid or gas injection
- well bore heaters (see "Heaters" on page 1075)
- combustion (see "Combustion" on page 1068).

The simulator has been designed to handle typical operating conditions for thermal recovery. This includes pressures between 1 and ~100 atmospheres (1 - ~100 bar), and temperatures from ambient to ~700 degrees Fahrenheit (~370 degrees C).

Four phases are allowed in the simulation:

- an oil phase that contains only hydrocarbon components,
- a gas phase that can contain both hydrocarbon and water components, and
- a water phase that contains only a water component,
- a solid phase (for use with chemical reactions, "Reactions involving solid components" on page 151)

'Foamy oil' can be modeled using the thermal option. Although foamy oils occur at reservoir temperatures, the phenomenon is associated with heavy oils, so is included with the thermal option, see "Foamy oil" on page 1068.

The thermal option can be used for dual porosity grids, but the combination of the thermal option and dual porosity makes extra assumptions about the distribution of the rock volume between the porosities: see "Dual porosity" on page 1077.

The thermal option can be run in one of three modes:

- live oil, using K-values to define equilibrium,

- dead oil, when the hydrocarbon components are non-volatile,
- black oil, using Temperature option keywords, see "Temperature option" on page 1009.

The next section contains a list of keywords that can only be used with the Thermal live oil and dead oil options, and those ECLIPSE keywords that have special meaning when used with the Thermal option. This is followed by sections describing the thermal properties and features that can be modeled. Example data sets are also provided.

Thermal keywords

The ECLIPSE 300 Thermal option can be selected by entering the THERMAL keyword in the RUNSPEC section of the DATA file. Fluid properties can be specified in three different ways:

LIVEOIL allows the simulation of multiple hydrocarbon components in both oil and gas phases.

DEADOIL the hydrocarbon component cannot appear in the gas phase. A simpler set of property keywords is used. More than one hydrocarbon component is not allowed.

BLACKOIL properties are specified using the Temperature option keywords, see the "Temperature option" on page 1009.

Thermal keywords can be used in the SUMMARY section and the SCHEDULE section. This option is very similar to the ECLIPSE 300 Temperature option (which is selected by using the TEMP keyword instead of the THERMAL keyword). However extra features are allowed with the blackoil thermal option (for example, heat loss, heaters and steam injection)

The following list contains those keywords that can only be used with the Thermal option, as well as general ECLIPSE 300 keywords (marked*) that have extra arguments or may be used in different ways in thermal runs.

RUNSPEC section keywords

The RUNSPEC keywords are:

Table 69.1 RUNSPEC keywords

Keyword	Description
BLACKOIL	Request use of Black Oil properties
DEADOIL	Request use of Dead Oil properties
JALS	Switch on the more powerful JALS solver. Recommended for thermal simulations
HEATDIMS	Heater keyword dimensions
LIVEOIL	Request use of Live Oil properties (the default)
ROCKDIMS	Base and cap rock dimensions
TCBDIMS	Thermal cut-back table dimensions
THERMAL	Request use of Thermal option

GRID section keywords

The GRID keywords are:

Table 69.2 GRID keywords

Keyword	Description
ACTNUM*	Active grid block identification
HEATCR	Rock volumetric heat capacity
HEATCRT	Rock volumetric heat capacity T coefficient
MINROCKV	Minimum rock volume
MINRV	Minimum rock volume (alias for MINROCKV)
NNC*	Ninth argument is conductivity across the NNC
ROCKCON	Base and cap rock connections to the reservoir
ROCKPROP	Base and cap rock properties
ROCKSPLV	Non-default proportions of the rock in each porosity (for dual porosity).
SIGMATH	Matrix-fracture thermal conductivity multiplier
THCONMF	Matrix-Fracture thermal conductivity
THCGAS	Thermal conductivity for gas phase
THCOIL	Thermal conductivity for oil phase
THCONR	Thermal conductivity of rock and fluid
THCROCK	Thermal conductivity for rock
THCSOLID	Thermal conductivity for solid phase
THCWATER	Thermal conductivity for water phase
THCONSF	Thermal conductivity saturation-dependence

EDIT section keywords

The EDIT keywords are:

Table 69.3 EDIT keywords

Keyword	Description
HEATTR	Heat transmissibility overwrite
HEATTTHT	Heat transmissibility overwrite
HEATTX	Heat transmissibility overwrite
HEATTY	Heat transmissibility overwrite
HEATTZ	Heat transmissibility overwrite
ROCKV	Rock volume

PROPS section keywords

Property keywords are given in tables Table 69.4, Table 69.5 and Table 69.6. Some keywords can be used with DEADOIL, some with LIVEOIL, and some with both.

Table 69.4 Rock-fluid properties

Keyword	Liveoil	Deadoil	Description
ENKRVT	x	x	Endpoint relative permeabilities versus temperature
ENPCVT	x	x	Endpoint capillary pressures versus temperature
ENPTVT	x	x	Endpoint saturations versus temperature

Table 69.5 Hydrocarbon properties

Reservoir Keyword	Surface Keyword	Liveoil	Deadoil	Description
ACF*		x		Component critical temperature (used with KVWI)
CCTYPE*	CCTYPES*	x		Component liquid compressibility type
CREF		x		Component compressibilities
CVTYPE	CVTYPES	x		Component volatility (dead, live, gaseous)
DREF*	DREFS*	x		Component densities
DREFT*	DREFTS*	x		Component densities (alias for DREF)
GASVISCF		x		Gas viscosity function of temperature
GASVISCT		x		Gas viscosity table as a function of temperature
GREF*		x		Reference gravities for live oil components
HEATVAP		x		Component heat of vaporization
HEATVAPE		x		Heat of vaporization exponent
HEATVAPS		x		Component heat of vaporization at standard temperature
KVCR	KVCRS	x		K-value correlation parameters
KVTABTn		x		Tables of K-values
KVTEMP		x		Temperatures for K-value tables
KVWI		x		Wilson K-value correlation parameters
OILCOMPR			x	Oil compressibility, expansion coefficient
OILMW			x	Oil molecular weight
OILSPECH			x	Oil specific heat
OILVINDX		x		Oil viscosity index
OILVISCC		x		Oil viscosity-temperature correlations
OILVISCF		x		Oil viscosity function
OILVISCT		x	x	Oil viscosity table as a function of T

Table 69.5 Hydrocarbon properties

Reservoir Keyword	Surface Keyword	Liveoil	Deadoil	Description
OILVTIM		x	x	Oil viscosity temperature interpolation method
PCRIT		x		Component critical pressure (used with KVWI)
PREF	PREFS	x		Component reference pressures
PREFT*	PREFTS*	x		Component reference pressures (alias for PREF)
REACENTH		x	x	Reaction enthalpy
SPECHA		x		Oil specific heat, 1st coefficient
SPECHB		x		Oil specific heat, 2nd coefficient
SPECHG		x		Gas specific heat, 1st coefficient
SPECHH		x		Gas specific heat, 2nd coefficient
SPECHS		x		Solid specific heat, 1st coefficient
SPECHT		x		Solid specific heat, 2nd coefficient
TCRIT*		x		Component critical temperature (used with KVWI)
THANALV		x	x	Analytic oil viscosities
THERMEX1		x		Liquid thermal expansion coefficient
TREF*	TREFS*	x		Component reference temperatures
TREFT*	TREFTS*	x		Component reference temperature (alias for TREF)
VISCREF		x	x	Reference conditions for the viscosity/temperature tables
ZFACT1	ZFACT1S	x		Z-factor first coefficient
ZFACTOR	ZFACTORS	x		Z-factor zeroth coefficient

Table 69.6 Water properties

Keyword	Liveoil	Deadoil	Description
KVCRWAT	x	x	Water K-value correlation
THANALB	x	x	Analytic water and steam densities
THANALH	x	x	Analytic water and steam enthalpies
THSVC	x	x	Steam viscosity coefficients
THTABB	x	x	Tabular water and steam densities
THWVC	x	x	Water viscosity coefficients
WATDENT	x	x	Water density temperature dependence
WATVISCT	x	x	Water viscosity table as a function of T

REGIONS section keywords

The REGIONS keywords are:

Table 69.7 REGIONS keywords

Keyword	Description
THERMNUM	Thermal region numbers

SOLUTION section keywords

The SOLUTION keywords are:

Table 69.8 SOLUTION keywords

Keyword	Description
RPTRST*	Controls solution output for GRAF: see Table 69.17
RPTSOL*	Controls PRINT file output: see Table 69.17

SUMMARY section keywords

A list of SUMMARY keywords specific to the Thermal option is given below.

Table 69.9 Field, Group and Well energy rates

Field	Group	Well	Information
FERCK			Energy in Rock
FEOIL			Energy in Oil
FEGAS			Energy in Gas
FEWAT			Energy in Water
FETOT			Total Energy
FEAVE			Average Energy
FERPC			Energy Percentage in Rock
FEOPC			Energy Percentage in Oil
FEGPC			Energy Percentage in Gas
FEWPC			Energy Percentage in Water
FEPR	GEPR		Energy Production Rate
FEPT	GEPT		Energy Production Total
FEIR	GEIR		Energy Injection Rate
FEIT	GEIT		Energy Injection Total
FHLR			Heat loss rate
FHLT			Heat loss total
FHTR		WHTR	Heater energy rate (WHTR requires heater names)

Table 69.9 Field, Group and Well energy rates (Continued)

Field	Group	Well	Information
FHTT		WHTT	Heater energy total (WHTT requires heater names)
		WENE	Energy Density (specified)
		WENO	Energy Density of oil (specified)
		WENG	Energy Density of gas (specified)
		WENW	Energy Density of water (specified)
		WAEN	Energy Density (actual)
		WAEO	Energy Density of oil (actual)
		WAEG	Energy Density of gas (actual)
		WAEW	Energy Density of water (actual)
		WENR	Energy Rate
		WEOR	Energy rate for oil
		WEGR	Energy rate for gas
		WEWR	Energy rate for water
		WENT	Cumulative energy
		WERO	Ratio of oil energy to total energy
		WERG	Ratio of gas energy to total energy
		WERW	Ratio of water energy to total energy

Table 69.10 Miscellaneous quantities

Field	Group	Well	Information
		WSQU	Steam quality
		WTEMP	Temperature
FSTPR	GSTPR	WSTPR	Steam production rate
FSTPT	GSTPT	WSTPT	Steam production total
FOSRC	GOSRC		Cumulative oil steam ratio
FSORC	GSORC		Cumulative steam oil ratio

Table 69.11 Multi-segment wells

Segment	Information
SHIMR	Segment Enthalpy Import Rate
SHRMR	Segment Enthalpy Removal Rate
SHIMT	Segment Enthalpy Import Total
SHRMT	Segment Enthalpy Removal Total
SENE	Segment Energy

Table 69.11 Multi-segment wells

Segment	Information
SHFR	Segment Enthalpy Flow Rate
SSQU	Segment Steam Quality
STEM	Segment Temperature

Table 69.12 Inter-region flow quantities

Region	Information
REFR	Inter-region energy flow rates
REFT	Inter-region energy flow totals

Table 69.13 Block quantities

Block	Information
BTEMP	Temperature
BLMF	Liquid mole fractions
BWMF	Water mole fractions
BENERGY	Internal energy/bulk volume
BHOIL	Molar Enthalpy of Oil
BHGAS	Molar Enthalpy of Gas
BHSOL	Molar Enthalpy of Solid
BHWAT	Molar Enthalpy of Water
BSPENOIL	Molar Energy of Oil
BSPENGAS	Molar Energy of Gas
BSPENWAT	Molar Energy of Water
BSPENROC	Specific Internal Energy of Rock
BGMEO	Enthalpy mobility of Oil
BGMEG	Enthalpy mobility of Gas
BGMEW	Enthalpy mobility of Water
BTCMULT	Saturation dependent conductivity factor
BSFOIL	Saturation of foamy oil. Calculated as the saturation of the components in the oil phase with gas-like compressibility types.
BSFSOL	Saturation of foamy solids. Calculated as the saturation of the components in the solid phase with gas-like compressibility types.

Table 69.14 Simulator performance quantities

Mnemonic	Information
MAXDE	Maximum change in energy density for each time step
MAXDT	Maximum change in temperature for each time step

Table 69.15 Analytic aquifer quantities

Field	Aquifer	Information
FAQER	AAQER	Aquifer thermal energy influx rate
FAQET	AAQET	Cumulative aquifer thermal energy influx
	AAQTEMP	Aquifer temperature
	AAQENTH	Aquifer molar enthalpy

SCHEDULE section keywords

The SCHEDULE keywords are:

Table 69.16 SCHEDULE keywords

Keyword	Description
AIMCON*	AIM selection criteria
CVCRIT*	Convergence criteria
HEATER	Position and properties of heater connections
HEATERL	Position and properties of heater connections in LGR
REACCRIT	Reaction interpolation criteria
RPTPRINT*	Item 14 can be used to control heater output
RPTRST*	Controls solution output: see Table 69.17
RPTSCHED*	Controls print file solution output: see Table 69.17
TRANGE	Specify minimum and maximum reservoir temperature
TSCRIT*	Time-stepping control
WCONINJE*	Specify well injection control Specify mixed steam/gas injection
WCONPROD*	Specify well production targets Specify steam production rate
WECON*	Specify well economic limits Specify well temperature limit
WELLTCB	Specify thermal cut-back for wells
WELLTCBC	Well Temperature Cut-Back critical temperatures
WELLTCBT	Tables to be used for thermal cut-back

Table 69.16 SCHEDULE keywords

Keyword	Description
WELSEGS*	Defines thermal properties of the tubing/casing Defines the multi-segment well structure
WELTARG*	Change well production targets. Change steam production rate
WINJTEMP	Specify steam quality or temperature or enthalpy of injection fluid
WSEGEXSS*	Defines external source / sinks for multi-segment wells. Specify the temperature of imported gas
WSEGHEAT	Specify the segment heat transfer to a completion, another segment or to an external source / sink.

SOLUTION and SCHEDULE section output

Grid data, for output to GRAF or the PRINT file, can be output from the SOLUTION section and the SCHEDULE section. A list of mnemonics that can be used is given below.

Table 69.17 RPTRST, RPTSOL and RPTSCHED output controls

Input Keyword or Mnemonic	Output
ENERGY	Total energies/bulk volume (energy/(rock + pore volume))
FLOE	Inter-block energy flow
GMEG	Gas phase energy mobility
GMEO	Oil phase energy mobility
GMEW	Water phase energy mobility
HGAS	Gas enthalpy
HOIL	Oil enthalpy
HSOL	Solid enthalpy
HWAT	Water enthalpy
TCMULT	Saturation dependent thermal conductivity transmissibility multiplier
RATT	Ratio of actual temperature change to temperature target (RPTRST only)
SL	Liquid mole fractions
SPENGAS	Gas internal energy
SPENOIL	Oil internal energy
SPENWAT	Water internal energy
SV	Vapor mole fractions
SW	Water mole fractions
T2	Temperature at 2-3 phase transition
T3	Bubble point temperature
TEMP	Grid block temperature
TET	Temperatures at last time level
TO	Temperature at which the oil phase leaves the system

Table 69.17 RPTRST, RPTSOL and RPTSCHED output controls (Continued)

Input Keyword or Mnemonic	Output
TW	Temperature at which the water phase leaves the system
UGAS	Gas phase specific volumes
UOIL	Oil phase specific volumes
UWAT	Water phase specific volumes
V2	Vapor mole fraction at the system 2-3 phase transition

All sections

:

Table 69.18 Other keywords

Keyword	Option	Description
DEBUG3*	Controls debug output. The following items are specific to the Thermal option:	
	70	Debug from thermal
	77	Debug of heat loss to analytic rocks
	78	Debug of thermal cut-back
OPTIONS3*	Activates special program options. The following items are either specific to the Thermal option or have different meanings:	
	1	Set to -1 to obtain implicit hydrostatic heads in the well model.
	13	If < 0 the post-98A method of normalizing the solution change in the non-linear iterations is used.
	18	If = 1 switch off Appleyard Chop.
	21	If = -1 switch off attempts to stop Newton iterations oscillating between alternate states.
	22	If > 0 causes the program to use an alternative set of tables from Haywood instead of the tables from Perry and Green.
	27	If > 0 switch on temperature-based Appleyard chop.
	40	Enables screening of the oil, water and gas saturations from a GETSOL. SWAT values below 10^{-6} are set to 10^{-6}. This can resolve problems that arise in initializing the flash calculation in waterless cells.

Table 69.18 Other keywords

Keyword	Option	Description
OPTIONS3 (continued)	46	A positive value sets the number of Newton iterations within a time step in which a well can be revived after being closed because it is unable to flow given the current reservoir conditions. The default behavior for the 2001A release when not using the Thermal option is to revive such a well only during the first NUPCOL Newton iterations. When the Thermal option is used, the default is to allow 21 revivals within a time step before the well is closed. If this switch is set to a negative value, no well revivals are permitted. If however the switch is set to -999 the simulator reverts to the pre-2001A treatment, whereby the number of revivals permitted within a time step was fixed at 21.
	52	In 2001A and later releases, the horizontal thermal transmissibility is based on the full cell cross-sectional area and does not include a net-to-gross factor. If this option is set to 1, the pre-2001A calculation, which includes net-to-gross, is used.
	92	Alternative time-stepping strategy will be implemented. In thermal runs (2006.1 onward) this option is automatically selected and can be turned off by setting a positive value.
	135	If set to 1, an alternative implementation for predicting the energy density in the HEATER algorithm is activated.
	158	If set to 1, the Appleyard Chop is applied to components when they appear or disappear. This behavior is the default for simulations using the chemical reactions feature but can be disabled by setting this OPTIONS3 switch to -1.
	160	If set to 1, the pre-2008.2 thermal non-linear damping is restored.
	161	If set to 1 an alternative procedure for simulations with rock-filled cells is activated. Mathematically this is a null change but variations due to numerical rounding may occur.
	190	If set to a non-zero value, the pre-2009.1 setting of well enthalpies in thermal decks is activated. From 2009.1 onwards, the well enthalpies are set whenever an action is triggered.
	214	If set to a non-zero value, this enables the pre-2009.2 John Appleyard Linear Solver (JALS).
	217	If set to a non-zero value, the pre-2010.1 behavior for thermal heavy-oil equilibration will be restored.

Table 69.18 Other keywords

Keyword	Option	Description
OPTIONS3 (continued)	218	If set to a non-zero value, the pre-2010.1 thermal temperature limits will be restored.
	219	If set to a non-zero value, this enables the pre-2010.1 thermal rock-filled convergence method.
	220	If set to 1, this enables the thermal oil viscosity-temperature correlation in the OILVISCC keyword, which is based on $\log_{10}$.

Formulation

The equations used to describe thermal processes are similar to those used in the compositional simulation, "Formulation of the Equations" on page 341, but with three important differences: the addition of an energy variable and an energy equation; the presence of a water component in the gas phase as well as the water phase; and temperature dependence of properties.

The thermal simulator does not use an equation of state to determine the thermodynamic properties. Instead you must supply K-values to determine equilibrium and densities, viscosities and enthalpies for each component in each phase.

Primary variables

The primary solution variables, used for cells in all phase states, are

$X = (P, m_1, \ldots, m_N, m_w, e)$

where

P is the pressure,

$m_1, \ldots, m_N$ are molar densities of each hydrocarbon component,

m_w is the molar density of the water component, and

e is the bulk internal energy density.

The molar densities of the fluid components are measured as moles per unit reservoir volume, while the energy density is measured as energy per unit bulk volume. The bulk volume includes both the reservoir volume and the rock volume.

Conservation equations

The primary variables are found by solving conservation equations for each component, for energy and for volume. These equations are calculated using a fully implicit scheme.

The non-linear residual, R_{fl}, is derived from the component conservation equation for each fluid component (hydrocarbon and water) in each grid block at each time step:

$$R_{fl} = \frac{d}{dt}(V_p m_{fl}) + F_{fl} + Q_{fl} = 0 \qquad \text{[EQ 69.1]}$$

where

V_p is the pore volume,

F_{fl} is the net flow rate into neighboring grid blocks and

Q_{fl} is the net flow rate into wells during the time step.

The non-linear residual, R_e, is derived from the energy conservation equation in each grid block at each time step:

$$R_e = \frac{d}{dt}(V_b e) + F_e + C_e + Q_{HL} + Q_e = 0 \quad \text{[EQ 69.2]}$$

where

V_b is the bulk volume,

F_e is the convective enthalpy flow rate into neighboring grid blocks,

C_e is the conductive energy flow rate into neighboring grid blocks,

Q_{HL} is the conductive energy flow rate to the surrounding rocks (heat loss), and

Q_e is the net enthalpy flow rate into wells during the time step.

The non-linear residual, R_v, is derived from the conservation of volume in each grid block at each time step:

$$R_v = V_p - V_f = 0 \quad \text{[EQ 69.3]}$$

where V_f is the fluid volume.

Thermodynamic equilibrium condition

The phase properties used in the residual and jacobian equations (that is the fluid volume and the flow terms) can be calculated from the component properties once the mole fractions of each component in each phase have been determined.

The number of moles of each component (including water) can be written as

$$m_c = m_t \cdot L \cdot x^c{}_o + m_t \cdot V \cdot x^c{}_g + m_t \cdot W \cdot x^c{}_w, \quad \text{[EQ 69.4]}$$

where $m_t \cdot L \cdot x^c{}_o$, $m_t \cdot V \cdot x^c{}_g$ and $m_t \cdot W \cdot x^c{}_w$ are the number of moles in the oil, vapor and water phases. m_t is the total fluid molar density given by

$$m_t = \sum_c m_c + m_w \quad \text{[EQ 69.5]}$$

L, V and W are the oil, vapor and water mole fractions and $x^c{}_p$ are the component phase fractions.

Two simplifying conditions are imposed, that there is no water in the oil phase, and no hydrocarbon in the water phase:

$x^w{}_o = 0$,

$x^c{}_w = 0$, $c = 1,\ldots,N$

$x^w{}_w = 1$,

if $W > 0$

It is also assumed the three phases are in thermodynamic equilibrium, determined by user defined K-values:

$$x^c{}_g = K^c(P, T) \cdot x^c{}_o \quad \text{[EQ 69.6]}$$

$$x^w{}_g = K^w(P,T)\cdot x^w{}_w \qquad \text{[EQ 69.7]}$$

The K-values are functions of pressure P and temperature T only.

Using these equations the mole fractions of each component in each phase can be expressed in terms of the primary variables and the four flash variables, L, V, W, and T:

$$x^c{}_o = \frac{m_c}{m_t}\cdot\frac{1}{L+K^c\cdot V} \quad c = 1,\ldots,N \quad L>0 \qquad \text{[EQ 69.8]}$$

$$x^c{}_g = \frac{m_c}{m_t}\cdot\frac{K^c}{L+K^c\cdot V} \quad c = 1,\ldots,N \quad V>0 \qquad \text{[EQ 69.9]}$$

$$x^w{}_g = K^w \quad V>0,\ W>0 \qquad \text{[EQ 69.10]}$$

Total mole fractions

The total mole fractions, which are used in [EQ 69.8] and [EQ 69.9] are defined as

$$z_c = \frac{m_c}{m_T} \qquad z_w = \frac{m_w}{m_T}$$

[EQ 69.5] can be written in terms of the mole fractions

$$\sum_c z_c + z_w = 1 \qquad \text{[EQ 69.11]}$$

The total mole fractions differ from the hydrocarbon mole fractions:

$$\zeta_c = \frac{m_c}{m_T - m_w} = \frac{z_c}{1-z_w} \qquad \text{[EQ 69.12]}$$

Substituting this into [EQ 69.11] gives

$$\sum_c \zeta_c = 1 \qquad \text{[EQ 69.13]}$$

Thermal flash

The four flash variables can be found from the primary variables by solving the four flash equations in each cell:

$$\sum_c x^c{}_o = 1 \quad \text{if} \quad L>0, \qquad \text{[EQ 69.14]}$$

$$\sum_c x^c{}_g + x^w{}_g = 1 \quad \text{if} \quad V>0, \qquad \text{[EQ 69.15]}$$

$$L+V+W=1 \quad \text{and} \qquad \text{[EQ 69.16]}$$

$$V_b\cdot e = V_p\cdot m_t\cdot(L\cdot e_o + V\cdot e_g + W\cdot e_w) + (V_b - V_p)\cdot e_r, \qquad \text{[EQ 69.17]}$$

where e_o, e_g and e_w are the user defined molar energies of the oil, gas and water phases, and e_r is the user defined energy per unit volume of rock.

Fluid volume

The specific volume of each phase can be determined from user defined molar densities for each phase (b_o, b_g and b_w):

$$V_o = \frac{m_t L}{b_o}, \quad V_g = \frac{m_t V}{b_g}, \quad V_w = \frac{m_t W}{b_w} \qquad \text{[EQ 69.18]}$$

The volume balance residual [EQ 69.3] can then be written as:

$$R_v = V_p - V_p \cdot (V_o + V_g + V_w) = 0 \qquad \text{[EQ 69.19]}$$

The phase saturations S_o, S_g and S_w are defined as

$$S_o = \frac{V_o}{V_o + V_g + V_w}, \quad S_g = \frac{V_g}{V_o + V_g + V_w}, \quad S_w = \frac{V_w}{V_o + V_g + V_w} \qquad \text{[EQ 69.20]}$$

When equation [EQ 69.19] holds, the phase saturations are equal to the phase specific volumes. Therefore

$$S_o = \frac{m_t L}{b_o} \quad S_g = \frac{m_t V}{b_g} \quad S_w = \frac{m_t W}{b_w} \qquad \text{[EQ 69.21]}$$

The number of moles of each component [EQ 69.4] and the energy density [EQ 69.17] can now be written in terms of the saturations:

$$m_c = S_o \cdot b_o \cdot x^c{}_o + S_g \cdot b_g \cdot x^c{}_g + S_w \cdot b_w \cdot x^c{}_w \qquad \text{[EQ 69.22]}$$

$$\begin{aligned} &V_b \cdot e \\ &= V_p \cdot (S_o \cdot b_o \cdot e_o + S_g \cdot b_g \cdot e_g + S_w \cdot b_w \cdot e_w) + (V_b - V_p) \cdot e_r \end{aligned} \qquad \text{[EQ 69.23]}$$

These two equations can be used for initializing the simulation - when the primary solution variables need to be calculated from user defined saturations.

Inter-block flows

The net flow rate of a hydrocarbon or water component c from cell i into neighboring cells is obtained by summing the component flow over all phases p and over the neighboring cells n,

$$F_{ci} = \sum_n \Upsilon_{ni} \sum_p M_p^c \cdot dP_{pni} \qquad \text{[EQ 69.24]}$$

The net flow rate of energy from cell i into neighboring cells is obtained in a similar manner:

$$F_{ei} = \sum_n \Upsilon_{ni} \sum_p M_p^e \cdot dP_{pni} \qquad \text{[EQ 69.25]}$$

In these formulae

Υ_{ni} is the transmissibility between cells n and i,

M_p^c is the generalized mobility of component c in phase p,

$M^e{}_p$ is the generalized mobility of enthalpy in phase p and

dP_{pni} is the potential difference of phase p between cells n and i.

The generalized mobilities are given by

$$M_p^c = x_p^c \cdot k_{rp}(S_p) \cdot \frac{b_p}{\mu_p} \text{ and} \qquad \text{[EQ 69.26]}$$

$$M_p^e = H_p \cdot k_{rp}(S_p) \cdot \frac{b_p}{\mu_p} \qquad \text{[EQ 69.27]}$$

where

x_p^c is the mole fraction of component c in phase p,

H_p is the molar enthalpy in phase p.

k_{rp} is the relative permeability of phase p,

S_p is the saturation of phase p,

b_p is the molar density of phase p, and

μ_p is the viscosity of phase p.

The fluid mobilities are evaluated in the upstream cell for each phase p separately (cell n if dP_{pni} is positive, cell i if dP_{pni} is negative). The upstream calculation applies separately for each phase (oil, water, gas) so that, for example, oil may flow from cell i to cell n while water flows from cell n to cell i.

The potential difference is made up of a pressure term, a capillary pressure term and a hydrostatic term,

$$dP_{pni} = P_n - P_i + P_{cpn} - P_{cpi} - \rho_{pni} G(D_n - D_i) \qquad \text{[EQ 69.28]}$$

where

P_{cp} is the capillary pressure for phase p,

ρ_p is the mass density of phase p,

G is the acceleration due to gravity, and

D is the cell center depth.

Well flows

The rate of flow of component c into a production well w from cell i is obtained by summing the component flow over all phases p:

$$Q_{cwi} = -\Upsilon_{wi} \cdot \sum_p (P_{pi} - H_{iw} - P_{bh}) \cdot M_{pi}^c \qquad \text{[EQ 69.29]}$$

Similarly the rate of flow of energy into a production well w from cell i is

$$Q_{ewi} = -\Upsilon_{wi} \cdot \sum_p (P_{pi} - H_{iw} - P_{bh}) \cdot M_{pi}^e \qquad \text{[EQ 69.30]}$$

Here

Υ_{wi} is the well connection transmissibility factor,

H is the hydrostatic head correction,

P_{bh} is the bottom hole pressure,

M_{pi}^c is the generalized mobility of component c in phase p and

M_{pi}^e is the enthalpy mobility in phase p.

See "Well Inflow Performance" on page 1187 for further information.

Heat conduction

The heat conduction term for each cell i is given by summing conduction between all neighboring cells n:

$$C_{ei} = -\sum_n \Psi_{ni} \cdot \Lambda \cdot (T_i - T_n) \qquad \text{[EQ 69.31]}$$

Ψ_{ni} is the conduction transmissibility between cells n and i

Λ is the upstream saturation dependent conductivity multiplier.

Solid Phase

Solid components can be used in thermal simulations. A solid component can not also appear in the liquid or gas phase, but it can be created by chemical reactions, see "Chemical Reactions" on page 147. If solid components are modeled, then a number of modifications are required to the formulation described above.

The volume balance residual [EQ 69.3] is modified to include the solid volume V_s:

$$R_v = V_p - V_f - V_s = 0$$

The total fluid molar density m_t is still calculated by [EQ 69.5], but the sum is taken over the fluid components only.

The energy density equation, [EQ 69.17], is modified to include the solid energy e_s:

$$V_b \cdot e = V_p \cdot m_t \cdot (L \cdot e_o + V \cdot e_g + W \cdot e_w) + V_p \cdot e_s + (V_b - V_p) \cdot e_r$$

Equation [EQ 69.19], which is derived from the volume balance equation, is replaced by

$$R_v = V_p - V_p \cdot (V_o + V_g + V_w + V_{sol}) = 0$$

where V_{sol} is the specific solid volume.

The equations for the saturations [EQ 69.20] are modified to include the solid volume:

$$\hat{S}_o = \frac{V_o}{V_o + V_g + V_w + V_{sol}} = \frac{V_f}{V_f + V_{sol}} \cdot S_o$$

$$\hat{S}_g = \frac{V_g}{V_o + V_g + V_w + V_{sol}} = \frac{V_f}{V_f + V_{sol}} \cdot S_g \; \hat{S}_w = \frac{V_w}{V_o + V_g + V_w + V_{sol}} = \frac{V_f}{V_f + V_{sol}} \cdot S_w$$

$$\hat{S}_{sol} = \frac{V_{sol}}{V_o + V_g + V_w + V_{sol}}$$

The modified saturations satisfy

$$\hat{S}_o + \hat{S}_g + \hat{S}_w + \hat{S}_{sol} = 1$$

[EQ 69.20] is still used to define normalized saturations S_o, S_g and S_w, representing the fraction of the fluid volume that is in each fluid phase. The normalized saturations satisfy

$$S_o + S_g + S_w = 1$$

The normalized saturations are used in calculating the relative permeabilities in [EQ 69.26] and [EQ 69.27].

Solution method

For each cell there are $N+3$ variables (pressure, N hydrocarbon components, one water component, and energy) and $N+3$ equations ($N+1$ component conservation equations, including water [EQ 69.1], energy conservation [EQ 69.2] and volume balance [EQ 69.3]). The equations can be solved using a fully implicit method (`FULLIMP` / default), an implicit pressure, explicit properties method (`IMPES`) or a combination of both, (`AIM`).

The component conservation equations are written as

$$R_{fl} = \frac{V_p(P_{t+\Delta t}) \cdot m_{fl}(t+\Delta t) - V(P_t) \cdot m_{fl}(t)}{\Delta t} + F_{fl} + Q_{fl} = 0$$

where, in an implicit cell the flow terms and well terms are evaluated at the new time:

$$F_{fl} = F_{fl}(P_{t+\Delta t}, T_{t+\Delta t}, S_{t+\Delta t})$$

$$Q_{fl} = Q_{fl}(P_{t+\Delta t}, T_{t+\Delta t}, S_{t+\Delta t})$$

and in the IMPES method the flow terms and well terms are evaluated using the new pressure but with properties at the previous time step:

$$F_{fl} = F_{fl}(P_{t+\Delta t}, T_t, S_t)$$

$$Q_{fl} = Q_{fl}(P_{t+\Delta t}, T_t, S_t)$$

The energy conservation equation is dealt with in a similar manner, but the volume balance equation is always evaluated at the new time step.

The non-linear residual equations

$$R(X) = 0$$

are solved iteratively,

$$X_n = X_{n-1} + \Delta X_{n-1} \qquad \text{[EQ 69.32]}$$

The solution increment ΔX_{n-1} is found by solving the linear matrix equation

$$0 = R(X_{n-1}) + J \cdot \Delta X_{n-1}$$

where J is the jacobian matrix:

$$J = \left(\frac{\partial R}{\partial X}\right)\Bigg|_{X = X_{n-1}}$$

The diagonal jacobian element corresponding to an IMPES cell has a simple form which can be solved relatively easily:

$$J = \begin{array}{l} \\ \text{Comp1} \\ \text{Comp2} \\ \dots \\ \text{Energy} \\ \text{Volume bal} \end{array} \begin{array}{c} \begin{array}{ccccc} P & m_1 & m_2 & \dots & e \end{array} \\ \begin{bmatrix} x & x & 0 & \dots & 0 \\ x & 0 & x & \dots & 0 \\ \dots & \dots & \dots & \dots & \dots \\ x & 0 & 0 & \dots & x \\ x & x & x & \dots & x \end{bmatrix} \end{array}$$

Linear Solvers

To compute the solution increment ΔX the linear system involving the Jacobian matrix must be inverted. Whilst the default WARP linear solver may be adequate for relatively simple simulations better performance is usually achieved with the JALS solver. The JALS solver can be activated using the `JALS` keyword in the `RUNSPEC` section.

Thermal properties

Rocks

Normally in ECLIPSE, a cell is inactive if its porosity is zero. However, in the thermal option, rock filled cells are considered as active cells since they conduct heat. Therefore, a cell is inactive if both its pore volume and rock volume are zero, or, if the ACTNUM keyword has been used to make it inactive.

Rock heat capacity

The enthalpy per unit volume of rock is given by

$$H_r = C_{r0}(T - T_{\text{ref}}) + \frac{C_{r1}(T - T_{\text{ref}})^2}{2} \qquad \text{[EQ 69.33]}$$

where the volumetric heat capacity C_{r0} is defined by the HEATCR keyword, the temperature coefficient C_{r1} is defined by the HEATCRT keyword and the reference temperature T_{ref} is specified with the STCOND keyword.

Thermal conductivity

There are two ways to define the thermal conductivity:

1. Using keyword THCONR to define the thermal conductivity of a rock and liquid filled cell, and optionally THCONSF.
2. Using keyword THCROCK, and depending on the phases present in the model, the phase thermal conductivities, THCOIL, THCWATER, THCGAS and THCSOLID.

For both options, we define the cell average thermal conductivity, THCAVE, which is used to determine the thermal transmissibilities.

The keywords from these two separate options **must** not be mixed together in the dataset.

Option 1: using THCONR

The THCONR keyword defines the thermal conductivity of a rock and liquid filled cell.

It can be calculated from the rock, oil, gas and water conductivities (K_r, K_o, K_g, K_w) by first determining an average liquid conductivity

$$K_l = (K_o + K_w)/2$$

The conductivity for a liquid filled cell can then be calculated as a porosity weighted average of the liquid and rock conductivities:

$$\text{THCAVE} = \text{THCONR} = \theta \cdot K_l + (1 - \theta) \cdot K_r$$

THCAVE is used to calculate the thermal transmissibilities in the reservoir.

Option 2: using THCROCK

Depending on the phases in the model, the average cell thermal conductivity (THCAVE) is defined as a porosity weighted average of the phase and rock conductivities using the expression below:

$$\text{THCAVE} = \phi\frac{(\text{THCOIL} + \text{THCGAS} + \text{THCWATER} + \text{THCSOLID})}{\text{NPHAS}} + (1-\phi)\text{THCROCK}$$

In this expression, ϕ is the grid porosity; NPHAS denotes the number of phases in the model; THCOIL, THCGAS, THCWATER and THCSOLID specify the phase thermal conductivities; and THCROCK specifies the rock thermal conductivity.

If some phases are not present in the model, then the relevant phase thermal conductivity will be ignored.

THCAVE is used to calculate the thermal transmissibilities in the reservoir.

Thermal transmissibility

Thermal transmissibilities are calculated in a similar manner to the flow transmissibilities, with the thermal conductivity being used instead of the permeabilities. For example, using corner point geometry, the thermal transmissibility between cells i and j is given by $H^{i\to j}$ where

$$1/H^{i\to j} = 1/H_i + 1/H_j \qquad \text{[EQ 69.34]}$$

$$H_i = THCAVE_i \cdot \frac{A \cdot D_i}{D_i \cdot D_i} \qquad \text{[EQ 69.35]}$$

$$A \cdot D_i = A_X \cdot D_{iX} + A_Y \cdot D_{iY} + A_Z \cdot D_{iZ} \qquad \text{[EQ 69.36]}$$

and

$$D_i \cdot D_i = D_{iX}^2 + D_{iY}^2 + D_{iZ}^2 . \qquad \text{[EQ 69.37]}$$

Here A_X, A_Y and A_Z are the X-,Y- and Z- projections of the mutual interface area of cell *i* and cell *j*, and D_{iX}, D_{iY} and D_{iZ} are the X-, Y- and Z- components of the distance between the center of cell *i* and the center of the relevant face of cell *i*. See "Transmissibility Calculations" on page 1103 for further details.

Thermal transmissibilities can also be entered directly, using the keywords HEATTX, HEATTY, HEATTZ (or HEATTR, HEATTTHT, HEATTZ in radial coordinates).

Definition of multiplier for Heat Conduction

Option 1: using THCONR

Thermal transmissibilities can be multiplied by a saturation dependent factor Λ. Following Coats [Ref. 54], it is assumed that the thermal transmissibility is a function of the gas saturation, S_g, only. This method does not distinguish between the thermal transmissibility of a rock filled with oil and the thermal transmissibility of a rock filled with water.

The multiplier Λ is defined to be:

$$\Lambda = (1 - \alpha S_g) \qquad \text{[EQ 69.38]}$$

where α is specified by the THCONSF keyword.

From the definition of THCONR above, a value for α can be determined as:

$$\alpha = \theta \cdot (K_l - K_g)/\text{THCONR}$$

To check these values, we can calculate the conductivities for a liquid filled cell $S_g = 0$ and a gas filled cell $S_g = 1$:

$$\text{THCONR} \cdot \Lambda(0) = \theta \cdot K_l + (1-\theta) \cdot K_r$$

$$\mathrm{THCONR} \cdot \Lambda(1) = \theta \cdot K_g + (1-\theta) \cdot K_r$$

which are just the porosity weighted averages of the rock and fluid conductivities

Option 2: using THCROCK

Thermal transmissibilities can be multiplied by a saturation dependent factor Λ. For this option, Λ takes the form:

$$\Lambda = (\phi(\Sigma_p S_p K_p) + (1-\phi)K_{rock})/\mathrm{THCAVE}$$

In the above expression, ϕ is the porosity; THCOIL is denoted by K_{oil}, THCGAS is denoted by K_{gas}, THCWATER is denoted by K_{water}, THCSOLID is denoted by K_{solid} and THCROCK, specifying the rock thermal conductivity, is denoted by K_{rock}. The phase saturations are represented by S_p.

Dual Porosity or Dual Permeability

A matrix-fracture thermal conductivity can be set using the THCONMF keyword. The SIGMATH keyword can be used to set the multiplier used in calculating the matrix-fracture thermal transmissibilities.

Hysteretic rock compaction

During high pressure cyclic steam injection (huff and puff), the rock in the reservoir may be stressed beyond its elastic limits. A simple hysteretic model of rock compaction, the BOBERG option of keyword ROCKCOMP, designed specifically for these circumstances, can be used: see "Rock Compaction" on page 859.

Relative permeability curves

Keywords ENPTVT, ENKRVT, and ENPCVT specify the temperature-dependence of relative permeability curves. The form of these keywords is similar to the standard ECLIPSE 100/300 end-point-scaling keywords ENPTVD, ENKRVD, and ENPCVD. In the thermal option, end-point-scaling keywords cannot be used in the same data set as temperature-dependent relative permeability keywords.

Live oil

For a live oil run you must specify temperature dependent properties for each component. The properties required for live components are, K-values to determine equilibrium between the oil and gas phase, and the phase enthalpies, densities and viscosities. These properties are output to the print file in tabular form. The program PVTi can be used to output K-values, enthalpy and density data.

Volatility type

In a live oil run, not all the components need be live. The CVTYPE keyword can be used to define a component as non-volatile, non-condensable or solid.

Non-volatile and non-condensable components

A component can be made non-volatile or non-condensable using the CVTYPE keyword. This keyword also resets the component critical temperature, to 10^4 for non-volatile components and to 10^{-6} for non-condensable components.

For back-compatibility with earlier versions of ECLIPSE, components with critical temperatures of 10^4 and above are also assumed to be non-volatile, while those with temperatures of 10^{-6} are assumed to be non-condensable.

For surface calculations, the volatility types are set by default to the reservoir values. If different volatility types are required at the surface, the CVTYPES keyword can be used. In this case the TCRITS keyword should also be used to ensure the correct critical temperatures are used in the surface calculations.

Solid phase components

A component can be made a solid using the CVTYPE keyword. Solid components have density and enthalpy. A solid component cannot appear in the oil phase or the liquid phase, so its K-values can be defaulted.

K-values

Distribution of volatile components between the oil and gas phases is determined by K-values. The mole fraction y^c of volatile component c in the gas phase is given by $y^c = K^c(P, T) \cdot x^c$, where x^c is the mole fraction of component c in the oil phase and $K^c(P, T)$ is the equilibrium K-value, which depends on the pressure P and the temperature T. Three methods are available for specifying oil component K-values:

1 Keyword KVCR uses the K-value correlation given in [Ref. 54] and [Ref. 58], where the K-value for each oil component is given by

$$K(P, T) = (A + B/P + C \cdot P) \cdot e^{-D/(T-E)} \qquad \text{[EQ 69.39]}$$

Some care needs to be taken when using this formula to ensure that K is always positive and increases as T increases. A simplified version that avoids these problems can be used:

$$K(P, T) = (B/P) \cdot e^{-D/T} \qquad \text{[EQ 69.40]}$$

Values for B and D can be calculated by setting

$$K = P_{sat}/P$$

and finding the best straight line

$$\ln(Psat) = \ln(B) - D/T$$

through a set of points $(1/T, P_{sat})$. Suggested values for some components are given in Table 69.19 and Table 69.20.

Table 69.19 K-value coefficients (Field units)

		B (Psi)	D (^{o}R)
C1	Methane	168900	1904
C2	Ethane	170400	3062

Table 69.19 K-value coefficients (Field units) (Continued)

		B (Psi)	D (^{o}R)
C3	Propane	528400	4571
C6	Hexane	686800	6598
C10	Decane	1214000	9966

Table 69.20 K-value coefficients (Metric units)

		B (Bars)	D (K)
C1	Methane	11650	1058
C2	Ethane	11750	1701
C3	Propane	36440	2539
C6	Hexane	47360	3666
C10	Decane	83740	5537

2 Keyword KVWI uses the Wilson correlation [Ref. 68], where the K-value for each oil component is given by

$$K(P,T) = e^{5.372697 \cdot (1+A_c) \cdot (1-T_{crit}/T)} \cdot P_{crit}/P \qquad \text{[EQ 69.41]}$$

This is the same as [EQ 69.40] with coefficients

$$B = e^{5.372697 \cdot (1+A_c)} \cdot P_{crit}$$

$$D = 5.372697 \cdot (1+A_c) \cdot T_{crit} \qquad \text{[EQ 69.42]}$$

determined from the acentric factor A_c, critical temperature T_{crit} and critical pressure P_{crit}, which are defined by keywords ACF, TCRIT and PCRIT respectively.

3 Keywords KVTABTn and KVTEMP can be used to enter the K-values as tables.

Surface K-values

The default K-values for surface calculations are the reservoir K-values. Different values can be used if required:

1 If keyword KVCR is used to set the reservoir K-values, keyword KVCRS can be used to set the surface K-values.

2 If keyword KVWI is used to set the reservoir K-values, keywords ACFS, TCRITS and PCRITS can be used to set the coefficients of the Wilson correlation [EQ 69.41] for surface conditions.

Density

Oil phase density

The molar density b_o of the oil phase is given by:

$$b_o = 1/V_{oil} \qquad \text{[EQ 69.43]}$$

where the molar specific volume V_{oil} of the oil phase is calculated using Amagat's law of partial volumes:

$$V_{oil} = \sum_{c=1}^{N_c} x^c V_{oil}{}^c \qquad \text{[EQ 69.44]}$$

and x^c is the mole fraction of component c in the oil phase. The component oil phase volume $V_{oil}{}^c$ is given by:

$$V_{oil}{}^c = \frac{MW^c}{\rho^c} \qquad \text{[EQ 69.45]}$$

where MW^c is the molecular weight of component c given by the MW keyword, and the component oil phase density ρ^c is given by:

$$\rho^c = \frac{\rho_{ref}^c}{(1 + C_{T1}(T - T_{ref}))(1 - C_p(P - P_{ref}))} \qquad \text{[EQ 69.46]}$$

The reference density ρ_{ref}^c is defined by the DREF keyword; the standard temperature and pressure, T_{ref} and P_{ref} are defined by keywords TREF and PREF (or by STCOND); the thermal expansion coefficient C_{T1} is defined by the THERMEX1 keyword; and the component isothermal compressibility C_P is defined by the CREF keyword.

Oil phase density at the surface

For surface calculations, the optional keywords MWS, DREFS,TREFS and PREFS can be used to supply the values used in equations [EQ 69.45] and [EQ 69.46]. The default, if these are not supplied, is to use the reservoir values given by MW, DREF, TREF and PREF (or STCOND).

Gas phase density

The molar density of the gas phase b_g is given by:

$$b_g = 1/V_{gas} \qquad \text{[EQ 69.47]}$$

where the molar specific volume V_{gas} of the gas phase is

$$V_{\text{gas}} = \sum_{c=1}^{N_c} y^c V_{gas}{}^c + y^w V_{gas}{}^w \qquad \text{[EQ 69.48]}$$

and y^c is the mole fraction of component c in the gas phase. Each hydrocarbon component gas phase molar volume $V_{gas}{}^c$ is obtained from the gas law,

$$PV_{gas}{}^c = Z^c RT \qquad \text{[EQ 69.49]}$$

The component Z-factor Z^c is given by an empirical formula

$$Z^c = Z_0{}^c - Z_1{}^c \cdot \frac{P}{RT} \qquad \text{[EQ 69.50]}$$

where the coefficients $Z_0{}^c$ and $Z_1{}^c$ are specified by the user with the ZFACTOR and ZFACT1 keywords.

The molar specific water vapor volume $V_{gas}{}^w$ used in [EQ 69.48] is calculated from the steam density.

Determining Z-factor coefficients from the Joule-Thomson coefficient

For an ideal gas the coefficients $Z_0{}^c$ and $Z_1{}^c$ are 1 and 0 respectively. The default values are in fact 0.96 and 0, which represents a non-ideal gas, but with constant Z-factors at reservoir conditions. This simple approach can be extended by calculating average reservoir values of $Z_0{}^c$ in PVT*i* and exporting the data using the ZFACTOR keyword. However, in some circumstances it may be necessary to use a temperature dependent Z-factor by specifying values of the coefficient $Z_1{}^c$. It can be seen by substituting [EQ 69.50] into [EQ 69.49] that the coefficient $Z_1{}^c$ has dimensions volume / mole:

$$V_{gas}{}^c = Z_o{}^c \cdot \frac{RT}{P} - Z_1{}^c \qquad \text{[EQ 69.51]}$$

$Z_1{}^c$ can be determined from the Joule-Thomson coefficient μ. For a steady state (constant enthalpy) gas flow the change of temperature as a result of a pressure change is given by [Ref. 101]:

$$\mu = \left(\frac{\partial T}{\partial P}\right)_H = \frac{1}{(MW \cdot C_p)} \cdot \left(T \cdot \left(\frac{\partial V}{\partial T}\right)_P - V\right) \qquad \text{[EQ 69.52]}$$

where MW and C_p are the fluid molecular weight and specific heat. Substituting [EQ 69.48] and [EQ 69.51] into [EQ 69.52] gives

$$\mu = \sum_1^{N_c} y^c \cdot \mu^c + y^w \cdot \mu^w \qquad \text{[EQ 69.53]}$$

where

$$\mu^c = \frac{Z_1{}^c}{(MW_{gas} \cdot C_{gas})} \qquad \text{[EQ 69.54]}$$

Hence the coefficient $Z_1{}^c$ can be determined using [EQ 69.54] from the component Joule-Thomson coefficient μ^c and the average molecular weight and heat capacity of the gas phase. The coefficient $Z_1{}^c$ is also used to calculate the component enthalpy [EQ 69.60] and [EQ 69.73].

Foamy oil

When modeling foamy oil it may be necessary to model a component that has a gas-like compressibility in the oil phase (see "Foamy oil" on page 1068). This can be done using keywords CCTYPE. In this case [EQ 69.45] is replaced by

$$V_{oil}{}^c = \frac{Z^c RT}{P} \qquad \text{[EQ 69.55]}$$

Keyword CCTYPES can be used to set (or reset) the compressibility at surface conditions.

Solid phase density

If a solid phase is modeled, the molar specific volume of solid phase components V_{sol} is calculated in a similar manner to the oil volume:

$$V_{sol} = \sum_{c=1}^{N_c} s^c V_{sol}{}^c \qquad \text{[EQ 69.56]}$$

where s^c is the mole fraction of component c in the solid phase. The component solid phase volume $V_{sol}{}^c$ is given by:

$$V_{sol}{}^c = \frac{MW^c}{\rho^c} \qquad \text{[EQ 69.57]}$$

where MW^c is the molecular weight of component c given by the MW keyword, and the component solid phase density ρ^c is given by:

$$\rho^c = \frac{\rho_{ref}^c}{(1 + C_{T1}(T - T_{ref}))(1 - C_p(P - P_{ref}))} \qquad \text{[EQ 69.58]}$$

The reference density ρ_{ref}^c is defined by the SDREF keyword; the standard temperature and pressure, T_{ref} and P_{ref} are defined by keywords STREF and SPREF (or by STCOND); the thermal expansion coefficient C_{T1} is defined by the STHERMX1 keyword; and the component isothermal compressibility C_P is defined by the SCREF keyword.

Enthalpy

The enthalpy of the oil phase is calculated using a mole fraction weighted average of the component enthalpies:

$$H_{oil} = \sum_{c=1}^{N_c} x^c \cdot MW^c \cdot H_{oil}^c(T) \qquad \text{[EQ 69.59]}$$

where x^c is the mole fraction of component c in the oil phase and MW^c is the component molecular weight given by the MW keyword.

The enthalpy of the hydrocarbon components in the gas phase is calculated using a mole fraction weighted average of the component enthalpies:

$$H_{gas} = \sum_{c=1}^{N_c} y^c \cdot (MW^c \cdot H_{gas}{}^c(T) + H_{JT}{}^c(P)) \qquad \text{[EQ 69.60]}$$

where y^c is the mole fraction of component c in the gas phase. The gas phase component enthalpies include both a temperature dependent term $H_{gas}{}^c$ and a pressure dependent (Joule-Thomson) term $H_{JT}{}^c$.

The enthalpy of the hydrocarbon components in the solid phase is calculated using a mole fraction weighted average of the component enthalpies:

$$H_{sol} = \sum_{c=1}^{N_c} s^c \cdot MW^c \cdot H_{sol}{}^c(T) \qquad \text{[EQ 69.61]}$$

where s^c is the mole fraction of component c in the solid phase.

The component enthalpies in the oil and gas phases are calculated from component specific heats and heats of vaporization. Specific heat values must be given for each hydrocarbon component in at least one fluid phase. If they are only specified in one phase, then the heat of vaporization should be given. The following options are therefore available for the fluid phase enthalpies:

- Set the specific heat of components in the oil and gas phases.
- Set the specific heat of components in the oil phase and the heat of vaporization.
- Set the specific heat of components in the gas phase and the heat of vaporization.

Solid phase enthalpies are only required in simulations where a solid phase is present. The component enthalpies in the solid phase are calculated from component specific heats.

Specific heats

The specific heats C^c_{oil}, C^c_{gas} and C^c_{sol} of a component c in the oil, gas and solid phases are given by

$$C^c_{oil} = C^c_{oil_1} + C^c_{oil_2} \cdot (T - T_{st}) \qquad \text{[EQ 69.62]}$$

$$C^c_{gas} = C^c_{gas_1} + C^c_{gas_2} \cdot (T - T_{st}) \qquad \text{[EQ 69.63]}$$

$$C^c_{sol} = C^c_{sol_1} + C^c_{sol_2} \cdot (T - T_{st}) \qquad \text{[EQ 69.64]}$$

where the oil phase coefficients are defined by keywords `SPECHA` and `SPECHB`, the gas phase coefficients are defined by keywords `SPECHG` and `SPECHH` and the solid phase coefficients are defined by keywords `SPECHS` and `SPECHT`. T_{st} is the standard temperature defined by `STCOND`. The specific heats must always be positive.

If oil phase specific heats are defined, the molar enthalpy of component c in the oil phase is:

$$H^c_{oil} = \int_{T_{st}} C^c_{oil} dT = C^c_{oil_1} \cdot (T - T_{st}) + \frac{1}{2} \cdot C^c_{oil_2} \cdot (T - T_{st})^2 \qquad \text{[EQ 69.65]}$$

For temperatures above the critical temperature, $T > T^c_{crit}$, the oil phase molar enthaply is set equal to the gas phase molar enthalpy.

If gas phase specific heats are defined, the molar enthalpy of component c in the gas phase is

$$H^c_{gas} = H^c_{vaps} + \int_{T_{st}} C^c_{gas} dT \qquad \text{[EQ 69.66]}$$

$$= H^c_{vaps} + C^c_{gas_1} \cdot (T - T_{st}) + \frac{1}{2} \cdot C^c_{gas_2} \cdot (T - T_{st})^2$$

where H^c_{vaps} is the heat of vaporization at the standard temperature, defined by HEATVAPS. The oil and gas phase component enthalpies are illustrated in Figure 69.1.

Figure 69.1 Component enthalpy defined by specific heats

If a solid phase is simulated, the molar enthalpy of component c in the solid phase is

$$H^c_{sol} = \int_{T_{st}} C^c_{sol} dT = C^c_{sol_1} \cdot (T - T_{st}) + \frac{1}{2} \cdot C^c_{sol_2} \cdot (T - T_{st})^2 \qquad \text{[EQ 69.67]}$$

Heat of vaporization

A temperature dependent heat of vaporization $\Delta H^c(T)$ can be specified instead of either the oil or gas specific heat.

If the specific heat of a component c in the oil phase is not specified, then the oil phase enthalpy is calculated from the gas component enthalpy by

$$H_{oil}{}^c = H_{gas}{}^c - \Delta H^c(T) \qquad \text{[EQ 69.68]}$$

This is illustrated in Figure 69.2 where a linear function has been used for the heat of vaporization $\Delta H^c(T)$.

Figure 69.2 Component enthalpy defined by gas specific heat and heat of vaporization

Alternatively, if the specific heat of a component c in the gas phase is not specified, then the gas phase enthalpy is calculated from the oil component enthalpy

$$H_{gas}{}^{c} = H_{oil}{}^{c} + \Delta H^{c}(T) \qquad \text{[EQ 69.69]}$$

This is illustrated in Figure 69.3 where a linear function has been used for the heat of vaporization $\Delta H^{c}(T)$.

Figure 69.3 Component enthalpy defined by oil specific heat and heat of vaporization

The heat of vaporization is given by Edmister et al. [Ref. 59]:

$$\Delta H^{c}(T) = A \cdot (1 - T/T_{crit}{}^{c})^{B} \qquad T < T_{crit}{}^{c} \qquad \text{[EQ 69.70]}$$

$$\Delta H^{c}(T) = 0 \qquad T \geq T_{crit}{}^{c} \qquad \text{[EQ 69.71]}$$

where the constant A is defined by the `HEATVAP` keyword, the exponent B is defined by the `HEATVAPE` keyword, and $T_{crit}{}^{c}$ is the component critical temperature defined by the `TCRIT` keyword.

Heats of vaporization are usually obtained at the normal boiling point T_{nb}. The constant A, defined by the `HEATVAP` keyword can be found by inverting [EQ 69.70]:

$$A = \frac{\Delta H^{c}(T_{nb})}{(1 - T_{nb}/T_{crit})^{B}} \qquad \text{[EQ 69.72]}$$

where $\Delta H^{c}(T_{nb})$ is the heat of vaporization at the normal boiling point. The exponent B, entered with keyword `HEATVAPE`, is defaulted to 0.38, which is the value found in the literature (Edmister et al. [Ref. 59]). However, this can lead to convergence problems, as discussed in the next section.

Pressure dependent gas enthalpy

The gas enthalpy equation [EQ 69.60] includes a pressure dependent term $H_{JT}{}^{c}$. This term is only used when the temperature dependent terms in the oil and gas enthalpies are set using specific heats ([EQ 69.65] and [EQ 69.66]). It is not used when the heat of vaporization `HEATVAP` is used ([EQ 69.68] or [EQ 69.69]). The pressure dependent term is given by

$$H_{JT}{}^{c} = -Z_1{}^{c} \cdot (P - P_{st})$$ [EQ 69.73]

where $Z_1{}^{c}$ is specified with the ZFACT1 keyword and P_{st} is the standard pressure defined by STCOND.

Restrictions on specifying enthalpy data

The component enthalpies must obey the following relationships

$$H^{c}_{gas} \geq H^{c}_{oil} \text{ for } T < T_{crit}{}^{c}$$

$$\frac{dH_{oil}{}^{c}}{dT} > 0$$

$$\frac{dH_{gas}{}^{c}}{dT} > 0$$

ECLIPSE will issue an error message if any of these relationships does not hold. Errors can occur when specific heats are incorrectly specified, see Figure 69.4; or when using a non-linear heat of vaporization, see Figure 69.5. These problems can be avoided by specifying specific heats for both phases, with the oil specific heats equal to the gas specific heats Figure 69.6.

Figure 69.4 Specific heat data leading to Hoil > Hgas

Figure 69.5 Heat of vaporization data leading to decreasing gas enthalpy

Figure 69.6 Component enthalpy defined by equal specific heats

Sources of enthalpy data

Molar specific heats are often quoted as a third order polynomial

$$C_p = A + B \cdot T + C \cdot T^2 + D \cdot T^3$$

These can be converted to ECLIPSE input values C_{p_1} and C_{p_2} used in [EQ 69.62] - [EQ 69.64] as follows:

$$C_{p_1} = C_p(T_{st})/(MW) = (A + B \cdot T_{st} + C \cdot T_{st}^{\ 2} + D \cdot T_{st}^{\ 3})/(MW) \qquad \text{[EQ 69.74]}$$

$$C_{p_2} = \left.\frac{dC_p}{dT}\right|_{T_{st}} /(MW) = (B + 2 \cdot C \cdot T_{st} + 3 \cdot D \cdot T_{st}^{\ 2})/(MW) \qquad \text{[EQ 69.75]}$$

where MW is the component molecular weight.

Note The ECLIPSE specific heat input data is a function of the standard temperature T_{st} set by keyword `STCOND`

Specific heats calculated using [EQ 69.74] and [EQ 69.75] from data in Reid et al. [Ref. 56] are shown in Table 69.21 and Table 69.22.

The following table contains calculated specific heats in metric units at Ts = 20 deg C for selected components. These are calculated from data in Reid et al. [Ref. 56].

Table 69.21 Calculated specific heats in metric units at Ts = 20 deg C

	C_{p1}	C_{p2}
	kJ/kg/K	kJ/kg/K^2
Methane	2.199	3.505e-3
Ethane	1.725	4.645e-3
Propane	1.650	5.026e-3
n-Butane	1.669	4.570e-3
Isobutane	1.656	4.885e-3
n-Pentane	1.641	4.847e-3

Table 69.21 Calculated specific heats in metric units at Ts = 20 deg C

	C_{p1}	C_{p2}
	kJ/kg/K	kJ/kg/K^2
Isopentane	1.621	5.008e-3
n-Hexane	1.637	4.826e-3
n-Heptane	1.633	4.809e-3
n-Octane	1.630	4.798e-3
n-Nonane	1.627	4.808e-3
n-Decane	1.625	4.779e-3
n-Unadecane	1.624	4.772e-3
Dodecane	1.623	4.766e-3
n-Tridecane	1.621	4.763e-3
n-Tetradecane	1.620	4.758e-3
n-Pentadecane	1.619	4.754e-3
Hexadecane	1.618	4.752e-3
n-Heptadecane	1.618	4.751e-3
Octadecane	1.618	4.747e-3
n-Nonadecane	1.616	4.743e-3
n-Eicosane	1.615	4.775e-3

The following table contains calculated specific heats in field units at Ts = 20 deg C for selected components. These are calculated from data in Reid et al. [Ref. 56]

Table 69.22 Calculated specific heats in field units at Ts = 20 deg C (68 deg F)

	C_{p1}	C_{p2}
	BTU/lb/R	BTU/lb/R^2
Methane	5.226e-1	4.659e-4
Ethane	4.080e-1	6.201e-4
Propane	3.895e-1	6.718e-4
n-Butane	3.946e-1	6.099e-4
Isobutane	3.911e-1	6.528e-4
n-Pentane	3.876e-1	6.480e-4
Isopentane	3.827e-1	6.696e-4
n-Hexane	3.866e-1	6.452e-4
n-Heptane	3.857e-1	6.430e-4
n-Octane	3.849e-1	6.415e-4
n-Nonane	3.843e-1	6.430e-4
n-Decane	3.838e-1	6.391e-4
n-Unadecane	3.836e-1	6.381e-4

Table 69.22 Calculated specific heats in field units at Ts = 20 deg C (68 deg F)

	C_{p1}	C_{p2}
	BTU/lb/R	BTU/lb/R^2
Dodecane	3.833e-1	6.374e-4
n-Tridecane	3.829e-1	6.370e-4
n-Tetradecane	3.828e-1	6.363e-4
n-Pentadecane	3.825e-1	6.358e-4
Hexadecane	3.823e-1	6.355e-4
n-Heptadecane	3.821e-1	6.353e-4
Octadecane	3.821e-1	6.349e-4
n-Nonadecane	3.818e-1	6.344e-4
n-Eicosane	3.815e-1	6.387e-4

Viscosity

Oil viscosity temperature dependence

Oil component viscosities can be input in functional form (`OILVISCC` or `OILVISCF`) or tabular form (`OILVISCT`). The interpolation method for the tabular form can be specified using the `OILVTIM` keyword.

If the `OILVISCC` keyword is used the viscosity μ_o^c for oil component c can be specified using one of four standard formulae, the ASTM correlation:

$$\log_{10}\left(\mu_o^c + A\right) = B \cdot T^{-C} \qquad \text{[EQ 69.76]}$$

the Andrade formula ([Ref. 55] and[Ref. 56]):

$$\log_{10}\left(\mu_o^c\right) = A + B/T \qquad \text{[EQ 69.77]}$$

the Vogel formula ([Ref. 55] and [Ref. 56]):

$$\log_{10}\left(\mu_o^c\right) = A + B/(T + C) \qquad \text{[EQ 69.78]}$$

or a logarithmic formula:

$$\log_{10}\left(\mu_o^c\right) = A + B \cdot \log_{10}(T) \qquad \text{[EQ 69.79]}$$

where T is the temperature. The coefficients A, B and C can either be specified directly, or values of viscosity and temperature can be specified, in which case the simulator calculates the correlation coefficients.

The ASTM correlation can be expressed in a form similar to Prats [Ref. 57]:

$$\mu_o^c = \left(\mu_{o1}^c + A\right)^{10^{-x}} - A \qquad \text{[EQ 69.80]}$$

where $x = C \cdot \log_{10}(T/T_1)$, $(1 - A)$ is the limiting viscosity at infinite temperature, $B = \log_{10}\left(\mu_{o1}^{c} + A\right) \cdot T_1^{-C}$ and μ_{o1}^{c} is the viscosity at temperature T_1. Note that the formula is given in terms of the viscosity μ and not the kinematic viscosity $\nu = \mu/\rho$.

The keyword OILVISCF can be used to fit each oil component viscosity according to the equation

$$\mu_o^c = D \cdot T^B \qquad \text{[EQ 69.81]}$$

that is the same as the logarithmic formula specified by the OILVISCC keyword when $A = \log_{10} D$.

When all component viscosities have been calculated, the oil phase viscosity is calculated from

$$\log(\mu_o) = \sum_{c=1}^{N_c} f_c(x^c) \cdot \log\left(\mu_o^c\right) \qquad \text{[EQ 69.82]}$$

where the functions f_c are given by keyword OILVINDX, or defaulted to

$$f_c(x^c) = x^c.$$

Oil viscosity pressure dependence

The oil phase viscosity can be modified by a factor $\mu_p(P)/\mu_p(P_{vref})$ where μ_p and P_{vref} are defined by the keywords PVCO and VISCREF.

Gas viscosity temperature dependence

Gas component viscosities can be input in functional form (GASVISCF) or tabular form (GASVISCT).

If the functional form is used, each gas component viscosity is fitted according to the equation

$$\mu_g^c = A \cdot T^B \qquad \text{[EQ 69.83]}$$

where T is the temperature and c is the component number. The gas phase viscosity is calculated to be

$$\mu_g = \sum_{c=1}^{N_c+1} y^c \cdot \mu_g^c \qquad \text{[EQ 69.84]}$$

Example

For a two hydrocarbon component study, the PROPS section may look something like this

```
PROPS ====================================

-- Temperature variation with depth - deg F
TEMPVD
624 105 /
1500 105 /

-- Standard Conditions Temp deg F Press psia
STCOND
60 14.7 /

-- Water Saturation Rel Perm Function
SWFN
.2000 .0000 0.0
.4000 .00035 0.0
.5000 .00101 0.0
.6000 .00213 0.0
.7000 .0038 0.0
/

-- Gas Saturation Rel Perm Function
SGCN
.0000 .0000 0.0
.0200 .0000 0.0
.1000 .07059 0.0
.2000 .15882 0.0
.3000 .24706 0.0
.4000 .33529 0.0
.5000 .42353 0.0
.6000 .51176 0.0
.7000 .6000 0.0
/

-- Two Phase Oil Saturation Rel Perm Functions
SOF3
.2000 .0000 .0000
.3000 .0000 .1666
.4000 .02468 .33333
.5000 .12155 .500000
.6000 .30885 .66667
.7000 .59856 .83333
.8000 1.0000 1.0000
/
```

```
-- Stone 3-phase oil rel perm
STONE

-- Relative permeability endpoints vs temperature (deg F)
ENKRVT
-- temp Krwmax Krgmax Kromax Krwro  Krgro Krorg Krorw
   100  1*     1*     1*     0.0038 1*    1*    1*
   450  1*     1*     1*     0.1    1*    1*    1*     /
-- Saturation endpoints vs temperature (deg F)
ENPTVT
-- temp  Swc  Swir  Swmax  Sgc  Sgr  Sgmax  Sorw  Sorg
   100   0.20 0.20  1*     1*   1*   1*     0.3   0.2
   450   0.45 0.45  1*     1*   1*   1*     0.1   0.05 /


-- Component gas viscosities cp

GASVISCT
100  .0112  1
200  .0128  1
300  .0145  1
400  .016   1
/
-- Component oil phase viscosities cp
OILVISCT
 75 7500  7500
100 2000  2000
125  532   532
150  210   210
200   50    50
250   15    15
300    8.1   8.1
400    3.0   3.0
500    1.5   1.5
/

-- Crookston K-value coefficients
KVCR
1.89   1*
3226.8 1*
0      1*
362.7  1*
0      1*
/
```

```
-- Heats of vapourisation Btu/lb
HEATVAP
350  /
-- Critical temperatures deg R
TCRIT
358 10000 /
-- Critical pressures psia
PCRIT
666.37  1* /

-- Component reference densities in oil phase lb/ft**3
DREF
60.975 60.975 /
-- Component compressibilities in oil phase 1/psi
CREF
0.00002  0.0000126 /
-- Component specific heats Btu/lb/deg R
SPECHA
0.52 0.432 /
CNAMES
SGAS HEAVY /
MW
17.3 500 /

-- Thermal expansion coefficients 1/deg R
THERMEX1
.00035 .000315 /

-- Oil component gas phase compressibilities 1/psi
ZFACTOR
0.95 /

-- Water properties
PVTW
--  Pref       Bw        Cw          Vw        Cvw
--  PSIA       RB/STB    1/PSI       CPOISE    1/PSI
    75.000     1.0       3.E-08      .3        7.E-09
 /
-- Ref press  rock compressibility
-- psia       1/psi
ROCK
100  5.0E-05  /

-- global mole fraction vs depth
ZMFVD
624 0.0001 0.9999 /
```

Dead oil

If the oil phase is represented by a single non-volatile component, then the DEADOIL option can be used and a simple set of keywords can be used to specify the effective properties of the hydrocarbon component.

Molecular weight

The OILMW keyword can be used to specify the molecular weight.

Density

The oil density is defined as

$$\rho_o = \frac{\rho_{ref,o}}{[1 - c_{Po}(P - P_{ref})][1 + c_{T1}(T - T_{ref}) + c_{T2}(T - T_{ref})^2]} \quad \text{[EQ 69.85]}$$

where the oil compressibility c_{Po}, and the thermal expansion coefficients c_{T1} and c_{T2} are defined by the OILCOMPR keyword, $\rho_{ref,o}$ is defined by the DENSITY keyword and T_{ref} and P_{ref} are defined by the STCOND keyword.

Specific heat

A constant specific heat is used with DEADOIL. It is defined using the OILSPECH keyword.

Viscosity

Oil viscosity can be specified in tabular form (OILVISCT) or functional form (THANALV). The interpolation method for the tabular form can be specified using the OILVTIM method. The functional form follows Wingard [Ref. 60]:

$$\log_{10}(\mu_o) = A_o + B_o / T_K \quad \text{[EQ 69.86]}$$

where $A_o = -0.8772$, $B_o = 356.681$ and T_K is the temperature in K. This is the same as the live oil viscosity given by the Andrade formula [EQ 69.77].

Oil viscosity pressure dependence

The oil viscosity can be modified by a factor $\mu(P)/\mu(P_{vref})$ where $\mu(P)$ and P_{vref} are defined by the keywords PVCO and VISCREF.

Note Oil viscosity in the DEADOIL option cannot be lower than 0.1 cP. It will be reset to 0.1 cP.

Example

A typical set of dead oil keywords would be:

```
OILCOMPR
--Compressibility      therm.expansion coeff.
  5.00E-06             3.80E-04 0.0 /
OILSPECH
  specific heat
  0.5 /
OILMW
--effective molecular weight
  600.0 /
DENSITY
--for thermal, only oil density is needed.
  60.68 2* /
```

If multiple PVT regions are being used, a set of data should be entered for each PVT region. The number of PVT regions is entered as the 2nd parameter of TABDIMS.

Water and steam

Molecular weight

The molecular weight of water is defined as 18.015 g/gmol.

K-values

The K-value of water, K_{ww}, is defined as the saturation pressure of water $P_b(T)$ divided by the pressure:

$$K_{ww} = \frac{P_b}{P} \qquad \text{[EQ 69.87]}$$

By default the saturation pressure and temperature are obtained from the formula of Farouq Ali [Ref. 66]:

$$P_b = \left(\frac{T}{a}\right)^{1/b} \qquad \text{[EQ 69.88]}$$

$$T_b = aP^b \qquad \text{[EQ 69.89]}$$

Ejiougu et al. [Ref. 64] give the following values for the constants a and b when the temperature is measured in °C, and in MPa:

$a = 180.89, b = 0.2350$

Alternatively, the water K-value can be obtained from a correlation of the form

$$K_{ww} = \frac{P_b}{P} = (A + B/P + C \cdot P) \cdot e^{-D/(T-E)}$$

where the coefficients A, B, C, D and E are specified using the KVCRWAT keyword.

Water density

The water density can be calculated in one of three ways; using an internal analytic function; using an analytic function with user defined coefficients; or using internal steam tables.

Analytic formula with internally defined coefficients

Trangenstein's modification of Kell's 1975 correlations [Ref. 61] is used for the water density (default / THANALB):

$$\rho_w = \frac{A_0 + A_1 T_C + A_2 T_C^{\,2} + A_3 T_C^{\,3} + A_4 T_C^{\,4} + A_5 T_C^{\,5}}{1 + A_6 T_C} e^{c_{Pw}(P_P - A_7)} \qquad \text{[EQ 69.90]}$$

where ρ_w is in kg/m^3, the temperature T_C is in °C, the pressure P_P is in MPa, and

$A_0 = 999.83952$ $\quad A_1 = 16.955176$

$A_2 = -7.987\times10^{-3}$ $\quad A_3 = -46.170461\times10^{-6}$

$A_4 = 105.56302\times10^{-9}$ $\quad A_5 = -280.54353\times10^{-12}$

$A_6 = 16.87985\times10^{-3}$ $\quad A_7 = -10.2$

The water compressibility c_{Pw} (in MPa^{-1}) is defined by the PVTW keyword or calculated from the WATERTAB data.

Analytic formula with user defined coefficients

If the WATDENT keyword is used then the water density can be specified in a similar manner to the oil component densities [EQ 69.46]:

$$\rho_w = \frac{\rho_{ref,w}}{[1 - c_{Pw}(P - P_{ref})][1 + c_{T1}(T - T_{ref}) + c_{T2}(T - T_{ref})^2]} \qquad \text{[EQ 69.91]}$$

where the reference density is given by

$$\rho_{ref,w} = \frac{\rho_{s,w}}{B_w(P_{ref})}$$

and

$\rho_{s,w}$ is the water density at surface conditions given by the DENSITY keyword

P_{ref} is the reference pressure given by the PVTW keyword

$B_w(P_{ref})$ is the formation volume factor given by the PVTW keyword

c_{Pw} is the water compressibility defined by the PVTW keyword or calculated from the WATERTAB data.

T_{ref} is the reference temperature given by the WATDENT keyword

c_{T1} is the first coefficient of thermal expansion, given by the WATDENT keyword

c_{T2} is the second coefficient of thermal expansion, given by the WATDENT keyword

Steam tables

If the THTABB keyword is used, the water density is calculated from steam tables [Ref. 61].

Steam density

The steam density can be calculated in one of two ways; using an internal analytic function; or using internal steam tables.

Analytic formula with internally defined coefficients

The analytic formula is used by default or if the THANALB keyword is used. The steam density is calculated from the real gas law:

$$\rho_g = \rho_{gb}\frac{T_b}{T} \qquad \text{[EQ 69.92]}$$

where T_b is the boiling point temperature and ρ_{gb} is the density at the boiling point. The saturated water vapor density is taken from Tortike and Farouq Ali [Ref. 62]:

$$\rho_{gb} = \exp\left[C_0 + C_1 T_{bK} + C_2 T_{bK}^{\ 2} + C_3 T_{bK}^{\ 3} + C_4 T_{bK}^{\ 4} + C_5 T_{bK}^{\ 5}\right] \qquad \text{[EQ 69.93]}$$

where, ρ_{gb} is in kg/m^3, T_{bK} is the boiling point temperature in K and

$C_0 = -93.7072$ $\quad C_1 = 0.833941$ $\quad C_2 = -0.003208$

$C_3 = 6.57652\times10^{-6}$ $\quad C_4 = -6.93747\times10^{-9}$ $\quad C_5 = 2.97203\times10^{-12}$

Steam tables

If the THTABB keyword is used, the steam density is calculated from steam tables [Ref. 55].

Energies and enthalpies

Water and steam energies and enthalpies are read from steam tables. The default is to use tables from Perry and Green [Ref. 55]. Tables from Haywood [Ref. 65] can be selected using the OPTIONS3 keyword. Analytic formulae, based on Perry's tables, can also be used, keyword THANALH.

Water viscosity

The water viscosity can be defined in tabular form (WATVISCT) or in functional form (default). The functional form is taken from Grabowski et al. [Ref. 63]

$$\mu_w = \frac{A_w}{-1 + B_w T_F + C_w T_F^{\ 2}} \qquad \text{[EQ 69.94]}$$

where the temperature T_F is in °F and the default coefficients

$A_w = 2.1850$ $\quad B_w = 0.04012$ $\quad C_w = 5.1547\times10^{-6}$

The coefficients maybe modified using the THWVC keyword.

Water viscosity pressure dependence

The water viscosity can be multiplied by a factor $\mu_p(P)/\mu_p(P_{vref})$ where μ_p and P_{vref} are defined by the keywords PVTW and VISCREF.

Steam viscosity

The steam viscosity is taken from Ejiogu and Fiori [Ref. 64]

$$\mu_g = A_g + B_g T_C + C_g P_P^{D_g} \quad \text{[EQ 69.95]}$$

where the temperature T_C is in °C, the pressure P_P is in MPa and the default coefficients

$A_g = 4.9402 \times 10^{-3}$, $B_g = 5.0956 \times 10^{-5}$, $C_g = 2.9223 \times 10^{-6}$, $D_g = 2.5077$

The coefficients maybe modified using the `THSVC` keyword.

Combustion

The chemical reaction keywords, "Chemical Reactions" on page 147 can be used to model combustion. An example is given in "Example 1: Combustion" on page 157.

Foamy oil

'Foamy oil' occurs in some heavy oil reservoirs during cold primary production. It is believed that the highly viscous heavy oil traps small gas bubbles as they form, hindering the development of the gas phase. The result is lower than expected gas production, a prolonged solution gas drive and higher than expected oil production.

A number of methods of modeling foamy oil have been proposed, using one, two or three components to represent the gas:

- In the simplest method, a single component is used to represent the gas and the foamy oil effect is modeled by increasing the critical gas saturation. This method has been successfully used to simulate laboratory studies. However, it does not simulate the dynamic processes of formation and dispersion of the trapped gas, which are important in reservoir studies.
- In the two gas component model, one component is used to represent the gas in the oil phase and the (gaseous) foam phase, and a second component is used to represent the free gas. Formation of trapped gas is modeled by the usual flash equations. The dispersion of the trapped gas into free gas is modeled by a chemical reaction, see "Chemical Reactions" on page 147. The reduced mobility of the trapped gas can be modeled by increasing the gas phase viscosity of the first gas component. However, since the gas mobility is a weighted average of all the gas component mobilities, this will result in the trapped gas moving too fast, and the free gas moving too slowly.
- In the three gas component model, different components are used to represent the gas in the oil phase, foamy oil phase and gas phase. In the foamy oil phase, the trapped gas is modeled as an oil-phase component, or even a solid component. This ensures that the trapped gas moves with the oil phase (or is immobile if it is modeled as a solid). A chemical reaction is needed to represent the formation of trapped gas from dissolved gas, replacing the flash. This has the added advantage of allowing non-equilibrium effects to be modeled.

  In this model the reported oil saturation (or solid saturation) includes a contribution from the trapped gas. This contribution can be stored as a property `SFOIL` (or `SFSOL` for 'foamy solid') which can be output using keywords `RPTRST`, `RPTSCHED` and `RPTSOL` or with the summary vectors `BSFOIL` (or `BSFSOL`) .

Example

In the following example four components are used, one to represent the heavy oil and three to represent the gas in the oil, foam and gas phases. Keyword CVTYPE is used to set the volatility types of the three gas components. Keyword CCTYPE is used to set the compressibility of the trapped gas component.

Two reactions are used to represent the liberation of gas:

1 dissolved gas to trapped gas

2 trapped gas to free gas.

Reaction 1 uses equilibrium deviation reaction rates, defined by the EQLDREAC keyword.

To get the correct quantities of oil and gas at the surface, the keywords CVTYPES, TCRITS and KVCRS are used to reset values for the surface flash, and keyword CCTYPES is used to reset the oil phase density.

The RUNSPEC section will contain the following keywords:.

```
COMPS
  4 / 1 heavy component and 3 components representing gas

REACTIONS
  2  1  1 / 2 reactions, 1 equilibrium deviation reaction,
--                        1 equilibrium k-value
```

The PROPS section will contain the following keywords:

```
CNAMES
        HEAVY    DISS_GAS TRAP_GAS FREE_GAS /

-- Volatility

CVTYPE
        DEAD     DEAD     DEAD     GAS      / gas trapped in oil, or
--      DEAD     DEAD     SOLID    GAS      / trapped gas is immobile

KVCR
        0        1*       1*       1*
        1.214E6  1*       1*       1*
        0        1*       1*       1*
        9966     1*       1*       1*       /

-- Compressibility

CCTYPE
        1*       1*       GAS      1*       /

-- Reactions
--   1: DISS_GAS <-> TRAP_GAS
--   2: TRAP_GAS -> FREE_GAS

STOREAC
        0        1        0        0        0 / reaction 1
        0        0        1        0        0 / reaction 2
STOPROD
        0        0        1        0        0 / reaction 1
        0        0        0        1        0 / reaction 2
REACCORD
        0        0        0        0        0 / reaction 1
        0        0        1        0        0 / reaction 2

REACRATE
     0.1   0.5   / reactions 1 - 2

EQLDREAC
--   #  TYPE   XC   YC   KV
     1     1    2    0    1 / reaction 1 proportional to X2-1/K1

EQLDKVCR
0 1.704E5 0 3062 / K1 (K-value for ethane in field units)
```

```
-- For surface conditions use 'normal' component values

CVTYPES
        1*       1*       1*      1*       /
CCTYPES
        1*       1*       1*      1*       /
KVCRS
        0        0        0       0
        1.214E6  1.704E5  1.704E5 1.704E5
        0        0        0       0
        9966     3062     3062    3062     /
TCRITS
        1409.7   549.7    549.7   549.7    /
```

The REACENTH keyword could be used here to define a latent heat, for example a negative value could be used for reaction 1.

Thermal features

Heat loss

Heat loss to the overburden, underburden and sides of the reservoir can be modeled. The ROCKDIMS, keyword should be used to define the number of rock types outside the reservoir. Each rock type can have different properties, defined using the ROCKPROP keyword. The ROCKCON keyword is used to define which cells on the edge of the reservoir connect to each rock type.

Two methods can be used to calculate the heat loss, a numerical method and an analytic method. In the numerical method the heat loss rate per unit area is given by

$$-k \cdot \left.\frac{\partial T}{\partial n}\right|_{n=0}$$

where k is the conductivity, and the temperature gradient outside the reservoir $\frac{\partial T}{\partial n}$ is found by solving the temperature equation

$$c \cdot \frac{\partial T}{\partial t} = k \cdot \frac{\partial^2 T}{\partial n^2}$$

where c is the rock heat capacity.

In the analytic method, Vinsome and Westerveld's approximate solution of the temperature equation is used [Ref. 78]. The heat loss per unit area in time Δt is given by

$$\int_0^\infty c \cdot T(t+\Delta t)dn - \int_0^\infty c \cdot T(t)dn$$

Summary vectors FHLR and FHLT can be used to obtain information on the field heat loss rate and field heat loss total.

Initialization

Initial values of all the primary variables, P, m_c and e need to be determined at the beginning of the simulation. Four methods are available:

1. The solution can be read from a save file or a RESTART file generated by an earlier run.
2. The solution can be determined from properties set directly in each grid block.
3. The solution can be calculated by the equilibrium facility.
4. The pressure can be calculated by equilibrium and the other variables determined from properties set directly in each grid block.

Restarting from a previous run

There are two different ways of restarting from a previous run.

- With a fast restart the `LOAD` keyword can be used to read data that had been previously saved using the `SAVE` keyword. This will only work if the same version of the simulator is used for both runs.
- With a flexible restart the `RESTART` keyword can be used to read data that had been previously output using the `RPTRST` keyword.

Explicit initialization

In an explicit initialization the component molar densities [EQ 69.22] and the energy density [EQ 69.23] can be determined from the component phase mole fractions and the saturations [EQ 69.21]:

$$m_c = S_o \cdot b_o \cdot x^c{}_o + S_g \cdot b_g \cdot x^c{}_g + S_w \cdot b_w \cdot x^c{}_w \qquad \text{[EQ 69.96]}$$

$$e = \frac{V_p}{V_b} \cdot (S_o \cdot b_o \cdot e_o + S_g \cdot b_g \cdot e_g + S_w \cdot b_w \cdot e_w) + \left(1 - \frac{V_p}{V_b}\right) \cdot e_r \qquad \text{[EQ 69.97]}$$

The phase molar densities $b^m{}_p$ and energies e_p are functions of the pressure, temperature, component phase mole fractions and the component and rock properties. You must therefore specify

pressure `PRESSURE`,

temperature `TEMPI` or `TEMPVD`,

two of the three saturations `SOIL`, `SGAS` and `SWAT`, and

Data can also be read from `RESTART` files using the `GETSOL` keyword.

You must also specify the component liquid and vapor mole fractions $x^c{}_o$ and $x^c{}_g$. There are a three ways this can be done:

- directly using the `XMF` and `YMF` keywords,
- directly, as functions of pressure using the `XMFVP` and `YMFVP` keywords,
- using the `NEI` keyword to specify sample liquid mole fractions.

When specifying the initial component data directly, it is up to you to ensure the fluid composition is in thermodynamic equilibrium, see "Thermodynamic equilibrium condition" on page 1038. Otherwise, the solution and properties change the first time a flash calculation is performed. This difficulty is avoided if the `NEI` keyword is used.

Initialization with NEI

When the `NEI` keyword is used, the vapor mole fractions are set so that the composition is in thermodynamic equilibrium, [EQ 69.7]. The simulator tests to see if this composition gives the correct phase saturations - if not it adjusts the composition of the component with the highest gas phase mole fraction, so that the phase saturations are exactly matched.

Equilibration

In the equilibrium calculation a steady state solution of the flow equations is sought. This can be achieved if the pressure gradient in [EQ 69.28] is zero. For thermal simulations the heat conduction terms [EQ 69.31] should also be zero, although this will in fact be determined by the initial temperatures specified by the user. The equilibrium calculation is discussed in detail in "The equilibration algorithm" on page 547.

The `EQUIL` keyword is used to define the depths of the gas-oil contact and water oil-contact. If the reservoir does not contain a gas-oil contact the `EQUIL`. keyword can be used to define the pressure at a datum depth.

The `TEMPVD` keyword is used to define initial temperatures.

Either the `COMPVD`, `ZMFVD` or the `ZI` keyword is used to define the hydrocarbon mole fractions ζ_c , see [EQ 69.12].

Mixed initialization

It is possible to engineer a mixed initialization where the pressures are calculated from a reference pressure at a given depth and the other properties are set explicitly. To do this an initial equilibration run is performed using the `EQUIL` keyword. The pressures from this run are output to a restart file:

```
SOLUTION
EQUIL
...      /     set pressure at reference depth
RPTRST
PRESSURE /     output pressure
...
SCHEDULE
END
```

The equilibration run can be stopped at the start of the `SCHEDULE` section. In the main run, explicit initialization is used, but the pressures are read from the equilibration run:

```
SOLUTION
GETSOL
PRESSURE EQUILRUN 0 /   get pressure from previous run
/
SOIL
...                 /   set saturations
...
```

Although the equilibration run does not give the correct saturations, the pressures are reasonably accurate, since the oil and water densities are similar. These pressures are combined with the desired saturations in the main run.

Aquifers

The temperature of the aquifer water or gas is specified in the aquifer definition keyword (see the AQUCT, AQUFETP, AQUCHWAT, AQUCHGAS or AQUFLUX keywords) for analytic aquifers. If the temperature of the analytic aquifer is defaulted and the TEMPVD keyword is specified, the aquifer temperature will be interpolated from the TEMPVD table using the aquifer depth. Otherwise if the temperature of the analytic aquifer is defaulted and the TEMPVD keyword is not specified, the RTEMP temperature will be used.

The temperature of the numerical aquifer water is constant. If the TEMPVD keyword is used, the aquifer temperature will be the temperature at the aquifer depth. Otherwise the RTEMP temperature is used.

Heaters

Heaters can be placed in any grid cell using the HEATER keyword, or in an LGR cell using the HEATERL keyword. These keywords specify the maximum rate of heating H_{max} and a maximum temperature at which the heater should operate T_{max}. The number of heaters allowed is controlled by the HEATDIMS keyword.

Depending on the arguments in the HEATER keyword the heater rate is given either by

$$H = min[(T_{max} - T_c) \times R, H_{max}]$$

where T_c is the cell temperature and R the temperature-dependent heat injection rate, or

$$H = min\left(H_{max}, V \cdot \frac{\Delta E}{\Delta t}\right) \qquad \text{[EQ 69.98]}$$

where V is the cell volume, Δt is the time step, and the estimated change in energy density ΔE is given by

$$\Delta E = e_t(T_{max}) - e_{t-\Delta t}$$

where $e_t(T_{max})$ is the cell energy density when the cell temperature is T_{max} and $e_{t-\Delta t}$ is the cell energy density at the previous time step.

In conditions where there is no heat flow from a cell, a heater governed by [EQ 69.98] will heat the connection cell at a constant rate H_{max} until the cell reaches a temperature T_{max}. However, if there is energy flow into or out of the cell, then [EQ 69.98] is only approximate. In this case the heater rate may be less than H_{max} for temperatures near T_{max}, or the cell temperature may exceed T_{max} slightly; the discrepancies depend on the size of the heat flow and the size of the time step.

Output of heater performance data to the print file can be controlled by using item 14 of the RPTPRINT keyword.

Summary vectors WHTR and WHTT contain the energy rate and energy total for individual heaters and require a list of heater names. Vectors FHTR and FHTT contain the energy rate and energy total for all heaters.

Well inflow

Keywords COMPMOBI and COMPMBIL can be used to set the mobility used in calculating the completion flow for injection wells. Alternatively keywords COMPKRI and COMPKRIL can be used to set the relative permeability used in calculating the completion flow for injection wells.

Use of these keywords may improve the speed and stability of thermal runs - although there may be a slight loss in accuracy of the calculated injector BHP. This is not usually a matter of concern if the well is on a rate control. Furthermore, for steam injection, the injection pressure may not depend strongly on the mobility.

Keywords COMPAGH and COMPAGHL can be used to set average gravity heads for completions.

See "Well Inflow Performance" on page 1187 for further information on well inflow calculations.

Water, steam and gas injection

An injection fluid made up of water, steam, gas, or a mixture of all three, can be specified with the WCONINJE keyword. If the injection fluid contains hydrocarbon gas the WINJGAS keyword can be used to specify the gas composition.

The enthalpy of the injected fluid is calculated from data entered using the WINJTEMP keyword. If the injection fluid contains just steam or water there are four ways of entering the data:

1. Specify the steam quality Q_i and temperature T_i, pressure derived from the temperature.
2. Specify the steam quality Q_i and pressure P_i, temperature derived from the pressure.
3. Specify the pressure P_i and temperature T_i.
4. Specify the enthalpy H_i and pressure P_i.

The first two methods can be used for injection of two-phase saturated water and steam when the steam quality is between 0 and 1. The enthalpy of each phase is set to the saturated value at the specified temperature or pressure, and the injection fluid enthalpy is then calculated by interpolation:

$$H_i = Q_i \cdot H_s^{sat}(P_i, T_i) + (1 - Q_i) \cdot H_w^{sat}(P_i, T_i)$$

where

$$P_i = P_{sat}(T_i) \text{ and } T_i = T_{sat}(P_i)$$

The third method can be used for injection of single phase water or steam. The enthalpy is determined from steam tables

$$H_i = H_w(P_i, T_i) \text{ if } P_i > P_{sat}(T_i) \quad ,$$

or

$$H_i = H_s(P_i, T_i) \text{ if } P_i < P_{sat}(T_i) .$$

This method should not be used for two-phase injection, since the enthalpy of a saturated fluid is not uniquely defined by the pressure and temperature.

The fourth method can be used in both one-phase and two-phase cases.

If steam and gas are injected together then the same four ways can be used to specify the enthalpy. In methods 1,2 and 3 the enthalpy of the gas is calculated from the pressure and temperature and added to the steam enthalpy. If method 4 is used, the specified enthalpy is taken to be the enthalpy of the fluid.

If hydrocarbon gas is injected without steam then only methods 3 and 4 should be used to specify the fluid enthalpy.

Temperature at the bottom hole pressure

At the bottom hole pressure P_w, the injection enthalpy is equal to the enthalpy specified by the WINJTEMP keyword. Therefore the temperature of the well T_w can be found by solving

$$H(P_w, T_w) = H_i(P_i, T_i)$$

The well temperature will only be equal to the temperature specified using the WINJTEMP keyword, T_i, if the pressure P_i is equal to the bottom hole pressure P_w.

Steam trap

In DEADOIL simulations, steam production can be prevented by using either the WELLPROD or WCONPROD keyword to set the well BHP above the saturated water pressure in the well cells. The difference between the BHP and the saturated water pressure can be controlled either by specifying a pressure offset or a temperature offset.

The same keywords can be used to control the BHP in LIVEOIL simulations. However, if the reservoir contains volatile hydrocarbons, some steam may be produced at pressures above the water saturation pressure.

Steam production rate

The steam production rate can be controlled by using either the WELLPROD or WCONPROD keyword, or modified using the WELTARG keyword. The steam rate should be specified as a cold water equivalent (C.W.E.) volume/time. The rate is imposed at reservoir conditions, that is the well steam rate is calculated by summing the number of moles of water in the gas phase entering each completion. The simulator converts the molar rate to a C.W.E. rate by multiplying by the surface molar volume of water.

The same value of surface molar volume of water is used for steam rate control and water rate control. In dead oil runs it is calculated from the internal steam tables at the standard conditions (STCOND). In live oil runs it is calculated from the DENSITY keyword.

Dual porosity

The use of dual porosity grids in a thermal calculation makes additional assumptions about the distribution of rock volume between the porosities. In single porosity calculations, the rock volume is assumed to be the bulk volume minus the pore volume. This approach cannot be used with dual porosity, as it leads to the rock volume being represented multiple times, one in each porosity. Instead, default assumptions are made about the rock volume distribution, which can be overridden by the user.

The default behavior assumes that the rock is associated with the matrix porosity, as matrix pore volume is contained in pores in the rock, as compared with fracture pore volume, which is associated with gaps in the matrix rock. However, to represent the local thermal effect of rock bounding the fracture, which may cool or heat the fracture fluids before a thermal transient advances far into the matrix rock, an amount of rock equal to the fracture pore volume is associated with the fracture. To prevent negative rock volumes being calculated if the porosities are large, this fracture rock volume will be proportionately reduced, so that the matrix/fracture rock volume ratio equals that of the matrix/fracture pore volume ratio in the case that negative volumes would otherwise be produced.

This approach may not be appropriate in all cases, and the user has the option of overriding this behavior by providing a weighting factor for each cell in each porosity, using the ROCKSPLV keyword. In this case, the total rock volume, defined as the average bulk volume for all porosities for a particular cell, less the total pore volume over all porosities for that cell, is multiplied by the cell weighting factor specified by ROCKSPLV, to give the rock volume for that porosity. It is not required that the weighting factors sum to unity, and so this keyword can also be used as a multiplier.

The default behavior may also be appropriate for triple porosity, where the non-matrix porosities are larger voids. For general multiple porosity, where the sub-matrix porosities can have a variety of user-defined interpretations, the default behavior may or may not be appropriate, depending on whether the sub-matrix cells are used to model transients in the matrix, or for other purposes.

The use of the ACTNUM keyword with dual porosity and the thermal option creates additional options for combining the various data inputs, and the user should check carefully that the final result is what is expected, as the ACTNUM and rock proportioning behaviors are applied separately from each other.

Multi-segment wells

Multi-segment wells can be used with the Thermal option. Heat storage in the tubing and casing can be specified using the WELSEGS keyword. This keyword can also be used to specify longitudinal conduction in the tubing and casing between successive segments.

Heat conduction to the reservoir, and heat loss outside the reservoir can be specified using keyword WSEGHEAT. Heat conduction to the reservoir occurs between segments and completion grid cells. Therefore it may be necessary to define completions in each grid cell the well passes through. These completions can be shut if no flow occurs between the grid cell and the segment. The length of well in each completion cell can be defined using the COMPSEGS keyword.

WSEGHEAT can also be used to model radial heat conduction between non-successive segments, for example between tubing segments and casing segments.

For multi-segment well source terms, the keyword WSEGEXSS can be used to specify the temperature of imported gas.

Temperature dependent well cutback

Well production rates can be reduced if the well temperature exceeds given limits. Keywords TCBDIMS and WELLTCBT can be used to set up tables of temperatures and cut back factors. The keyword WELLTCB can be used to associate a well with a table and the keyword WELLTCBC can be used to control the switching from one rate to another. Note, these keywords have no effect if the well is controlled by BHP.

The WECON keyword can also be used to close a well if the temperature exceeds a given limit.

Temperature range checking

The keyword TRANGE can be used to check that the reservoir temperature stays within expected limits.

Thermal output

Temperature, energy, enthalpy and steam data can all be output to the SUMMARY file, PRINT file or for input to graphical post-processors. See the keywords SUMMARY, RPTSCHED and RPTRST.

Convergence criteria

Limiting non-linear solution updates

Limits P_{lim}, S_{lim} and T_{lim} can be set on the pressure change, specific volume change and temperature change during a non-linear iteration to help control the convergence process. The solution update ΔX_{n-1} in [EQ 69.32] is multiplied by a reduction factor λ, $0 < \lambda \leq 1$ chosen so that for every cell:

$$\lambda \cdot |\Delta P| \leq P_{lim}$$

$$\lambda \cdot \left| \Delta X' \cdot \frac{\partial V}{\partial X'} \right| \leq S_{lim},$$

$$\lambda \cdot \left| \Delta e \cdot \frac{\partial T}{\partial e} \right| \leq T_{lim}$$

where X' is any solution variable except the pressure

$$X' = (m_1, \ldots, m_N, m_w, e)$$

and V is the total specific volume, equal to the sum of the phase specific volumes defined in [EQ 69.18]. The temperature limit is only used in thermal simulations, and only in fully implicit cells. Limits S_{lim}, T_{lim} and P_{lim} are set using items 13, 14 and 15 of the CVCRIT keyword.

Appleyard chop

After the solution is updated new values of the saturations and temperatures are calculated by performing a flash. If the saturations in an implicit cell change significantly near a critical value - e.g. if a phase appears or disappears or becomes mobile / immobile - then the solution change in that cell is chopped, so that the new saturations are close to the critical point.

The Appleyard chop is also performed if a property change in an implicit cell is large. Chops are performed if the saturation change is greater than 0.2, or if the temperature change is greater than T_{lim}.

The saturation based chop can be switched off by using item 18 of keyword OPTIONS3. The temperature based chop can be switched off by using item 27 of keyword OPTIONS3. Both chops can be switched off by using item 25 of keyword OPTIONS3.

Non-linear convergence

The non-linear iterations are deemed to have converged when the reduction factor λ is greater than 0.9 and the pressure change, specific volume change and temperature change during a non-linear iteration are less than limits P_{con}, S_{con} and T_{con}:

$$\lambda \cdot |\Delta P| \le P_{con}$$

$$\lambda \cdot \left|\Delta X' \cdot \frac{\partial V}{\partial X'}\right| \le S_{con}$$

$$\lambda \cdot \left|\Delta e \cdot \frac{\partial T}{\partial e}\right| \le T_{con}$$

The temperature condition is only used in thermal simulations, and only in fully implicit cells. Limits P_{con}, S_{con} and T_{con} are set using items 1, 7 and 8 of the CVCRIT keyword.

Time stepping criteria

Rejecting a time step

A time step can be rejected even though the non-linear iterations have converged, if the solution change during the time step is too big. In these cases a smaller time step will be used. A time step will only be accepted if the change in specific volume, the throughput, TPT and TTE are less than limits S_{max}, TPT_{max} and TTE_{max}:

$$\sum_{X'} \left|X'_t - X'_{t-\Delta t}\right| \cdot \overline{\left|\frac{\partial V}{\partial X'}\right|} \le S_{max} \qquad \text{[EQ 69.99]}$$

$$TPT \le TPT_{max} \qquad \text{[EQ 69.100]}$$

$$TTE \le TTE_{max} \qquad \text{[EQ 69.101]}$$

where X' is any solution variable except the pressure:

$$X' = (m_1, \ldots, m_N, m_w, e),$$

$\overline{\left|\frac{\partial V}{\partial X'}\right|}$ is the average value of $\left|\frac{\partial V}{\partial X'}\right|$ over all cells

The throughput is the volume flow divided by the cell volume and is discussed further in "Throughput target (TPT)" on page 208. The throughput limit is only applied to IMPES cells.

The the time truncation error is a measure of the "acceleration" of the specific volume and is discussed further in "Time truncation error (TTE)" on page 208.

In thermal simulations three additional convergence criteria can be applied. The changes in saturation SAT_p of each phase p, temperature and energy in every cell must be less than limits SAT_{max}, T_{max} and e_{max}:

$$\left|SAT_{p,t} - SAT_{p,t-\Delta t}\right| \le SAT_{max} \quad \text{[EQ 69.102]}$$

$$\left|T_t - T_{t-\Delta t}\right| \le T_{max} \quad \text{[EQ 69.103]}$$

$$\left|e_t - e_{t-\Delta t}\right| \le e_{max} = c_p \cdot T_{max} \quad \text{[EQ 69.104]}$$

The maximum energy change is defined by the maximum temperature change, using a typical value of specific heat c_p = 2347 kJ/m^3/K = 35 Btu/ft^3/R. The saturation limit can be used instead of the solution change limit.

Limits TTE_{max}, TPT_{max}, S_{max}, SAT_{max} and T_{max} are set using items 7, 9, 13, 16 and 18 of the TSCRIT keyword.

Determining a new time step

New time steps Δt are chosen so that the predicted solution change is less than a specified target. Targets TTE_{tar}, TPT_{tar}, S_{tar}, P_{tar}, SAT_{tar}, T_{tar} are set for the time truncation error, throughput and changes in specific volume, pressure, saturation and temperature. An energy target e_{tar} is calculated from the temperature target.

$$e_{tar} = T_{tar} \cdot c_p$$

Changes in parameters for the new time step are estimated by multiplying the rate of change in the previous time step by the new time step length.

Values for targets TTE_{tar}, TPT_{tar}, S_{tar}, P_{tar}, SAT_{tar} and T_{tar} are set using items 6, 8, 10, 12, 15 and 17 of the TSCRIT keyword.

Determining implicit cells in AIM runs

The throughput and changes in specific volume and temperature for a time step are compared with targets:

$$R_P = (TPT_t)/(TPT_{tar})$$

$$R_S = (\Delta S_t)/(S_{tar})$$

$$R_T = (\Delta T_t)/(T_{tar})$$

These ratios can be output to the restart file by using mnemonics RATP, RATS or RATT with the RPTRST keyword. The ratios are extrapolated to the new time step $t + \Delta t$, and cells are made implicit if these extrapolated ratios exceed values set in AIMCON.

Instability in AIM runs due to thermal conduction

In AIM runs the implicit cells are determined using throughput criteria. However in thermal runs, conduction can cause instability in explicitly solved cells. To prevent this instability:

- Rock filled cells (or near rock filled cells) should be solved fully implicitly. Therefore, in AIM runs item 3 of the AIMCON keyword should be set to a positive value.
- Problems where conduction is significant should not be solved using AIM.

Table 69.23 Summary of convergence criteria and time stepping criteria

Notes	Parameter	Non linear iteration		Time step change		
		ΔX lim	ΔX con	max	tar	mnemonic
	Pressure	CVCRIT 15	CVCRIT 1		TSCRIT 12	PCT
	Specific volume	CVCRIT 13	CVCRIT 7	TSCRIT 13	TSCRIT 10	SCT
	Time truncation error			TSCRIT 7	TSCRIT 6	TTE
IMPES cells	Throughput			TSCRIT 9	TSCRIT 8	TPT
Thermal only	Temperature	CVCRIT 14	CVCRIT 8	TSCRIT 18	TSCRIT 17	TEC
Thermal only	e/cp			TSCRIT 18	TSCRIT 17	EnC
Thermal only	Saturation			TSCRIT 16	TSCRIT 15	SAC

Note If items 13 and 14 of keyword CVCRIT are not set, the default values for the non-linear step limits S_{lim} and T_{lim} are calculated from items 10 and 17 of keyword TSCRIT. This means that if the target specific volume is relaxed (item 10 of TSCRIT) - which may be the case if the target saturation is used (item 15 of TSCRIT) - then item 13 of keyword CVCRIT should be used to override the default value for the non-linear step limit S_{lim}.

Current limitations

At present a number of options cannot be used with the Thermal option. These include:

- "Diffusion" on page 211
- "Transport coefficients" on page 354
- "Non-Darcy Flow" on page 711
- "Optimization" on page 727
- "Surface Tension Effects" on page 983.

Converting a blackoil data set to thermal

To convert a blackoil data set to run with the thermal option, the blackoil fluid description must be modified to a compositional fluid description and the thermal and temperature dependent properties of the reservoir, fluids and wells must be described. A first step in this process is to use thermal keywords to create a data set that mimics the isothermal blackoil run. At this stage many of the temperature dependent properties can be defaulted. Once it has been confirmed that this gives similar results, the data set can be enhanced, by adding more components to improve the fluid description, and by including temperature effects and tuning the temperature dependent properties. This section gives suggestions for carrying out the first step of this process - running a blackoil simulation with the thermal simulator.

Converting ECLIPSE 100 to ECLIPSE 300 blackoil

If the blackoil data set is an ECLIPSE 100 data set, then you should check that it runs in ECLIPSE 300 too. If necessary remove any ECLIPSE 100 keywords that are not recognized by ECLIPSE 300.

ECLIPSE 100 data sets using the Temperature option ("Temperature option" on page 1009) can be run with ECLIPSE 300 if the `SCHEDULE` section data is converted to Thermal keywords - in particular `WTEMP` data should be changed to `WINJTEMP` data.

RUNSPEC section

Add the keyword `THERMAL` and remove the keyword `BLACKOIL`.

If there is no gas in the blackoil simulation, the thermal simulation can be run as a dead oil simulation by adding keyword `DEADOIL`. Otherwise the thermal simulation can be run as a live oil with two components, one representing the blackoil oil, and the other representing the blackoil gas.

The number of hydrocarbon components must be defined with the `COMPS` keyword. As a first step use two components if both oil and gas are being simulated, and one component if only one hydrocarbon phase is being simulated.

If the data set contains horizontal wells, add the `HWELLS` keyword to the `RUNSPEC` section.

Reservoir properties

The rock heat capacity (`HEATCR`) and thermal conductivity (`THCONR`) must be specified in the `GRID` section. For an isothermal run the actual values are not important - a high rock heat capacity can be used to dampen any temperature fluctuations and the thermal conductivity can be set to zero.

In the `PROPS` section the reservoir temperature can be set using the `RTEMP` or `TEMPVD` keyword The surface temperature and pressure can be set using the `STCOND` keyword.

Dead oil fluid properties

Keywords, OILCOMPR, OILVISCT, OILSPECH and OILMW have to be added. The blackoil keywords (PVTO and PVDG) must be removed.

Live oil fluid properties

If black oil data is derived from PVT experiments, it should be possible to go back to the original data and derive thermal properties too. If this is not possible the keywords can be converted directly as described below. It is assumed that there are two components, an "oil" component which is entirely in the oil phase at surface conditions, and a "gas" component that is entirely in the gas phase at surface conditions. The component names should be defined using the CNAMES keyword

Molecular weights

Molecular weights MW^c of each component should be specified. Suggested values are 120 for the oil component and 25 for the gas component.

Volatility types

The surface volatility types (keyword CVTYPES) should be set to DEAD for the "oil" component and GAS for the "gas" component to ensure the surface oil and gas phases contain only the "oil" and "gas" components.

If the blackoil simulation does not contain vaporized oil (PVDG) then the reservoir volatility type (keyword CVTYPE) can be set to DEAD for the "oil" component. Alternatively, a small K-value should be used for the "oil" component.

If the blackoil simulation does not contain dissolved gas (PVDO) then the reservoir volatility type (keyword CVTYPE) can be set to GAS for the "gas" component. Alternatively, a large K-value should be used for the "gas" component.

K-values for live components

K-values can be calculated from R_s and R_v values as functions of pressure:

$$K_{oil} = \frac{R_s + F}{1 + F \cdot R_v} \cdot R_v$$

$$K_{gas} = \frac{R_s + F}{1 + F \cdot R_v} \cdot \frac{1}{R_s}$$

where F is the ratio of the molar volumes of gas to oil at surface conditions:

$$F = \frac{V_{gas,s}}{V_{oil,s}} = \frac{MW^{gas}}{MW^{oil}} \cdot \frac{\rho_{oil,s}}{\rho_{gas,s}}$$

Values of $\rho_{oil,s}$ and $\rho_{gas,s}$ can be taken from the DENSITY keyword. The units of F should be the same as those used for R_s and $1/R_v$. In field units, a factor of 0.005615 can be used to convert ft^3/ft^3 to $Mscf/stb$.

A K-value table (keyword KVTABTn) at reservoir temperature can be constructed from the R_s and R_v data taken from the PVTO and PVTG keywords. At least one more K-value table is required at a different temperature. If no other data is available [EQ 69.40] can be used

$$K \propto e^{-D/T}$$

with values of D taken from Table 69.19 or Table 69.20 to give K-values at a second temperature:

$$K(T_2) = K(T_1) \cdot e^{D \cdot (1/T_1 - 1/T_2)}.$$

Oil phase density

The oil formation volume factor B_o is given by

$$B_o = \left(V_{oil,r}{}^{oil} + \frac{R_s}{F} \cdot V_{oil,r}{}^{gas} \right) / V_{oil,s} \qquad \text{[EQ 69.105]}$$

where $V_{oil,r}{}^{c}$ is the reservoir oil phase volume per mole of component c. Values of $V_{oil,r}{}^{c}$ are determined from user defined densities and compressibilites (see [EQ 69.45] and [EQ 69.46]), which can be chosen to give a good fit for [EQ 69.105] at the reservoir temperature. As a first step the component compressibility CREF and thermal expansion coefficient THERMEX1 can be ignored to get approximate values for the reference density DREF:

$$B_o = \frac{\rho_{oil,s}}{\rho_{ref}{}^{oil}} + R_s \cdot \frac{\rho_{gas,s}}{\rho_{ref}{}^{gas}} \qquad \text{[EQ 69.106]}$$

Gas phase density

The gas formation volume factor B_g is given by

$$B_g = (F \cdot R_v \cdot V_{gas,r}{}^{oil} + V_{gas,r}{}^{gas}) / V_{gas,s} \qquad \text{[EQ 69.107]}$$

where $V_{gas,r}{}^{c}$ is the reservoir gas phase volumes per mole of component c:

$$V_{gas,r}{}^{c} = \frac{Z^c \cdot R \cdot T}{P} \qquad \text{[EQ 69.108]}$$

where R is the gas constant, P is the pressure, T the temperature, and the compressibility terms Z^c can be set for each component using the ZFACTOR keyword to match [EQ 69.107] at the reservoir temperature. The temperature dependence of these terms are fixed. A surface compressibility can be set using the ZFACTORS keyword to ensure the correct surface gas density:

$$V_{gas,s} = \frac{Z^{gas} \cdot R \cdot T}{P} = \frac{MW^{gas}}{\rho_{gas,s}} \qquad \text{[EQ 69.109]}$$

Oil phase viscosity

The thermal oil phase viscosity is given by [EQ 69.82] as a function of temperature and mole fractions. It can be rewritten in terms of R_s to give:

$$\log\mu_{oil} = \frac{1}{F+R_s}\cdot(F\cdot\log\mu_{oil}{}^{oil} + R_s\cdot\log\mu_{oil}{}^{gas}) \qquad \text{[EQ 69.110]}$$

If necessary an extra pressure dependent factor can included in this formula. Values of the component oil phase viscosities $\mu_{oil}{}^{c}$ can be found at the reservoir temperature to give a good fit for [EQ 69.110]. These values can be entered in the OILVISCT keyword at two temperatures to give a temperature independent oil viscosity.

Gas phase viscosity

The thermal gas phase viscosity is given by [EQ 69.84] as a function of temperature and mole fractions. It can be rewritten in terms of R_v to give:

$$\mu_{gas} = \frac{1}{F\cdot R_v+1}\cdot(F\cdot R_v\cdot\mu_{gas}{}^{oil} + \mu_{gas}{}^{gas}) \qquad \text{[EQ 69.111]}$$

Values of the component gas phase viscosities $\mu_{gas}{}^{c}$ can be found at the reservoir temperature to give a good fit for [EQ 69.111]. These values can be entered in the GASVISCT keyword at two temperatures to give a temperature independent gas viscosity.

Specific heats

For an isothermal run, the actual values of the specific heats are not important. However, the enthalpy of each component in the gas phase should be larger than the enthalpy of the same component in the oil phase. This can be checked by looking at the property output in the PRT file.

Other data

The TCRIT and the PCRIT keywords must be included in the data set. There are no component-wise default values. The users are recommended not to default these to their internal default values.

Water properties

In thermal simulations the temperature dependence of water properties is usually more significant than the pressure dependence. Internal tables are used to define the water density and viscosity. To mimic a blackoil run keywords WATDENT and WATVISCT should be used to define the temperature dependence of the water density and viscosity while the pressure dependence is taken from the PVTW keyword.

Initial conditions

Data from a blackoil run can be used to initialize a thermal run. The keyword GETSOL can be used to read pressure and saturation data from a blackoil restart file. Keywords XMF, YMF have to be added.

If the EQUIL keyword is used item 10 should be set.

Initial compositions can be set by noting that at the saturation pressure

$$z_{oil} = \frac{F}{F + R_s} \quad z_{gas} = \frac{R_s}{F + R_s}$$ [EQ 69.112]

In thermal runs, water is allowed in the gas phase. This can cause differences in the initialization if a gas cap is present. If the initial composition is not in equilibrium then temperatures and saturations will change the first time the composition is flashed. It is therefore recommended that initial pressures, temperatures and saturations are output to the print file (RPTSOL) or restart file (RPTSCHED) and checked for consistency with the input data.

Well properties

The fluid enthalpy for any injection wells can be specified with the WINJTEMP keyword.

Well BHPs (keywords WCONINJE and WCONPROD) should be set, since the default values in the Thermal option are different from the default values used with Blackoil.

Example

An example is provided on the release CD. CASE3T is a thermal version of the black oil data set CASE3B.

Examples

The following example files come with the ECLIPSE installation:

Table 69.24 Thermal example files

.DATA File	Description	Auxiliary files
THERM1	Based on problem 1A of [Ref. 67] single well cyclic steam injection radial grid 1 component dead oil	
THERM2	Based on problem 2A of [Ref. 67] steam drive in 1/8th of a nine spot pattern 1 component dead oil	THERM2.GRF
THERM3	Based on problem 3A of [Ref. 67] steam drive in 1/8th of a nine spot pattern 3 component live oil	THERM3.GRF
THERM4	Variant of THERM1 (above) single well cyclic steam injection radial grid 2 component live oil equilibration with a gas cap mixed gas and steam injection	
THERM5	Based on case 1 of [Ref. 84] dual well SAGD 2 component live oil MSW	THERM5.GRF WELL5.TXT
THERM5A	As above, but with perfectly horizontal wells.	
THERM5B	As above, but with inclined wells.	
THERM6	Based on case 2 of [Ref. 84] single well SAGD 2 component live oil MSW	
THERM7	Based on case 3 of [Ref. 84] single well cyclic steam injection 3 component live oil MSW	
THERM8	Based on case 3 of [Ref. 84] single horizontal well, cyclic steam injection 2 component live oil heat loss from surface to reservoir MSW	

Table 69.24 Thermal example files

.DATA File	Description	Auxiliary files
THERM9	Based on [Ref. 54] laboratory scale combustion experiment 5 component live oil chemical reactions non-equilibrium initialization (NEI)	THERM9.GRF OBS9.TXT
THERM10	Foamy oil	THERM10.GRF
THERM10A	As above, but without the foamy oil terms	
THERM10B	As above, but with reactions replacing the flash	
THERM10C	Foamy solids	
THERM11	Based on [Ref. 54] and combustion with coke 5 component live oil solid phase chemical reactions	THERM11.GRF
THERM12	4 volatile components Heaters LGRs Corner point geometry Fast and flexible restarts	
THERM12R		
THERM12S		
THERM13A	Comparison of AIM, FULLIMP and IMPES runs. Data sets are based on THERM3 (problem 3A of [Ref. 67]), with 2000 cells and 9 wells.	THERM13.GRF
THERM13F		
THERM13I		
THERM14D	Variant of THERM1 (problem 3A of [Ref. 67]). Comparison of drift-flux and homogeneous flow models in a multi-segment well.	THERM14.GRF
THERM14H		
THERM15	Dual porosity and Joule-Thomson effect.	THERM15.GRF
THERM15J		
CASE3E100	Based on [Ref. 115]. Conversion of a blackoil data set to a thermal data set	CASE3T.GRF
CASE3B		
CASE3T		
HEAT100	Conversion of a blackoil Temperature option data set to a thermal data set	HEAT.GRF
HEAT300		
HEAT500		

Selected bibliography

A full list of references is given in the "Bibliography" on page 1741. References with particular relevance to thermal simulation are repeated below in alphabetical order of author.

Abbott, M.M. and van Ness, H.G. *Thermodynamics with Chemical Applications* [Ref. 1]
Schaum's Outline Series, 2nd Edition, McGraw Hill, 1989

Aziz, K., Ramesh, A. B. and Woo, P. T. *Fourth SPE Comparative Solution Project: Comparison of Steam Injection Simulators* [Ref. 2]
Journal of Petroleum Technology, *Page 1576-1584,* December 1987

Beattie, C.I., Boberg, T.C. and McNab, G.S. *Reservoir Simulation of Cyclic Steam Stimulation in the Cold Lake Oil Sands* [Ref. 3]
SPE 18752, SPE Reservoir Engineering, May 1991

Coats, K. H. *In-Situ Combustion Model* [Ref. 4]
SPE Journal, 533-554, , December 1980

Crookston, H. B., Culham, W. E. and Chen, W. H. *Numerical Simulation Model for Thermal Recovery Processes* [Ref. 5]
Soc. Pet. Eng. J., Trans. AIME, 267, *Page 37-58,* Feb. 1979

Edmister, W. C. and Lee, B. I. *Applied Hydrocarbon Thermodynamics, Volume 1* [Ref. 6]
Gulf Publishing Company, Second Edition, , , , , 1984

Ejiogu, G. C. and Fiori, M. *High-Pressure Saturated-Steam Correlations* [Ref. 7]
J. Pet. Tech. 39, 12, *Page 1585-1590,* 1987

Farouq Ali, S. M. *Oil Recovery by Steam Injection* [Ref. 8]
Producers Publishing Co., Bradford, PA, , 1970

Grabowski, J. W. and Rubin, B. *A Preliminary Numerical Simulation Study of In-situ Combustion in a Cold Lake Oil Sands Reservoir* [Ref. 9]
The Journal of Canadian Petroleum Technology (Montreal),

Page 79-89, April-June 1981, **Haywood, R. W.** *Thermodynamic Tables in SI (Metric) Units* [Ref. 10]
Cambridge University Press, Third Edition, 1990

Perry, R. H. and Green, D.W. *Perry's Chemical Engineers' Handbook* [Ref. 11]
McGraw Hill, Sixth Edition, 1984

Prats, M. *Thermal Recovery* [Ref. 12]
SPE Monograph 7, 1986

Reid, R.C., Prausnitz, J. M. and Polling, The Properties of Gases and Liquids
McGraw-Hill, 1987

B. E., Stone, T.W., Bennett, J., Law, D.H.-S., Holmes, J.A. *Thermal Simulation with Multisegment Wells* [Ref. 13]
SPE 78131, SPE Reservoir Evaluation & Engineering, *Page 206-218,* June 2002

Tortike, W. S. and Farouq Ali, S. M. *Saturated Steam Property Functional Correlations for Fully Implicit Thermal Reservoir Simulation* [Ref. 14]
SPEJ, 11, *Page 471-474,* 1989

Trangenstein, J. A. *Analysis of a Model and Sequential Numerical Method for Thermal Reservoir Simulation* [Ref. 15]
The Mathematics of Oil Recovery, Ed King, P.R., Clarendon Press, Oxford, 1992

Vinsome, P.K.W. and Westerveld, J. *A simple method for predicting cap and base rock heat losses in thermal reservoir simulators* [Ref. 16]
The Journal of Canadian Petroleum Technology, (Montreal), *Page 87-90,* July-September 1980

Wingard, J. S. *Multicomponent Multiphase Flow in Porous Media with Temperature Variation* [Ref. 17]
PhD thesis, Department of Petroleum Engineering, Stanford University, 1988

Timing Reports

Chapter 70

Introduction

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

The mnemonic CPU in the RPTSCHED keyword (switch number 12) controls the output of timing reports. If output is requested, reports such as that shown below are produced at each report time.

ECLIPSE 100 Timing Reports

```
=========================================================
TIMING REPORTS AT CPU=   558 SECS - ELAPSED=   621 SECS
=========================================================
ZS      1  2  3
               RR      1  2  2
INDX    1  0  0
PR      1  0  0
IR      1  0  0
NNC     1  0  0
XR      1  1  2
JR      1  0  0
YW     12  0  2
TS     11 89 86
               NX     41 88 82
                               WC   148  1  1
                                            WCDC  148  0  0
                               CONV 148  2  2
                               PAIR 148  0  0
                               LXX  107 68 63
                                            WGIN  107  0  0
                                            GINV  107  8  8
                                            WRES  107  0  0
                                            ITER  107 57 53
                                                          NORM  980  2  2
                                                          SRCH  873 34 32
                                                          WADX  873  0  0
                                                          ORTH  873 14 13
                                                          UPDA  873  5  5
                                            WSOL  107  0  0
                               XUIT 107  2  2
                               LC   107 15 14
               API    41  0  0
               TTRC   41  0  0
WWX    11  0  0
MWXR   11  2  2
XW     11  4  4
```

At the top of the report is a statement of the total CPU and elapsed time taken so far by the program. The remainder of the report shows how these times were split between various sections of the program. Each program section is identified by a 4 character mnemonic, the most important of which are:

- `RR` input of the `GRID` section
- `PR` input of the `PROPS` section
- `XR` input of the `SOLUTION` section and equilibration
- `TS` advancing simulator through time
- `NX` solution of non-linear equations by Newton's method
- `WC` setting up well terms in coefficient matrix
- `CONV` checking convergence of non-linear equations
- `PAIR` pair method (experimental)

`MJAC` final assembly of Jacobian matrix

`LXX` solution of the linear equations by Nested Factorization

`GINV` linear solver initialization

`ITER` linear solver iteration

`NORM` linear solver norms (residual max. sums and squares)

`SRCH` preconditioning and matrix multiply

`ORTH` orthogonalization

`UPDA` updating solution and checking linear solver convergence

`XUIT` state changes, checks on physical reasonableness

`LC` setting up coefficient matrix (includes property look-up)

`API` updating API variables

`TTRC` updating tracer tracking variables

`WWX` output of well reports

`MWXR` output of fluid in place reports

`XW` output of printed tables of pressures, saturations etc.

`HMO` calculation of gradients for the Gradient option

Often, a section is split into smaller sections that are contained entirely within it. For example, NX is split into WC, CONV, MJAC, LXX, XUIT and LC. This structure is shown by the indentation in the above report.

The three numbers given by each section mnemonic are:

- The number of times the section was entered
- The percentage of the total CPU time spent in the section
- The percentage of the total elapsed time spent in each section.

Thus section LXX was entered 107 times; it accounted for 68% of CPU time and 63% of elapsed time. If there were no time-step chops then the number following NX (41) is the number of time steps so far. Thus the average number of non-linear updates per time step is 107/41 = 2.61. Non-linear convergence was checked 107+41 = 148 times (CONV). The average number of iterations to solve the linear equations was 873/107 = 8.16. 34% of the CPU time was devoted to solving the preconditioning equations by nested factorization (SRCH). Orthogonalization of each new search direction to the stack of previous search directions (ORTH) required 14% of the CPU time and 13% of the elapsed time. The time spent in ORTH and UPDA may increase **dramatically** if you are forced to use a negative or zero `NSTACK`. Always make `NSTACK` positive when possible.

Data inside TS is not recorded when while the CPU mnemonic is absent in `RPTSCHED` (or when switch 12 is set to zero). Thus if it is specified just before the final time step, you will get a report with reliable total CPU and elapsed times but with an incorrect breakdown of the details. Switching the internal clocks off in this way can save a few percent of the total computing bill.

ECLIPSE 300 Timing Reports

```
 ==========================================================
 TIMING REPORTS AT CPU=    52 SECS - ELAPSED=    60 SECS
 ==========================================================
--XUAIM                  1  0.0 %cpu  0.1 %elp
--LSPUT4                 3  0.0 %cpu  0.1 %elp
--TS                     3 98.7 %cpu 92.7 %elp
----AIM                117  2.1 %cpu  1.8 %elp
------XUAIM            117  0.0 %cpu  0.0 %elp
------LSPUT4           117  0.3 %cpu  0.3 %elp
----NLSTEP             117 93.0 %cpu 83.5 %elp
------DCSP             234 46.2 %cpu 40.3 %elp
--------DCED1A         236 28.2 %cpu 24.5 %elp
--------XSWAPE         236  0.0 %cpu  0.0 %elp
--------DCED1B         236  6.3 %cpu  5.5 %elp
------WSOLVS           234 15.7 %cpu 14.6 %elp
------NLRESD           234  2.1 %cpu  1.8 %elp
------XCNEWD           234 26.5 %cpu 23.5 %elp
--------XCMATV         234  7.2 %cpu  6.3 %elp
--------ELIM           234  2.1 %cpu  1.9 %elp
--------LSSET4         234  0.2 %cpu  0.2 %elp
--------LSOLV          234 13.0 %cpu 11.4 %elp
----------LWARP        234 13.0 %cpu 11.4 %elp
------------LGFAC      234  2.2 %cpu  2.1 %elp
------------LGPGMR     234 10.2 %cpu  8.8 %elp
--------------LGFBS   1877  6.0 %cpu  5.2 %elp
--------------LGMLT   1877  2.1 %cpu  1.8 %elp
--------BACKSU         234  1.8 %cpu  1.6 %elp
--------XLIMIT         234  0.2 %cpu  0.2 %elp
------NLUPDT           234  0.1 %cpu  0.2 %elp
----PREDA              585  0.0 %cpu  0.0 %elp
----PSUMR               21  0.0 %cpu  0.0 %elp
```

Again, as with the ECLIPSE 100 timing reports, the top of the report contains a statement which details the total CPU and elapsed time taken so far by the program. The remainder of the report shows how these times were split between various sections of the program. Each program section is identified by a 6 character mnemonic, the most important of which are

`AIM` Adaptive IMplicit method

`AJGRAD` Linear solution for adjoint vector

`DCSP` Main domain calculation

`GHMCAL` Main routine for gradient calculation

`GSOLVS` Supervises the gradient calculations at the end of a time step

`LSOLV` WARP linear solver

`LWARP` WARP linear solver

`NLCONV` Check for convergence of the non-linear equations solved using Newton's method

`NLRESD` Set up the residual for the Newton system

`NLSTEP` Solution of the non-linear equations by Newton's method

`NLTRAC` Solve for tracer tracking variables

`NLUPDT` Update the solution variables during a Newton iteration

`SAMG` Algebraic multi-grid solver used for geomechanics

`TS` Advances the simulator through time to the next report step

`WSOLVS` Solve the production and injection well system

`XCFLOW` Calculate the flows

`XCMATV` Set up the Jacobian matrix

`XLIMIT` Limit the solution changes during a Newton iteration

`XUAIM` Set up the Adaptive IMplicit (AIM) indicator array

Total Compressibility Checks

Chapter 71

Introduction

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

In black oil models, ECLIPSE 100 and ECLIPSE 300 check for positive compressibility of each single reservoir fluid as the PVT data is read (the formation volume factor must be a monotonically decreasing function of pressure assuming all other variables are held fixed).

However, ECLIPSE also checks that oil-gas mixtures have a positive total compressibility even when there is mass transfer between the two phases. For example, it is well known that oil sometimes swells when the pressure increases, apparently in violation of physical principles. This swelling occurs because gas is dissolving in the oil. Provided that the reduction in the volume of gas is greater than the increase in the volume of oil, the total volume of the oil-gas mixture reduces as the pressure increases, and the paradox is resolved.

It is possible to calculate a value for the total compressibility of a hydrocarbon system where mass exchange may occur between the reservoir liquid and vapor phases. Consider a volume V_g of separator gas and a volume V_o of stock tank oil at surface conditions. Compress the mixture to pressure P, volume V and assume that a two-phase equilibrium state is formed. The material balance equations for the oil and gas components are

$$V_g = \frac{V \cdot S_g}{B_g} + \frac{V \cdot R_s \cdot S_o}{B_o} \qquad \text{[EQ 71.1]}$$

$$V_o = \frac{V \cdot S_o}{B_o} + \frac{V \cdot R_v \cdot S_g}{B_g}. \qquad \text{[EQ 71.2]}$$

[EQ 71.1] and [EQ 71.2] allow the mixture volume V and gas saturation S_g to be computed. If the pressure of the system is increased, the volume occupied by the liquid-vapor system must decrease for physically well-defined fluids. The total compressibility of the system, $C_t = -(1/V)(dV/dP)$, can be shown to be given by the following expression

$$C_t = \frac{S_g}{B_g}\left(-\frac{dB_g}{dP} + \frac{dR_v}{dP} \cdot \frac{B_o - R_s B_g}{1 - R_s R_v}\right) + \frac{S_o}{B_o}\left(-\frac{dB_o}{dP} + \frac{dR_s}{dP} \cdot \frac{B_g - R_v B_o}{1 - R_s R_v}\right) \qquad \text{[EQ 71.3]}$$

where the pressure derivatives are taken along the saturation curve.

This can also be written as

$$C_t = S_g C_{t,gas} + S_o C_{t,oil} \qquad \text{[EQ 71.4]}$$

which splits the total compressibility into gas and oil dependent terms.

To derive this expression, we assume without loss of generality that

$$S_o + S_g = 1 \qquad \text{[EQ 71.5]}$$

and note that

$$dV_o/dp = dV_g/dp = 0 \qquad \text{[EQ 71.6]}$$

since the volumes of stock tank oil and separator gas are fixed.

Solving for S_o and S_g gives

$$S_o = \frac{R_v V_g - V_o}{V} \cdot \frac{B_o}{R_s R_v - 1} \qquad \text{[EQ 71.7]}$$

and

$$S_g = \frac{R_s V_o - V_g}{V} \cdot \frac{B_g}{R_s R_v - 1} \qquad \text{[EQ 71.8]}$$

And since $S_o + S_g = 1$ we have

$$V = \frac{1}{(R_s R_v - 1)} \cdot (V_o(R_s B_g - B_o) + V_g(R_v B_o - B_g)) \qquad \text{[EQ 71.9]}$$

Hence

$$C_t = -\frac{1}{V} \cdot \frac{dV}{dp} \qquad \text{[EQ 71.10]}$$

$$= -\frac{(R_s dR_v/dp + R_v dR_s/dp)}{(R_s R_v - 1)} + \frac{\left\{ V_o\left[R_s \frac{dB_g}{dp} + B_g \frac{dR_s}{dp} - \frac{dB_o}{dp}\right] + V_g\left[R_v \frac{dB_o}{dp} + B_o \frac{dR_v}{dp} - \frac{dB_g}{dp}\right]\right\}}{V_o(R_s B_g - B_o) + V_g(R_v B_o - B_g)}$$

which after some manipulation, and substitution for V_o and V_g using the above relations, leads to the desired expression for C_t.

Some simple limiting cases may be derived from [EQ 71.3].

- If $R_s = R_v = 0$ then

  $$C_t = -\frac{S_g}{B_g} \cdot \frac{dB_g}{dP} - \frac{S_o}{B_o} \cdot \frac{dB_o}{dP} > 0 \qquad \text{[EQ 71.11]}$$

  since the formation volume factor pressure derivatives are < 0.

- If $R_s > 0$ and $R_v = 0$ then

$$C_t = -\frac{S_g}{B_g} \cdot \frac{dB_g}{dP} - \frac{S_o}{B_o} \cdot \left(\frac{dB_o}{dP} - B_g \frac{dR_s}{dP}\right) \qquad \text{[EQ 71.12]}$$

so that $C_t < 0$ if $dB_o/dP > B_g dR_s/dP$ and S_g is sufficiently small.

ECLIPSE checks the total compressibility of the hydrocarbon PVT data as it is read from the input data file. For each PVT region in the simulation grid a pressure range is selected from the corresponding oil and gas PVT data tables to span the complete range of pressure data in the two tables. This pressure range is then subdivided into 30 equally spaced pressure nodes for the evaluation of the total hydrocarbon compressibility using [EQ 71.3].

The sample pressure range is the maximum of the oil pressure (maximum bubble point pressure in PVTO or maximum pressure specified in PVDO) and the gas pressure (maximum dew point pressure specified in PVTG or maximum gas pressure specified in PVDG).

ECLIPSE 100

The maximum sample pressure at which total compressibility is checked can be overridden by choosing the second item in the PMAX keyword. This can be used to check for negative compressibility in extrapolated regions.

At each pressure node two limiting compressibility values are calculated corresponding to $S_g = 0$ and $S_g = 1$ in [EQ 71.3]. If either or both values should be negative, a warning message will be issued indicating the offending pressure value and the gas saturation range for which the compressibility is negative. It should then be possible using [EQ 71.3] to determine how to change the saturated gas/oil PVT data to ensure the total compressibility remains positive for all pressures.

If the range of sample pressures extends above the maximum bubble point specified in the PVTO table or above the maximum dew point specified in the PVTG table, then ECLIPSE is forced to extrapolate above the highest entered bubble point or dew point. This extrapolation is linear in $1/B_o$ and $1/(B_o \mu_o)$ etc. (see the PVTO keyword). In this case, it is not unlikely that negative compressibilities could occur as a result of extrapolation.

In some cases, if the pressure is extrapolated severely, then the linearly-extrapolated formation volume factors may become unphysical. It is recommended that the highest bubble point nodes in the PVTO table are constructed so avoid extrapolations above the highest entered Rs in the PVTO table during the simulation (similarly, in a run with vaporized oil present, it is recommended that the highest dew point nodes in the PVTG table are constructed to avoid extrapolations above the maximum entered P_{dew} during the simulation).

ECLIPSE 100

The EXTRAPMS keyword can be used to instruct ECLIPSE 100 to report PVT table extrapolations during the simulation.

ECLIPSE 100

If the 21st switch in the DEBUG keyword is set > 0, ECLIPSE 100 reports the total gas and oil compressibilities $C_{t,gas}$ and $C_{t,oil}$ as well as $R_{s,sat}$, $B_{o,sat}$ and $R_{v,sat}$, $B_{g,sat}$ for the sample range of pressures to the debug file (extension .DBG) in tabular form. Whenever any negative compressibilities are encountered in the data checking, this debug output is automatically switched on. This may then be used to diagnose any negative compressibilities reported by ECLIPSE 100.

Note The effect of a negative total compressibility is usually to cause the simulator to experience numerical difficulties (convergence failures and/or erratic changes in the solution) in regions where the total compressibility is negative.

Transmissibility Calculations

Chapter 72

Introduction

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

This chapter describes the expressions used by ECLIPSE to calculate transmissibility values, between cells in the grid, and between cells and numerical aquifers.

The transmissibility calculations for each type of geometry are detailed in the next section. In the corner point case, transmissibility values between cells that are not neighbors in the basic indexing grid may be calculated on the same basis as those for neighboring cells. In both cases, the mutual interface area is calculated, and this may be non-zero for non-neighboring cells at faults.

Cell ordering and geometry

The basic grid indexing system is by natural order. Cells may be identified by their (i ,j, k) indices in the cartesian indexing grid, and in natural ordering, the i index changes most quickly, the k index most slowly. Internally, cells are held by active order, with any inactive cells (that is, those with zero pore volume) not requiring any storage.

Details of cell geometry may be entered using keywords such as DX, DY and DZ to specify cell sizes. This is a block center description, and cells are assumed to have a simple, regular shape.

For example, the pore volume of each cell is then $DX \cdot DY \cdot DZ \cdot \phi \cdot RNTG$, where ϕ is the porosity, $RNTG$ the net to gross ratio, and DX, DY and DZ the cell dimensions.

An alternative is the entry of cell corner points using the COORD and ZCORN keywords. The data associated with these keywords is usually prepared using a pre-processor such as PETREL, and entered from an included file. The cell is assumed to have edges defined as straight lines between the corner points, and the cell faces may be bilinear surfaces. Pore volumes may be calculated exactly for such a shape.

It is possible to have more than one set of COORD data, using the NUMRES keyword in the RUNSPEC section, and COORDSYS to assign layers to these sets.

Transmissibility calculations

There are at most three types of transmissibility calculation available in ECLIPSE.

The first is the traditional block centred form, based upon the distances between the cell centers only. This is the default calculation when DX, DY, DZ are specified.

Note For ECLIPSE 100 only, this corresponds to the use of the OLDTRAN keyword in the GRID section.

ECLIPSE 100

The second form also assumes a traditional block centered geometry but involves a different combination of cell cross-sectional areas and permeabilities from that assumed in the OLDTRAN formulation. The second form of transmissibility calculation is activated using the OLDTRANR keyword in the GRID section.

The third form is based upon the use of cell corner points, which are available to ECLIPSE when the COORD/ZCORN form of grid definition is used. In this case, it is possible to distinguish unambiguously between cell dip and fault displacement, and this option allows fault transmissibilities to be generated automatically.

Note For ECLIPSE 100 use, this third type of transmissibility calculation is specified by the use of the NEWTRAN keyword in the GRID section. This is the default calculation when COORD/ZCORN are specified.

ECLIPSE 100

Although it is normal to use OLDTRAN with DX, DY, DZ, TOPS etc. input, and NEWTRAN with COORD/ZCORN, all types of transmissibility calculation are available with both types of grid specification, as ECLIPSE holds grid data internally in both forms. Corner points are obtained from DX/DY/DZ/TOPS data by assuming flat blocks, and DX/DY/DZ values are obtained from corner points by evaluation of the distances between cell face center points. However, use of NEWTRAN with DX/DY/DZ/TOPS input may well produce large numbers of spurious fault connections, as the assumed flat grid blocks in neighboring columns will generally overlap, and this combination of options is not recommended.

Note If the grid has shales modeled as gaps in grid, the keyword MULTZ- must be specified.

Cartesian transmissibility calculations

Block center transmissibility calculations

In this case, X- and Y-direction transmissibility values in the cartesian case are obtained using cell center separations and cross sectional areas obtained from `DX`, `DY` and `DZ`, with a dip correction.

ECLIPSE 100

Note For ECLIPSE 100, this type of calculation is also referred to as `OLDTRAN`.

X- transmissibility

$$\mathrm{TRANX}_i = \frac{\mathrm{CDARCY} \cdot \mathrm{TMLTX}_i \cdot A \cdot \mathrm{DIPC}}{B} \qquad \text{[EQ 72.1]}$$

where

TRANX_i Transmissibility between cell i and cell j, its neighbor in the positive X-direction

CDARCY Darcy's constant

= 0.00852702 (E300); 0.008527 (E100) (`METRIC`)

= 0.00112712 (E300); 0.001127 (E100) (`FIELD`)

= 3.6 (`LAB`)

= 0.00864 (`PVT-M`)

TMLTX_i Transmissibility multiplier for cell i

A Interface area between cell i and j

DIPC Dip correction.

A, DIPC and B are given by the expressions:

$$A = \frac{\mathrm{DX}_j \cdot \mathrm{DY}_i \cdot \mathrm{DZ}_i \cdot \mathrm{RNTG}_i + \mathrm{DX}_i \cdot \mathrm{DY}_j \cdot \mathrm{DZ}_j \cdot \mathrm{RNTG}_j}{\mathrm{DX}_i + \mathrm{DX}_j} \qquad \text{[EQ 72.2]}$$

$$B = \frac{\left(\frac{\mathrm{DX}_i}{\mathrm{PERMX}_i} + \frac{\mathrm{DX}_j}{\mathrm{PERMX}_j}\right)}{2} \qquad \text{[EQ 72.3]}$$

$$\mathrm{DIPC} = \frac{\mathrm{DHS}}{\mathrm{DHS} + \mathrm{DVS}} \qquad \text{[EQ 72.4]}$$

with

$$\mathrm{DHS} = \left(\frac{\mathrm{DX}_i + \mathrm{DX}_j}{2}\right)^2$$

and

$$\mathrm{DVS} = [\mathrm{DEPTH}_i - \mathrm{DEPTH}_j]^2 .$$

RNTG is the net to gross ratio, which appears in the X- and Y- transmissibilities but not in the Z- transmissibility.

Y- transmissibility

The expression for the Y-transmissibility value is entirely analogous to the above, with the appropriate permutations of X, Y and Z.

Z- transmissibility

The Z-transmissibility is given by:

$$\mathrm{TRANZ}_i = \frac{\mathrm{CDARCY} \cdot \mathrm{TMLTZ}_i \cdot A}{B} \qquad \text{[EQ 72.5]}$$

where

TRANZ_i Transmissibility between cell i and cell j, its neighbor in the positive Z-direction (that is below cell i)

TMLTZ_i The Z-transmissibility multiplier for cell i

and

$$A = \frac{\mathrm{DZ}_j \cdot \mathrm{DX}_i \cdot \mathrm{DY}_i + \mathrm{DZ}_i \cdot \mathrm{DX}_j \cdot \mathrm{DY}_j}{\mathrm{DZ}_i + \mathrm{DZ}_j} \qquad \text{[EQ 72.6]}$$

$$B = \frac{\left(\frac{\mathrm{DZ}_i}{\mathrm{PERMZ}_i} + \frac{\mathrm{DZ}_j}{\mathrm{PERMZ}_j}\right)}{2}. \qquad \text{[EQ 72.7]}$$

Note that there is no dip correction in the Z-transmissibility case.

OLDTRANR transmissibility calculations

ECLIPSE 100

In this case, X- and Y-direction transmissibility values in the cartesian case are also calculated using cell center separations and cross-sectional areas obtained from `DX`, `DY` and `DZ`, with a dip correction.

The expressions for the cartesian case are given below:

X- transmissibility

$$\mathrm{TRANX}_i = \frac{\mathrm{CDARCY} \cdot \mathrm{TMLTX}_i \cdot \mathrm{DIPC}}{B} \qquad \text{[EQ 72.8]}$$

where

TRANX_i Transmissibility between cell i and cell j, its neighbor in the positive X-direction

CDARCY Darcy's constant (in the appropriate units)

TMLTX_i Transmissibility multiplier for cell i

DIPC Dip correction.

B and DIPC are given by the expressions:

$$B = \frac{\left(\frac{DX_i}{A_i \cdot PERMX_i} + \frac{DXj}{A_j \cdot PERMX_j}\right)}{2} \qquad \text{[EQ 72.9]}$$

where

$A_i = DY_i \cdot DZ_i \cdot NTG_i$

and

$A_j = DY_j \cdot DZ_j \cdot NTG_j$

with

$$DIPC = \frac{DHS}{DHS + DVS} \qquad \text{[EQ 72.10]}$$

$$DHS = \left(\frac{DX_i + DX_j}{2}\right)^2$$

and

$DVS = [DEPTH_i - DEPTH_j]^2$.

RNTG is the net to gross ratio, which appears in the X- and Y-transmissibilities but not in the Z-transmissibility.

Y- transmissibility

The expression for the Y-transmissibility value is entirely analogous to the above, with the appropriate permutations of X,Y and Z.

Z- transmissibility

The Z-transmissibility is given by:

$$TRANZ_i = \frac{CDARCY \cdot TMLTZ_i}{B} \qquad \text{[EQ 72.11]}$$

where

$TRANZ_i$ Transmissibility between cell i and cell j, its neighbor in the positive Z-direction (that is below cell i)

$TMLTZ_i$ The Z-transmissibility multiplier for cell i

$$B = \frac{\left(\frac{DZ_i}{A_i \cdot PERMZ_i} + \frac{DZ_j}{A_j \cdot PERMZ_j}\right)}{2} \qquad \text{[EQ 72.12]}$$

with

$A_i = DX_i \cdot DY_i$
$A_j = DX_j \cdot DY_j$

Note that there is no dip correction in the Z-transmissibility case.

Corner point transmissibility calculations

In this case, the transmissibility values are calculated from the X-,Y- and Z-projections of the mutual interface area of the two cells.

An inner product is then taken with the vector distance from the cell center to the center of the cell face, so that a dip correction is automatically incorporated.

ECLIPSE 100

Note For ECLIPSE 100, this type of calculation is also referred to as `NEWTRAN`.

X- transmissibility

The X-transmissibility is given by the expression:

$$\mathrm{TRANX}_i = \frac{\mathrm{CDARCY} \cdot \mathrm{TMLTX}_i}{\frac{1}{T_i} + \frac{1}{T_j}} \qquad \text{[EQ 72.13]}$$

where

$$T_i = \mathrm{PERMX}_i \cdot \mathrm{RNTG}_i \cdot \frac{A \cdot D_i}{D_i \cdot D_i}$$

with

$$(A \cdot D_i) = A_X \cdot D_{iX} + A_Y \cdot D_{iY} + A_Z \cdot D_{iZ}$$

and

$$(D_i \cdot D_i) = D_{iX}^2 + D_{iY}^2 + D_{iZ}^2 .$$

A_X, A_Y and A_Z are the X-,Y- and Z- projections of the mutual interface area of cell i and cell j (which need not be neighbors in the cartesian indexing grid), and D_{iX}, D_{iY} and D_{iZ} are the X-, Y- and Z-components of the distance between the center of cell i and the center of the relevant face of cell i, these centers being obtained as the appropriate average.

The expression for T_j is analogous.

Y- and Z- transmissibility

The Y- and Z-transmissibility expressions are similar, the net to gross ratio being absent from the Z expression.

The calculation is illustrated by the following figure (where `PERMX` is represented by Kx):

Figure 72.1 Y- and Z- transmissibility expressions

HALFTRAN transmissibility calculations

ECLIPSE 100

By observing that the transmissibility between cells i and j in the NEWTRAN formulation is composed of a harmonic average between half block transmissibilities T_i and T_j, we have rather more freedom than just using a single permeability value for each cell.

In fact, there are six possible permeability values associated with each cell in the half block transmissibilities. For instance, in cell i, half block x-direction permeabilities Kxil, Kxir contribute to the two x-direction transmissibilities affecting cell i. Similarly in the Y- and Z-directions we can use Kyil, Kyir and Kzil, Kzir.

The use of extra half block permeabilities provides a more accurate upscaling of permeability while retaining the same computational grid.

Within ECLIPSE 100, the HALFTRAN keyword activates the half block transmissibility calculation.

By writing the half block transmissibilities as

$$\bar{T}_{ir} = Kx_{ir} \cdot \mathrm{RNTG}_i \cdot \frac{A \cdot D_i}{D_i \cdot D_i} \qquad \text{[EQ 72.14]}$$

and

$$\bar{T}_{jl} = Kx_{jl} \cdot \mathrm{RNTG}_j \cdot \frac{A \cdot D_j}{D_j \cdot D_j}, \qquad \text{[EQ 72.15]}$$

it is apparent that in terms of the normal NEWTRAN calculation, then

$$\bar{T}_{ir} = \lambda_{ir} T_i$$

and

$$\bar{T}_{jl} = \lambda_{jl} T_j$$

with

$$\lambda_{ir} = Kx_{ir}/Kx_i \text{ and } \lambda_{jl} = Kx_{jl}/Kx_j .$$

When HALFTRAN is used, ECLIPSE 100 requires the MULTX and MULTX- values to be supplied equal to λ_{ir} and λ_{jl}.

At present, the FloGrid program carries out the half-block upscaling and present the modified MULTX and MULTX- data to the simulator in order to effect the half block transmissibility calculation. The PERMX etc. values should be entered as normal - the half block permeabilities are then obtained by multiplying PERMX by MULTX or MULTX- etc. The connection factors are calculated as usual using Peaceman's formula for the block center permeability PERMX etc.

Radial transmissibility calculations

There is no distinction between the method used to calculate radial transmissibility values whether grid data is supplied in `DR/DTHETA/DZ/INRAD/TOPS` form or `COORD/ZCORN` form.

The expressions used are based upon true radial flow between the pressure equivalent radii, with a dip correction.

R-transmissibility

$$\mathrm{TRANR}_i = \frac{\mathrm{CDARCY} \cdot \mathrm{TMLTR}_i \cdot \mathrm{DIPC}}{\frac{1}{T_i} + \frac{1}{T_j}} \qquad \text{[EQ 72.16]}$$

with

$$T_i = \frac{\mathrm{PERMR}_i \cdot \mathrm{RNTG}_i \cdot D\theta_i \cdot \mathrm{DZ}_i}{D_{1P}}$$

$$D_{1P} = \frac{R_1^2}{(R_2^2 - R_1^2)} \ln(R_1 / R_2) + \frac{1}{2}$$

and

$$T_j = \frac{\mathrm{PERMR}_j \cdot \mathrm{RNTG}_j \cdot \mathrm{D}\theta_j \cdot \mathrm{DZ}_j}{D_{2M}}$$

$$D_{2M} = \frac{R_3^2}{(R_3^2 - R_2^2)} \ln(R_3 / R_2) - \frac{1}{2}$$

where

R_1 is the inner radius of cell i,

R_2 is the outer radius of cell i,

R_3 is the outer radius of cell j.

Dip correction

The dip correction is given by:

$$DIPC = \frac{DHS}{DHS + DVS}$$

where

$$DHS = \left(\frac{1}{2}(R_3 - R_1)\right)^2$$

and

$$DVS = (\mathrm{DEPTH}_i - \mathrm{DEPTH}_j)^2 .$$

Azimuthal transmissibility

The azimuthal transmissibility is given by:

$$\mathrm{TRANT}_i = \frac{CDARCY \cdot TMLTT_i \cdot DIPC}{\frac{1}{T_i} + \frac{1}{T_j}} \qquad \text{[EQ 72.17]}$$

with

$$T_i = \frac{2 \cdot \mathrm{PERMT}_i \cdot \mathrm{RNTG}_i \cdot \mathrm{DZ}_i \cdot \ln(R_2/R_1)}{\mathrm{D}\theta_i}$$

where

R_1 is the inner radius,

R_2 is the outer radius.

T_j is similar.

It should be noted that ln() implies the natural logarithm throughout. This expression is also used in completing the circle in radial geometry.

Dip correction

The dip correction is given by:

$$DIPC = \frac{DHS}{DHS + DVS}$$

where

$$DHS = \left(\frac{1}{2}(D\theta_i + D\theta_j) \cdot \mathrm{R}_{\mathrm{mean}}\right)^2$$

and

$$DVS = (\mathrm{DEPTH}_i - \mathrm{DEPTH}_j)^2 .$$

For ECLIPSE 100, $\mathrm{R}_{\mathrm{mean}}$ takes a value of one, and for ECLIPSE 300, $\mathrm{R}_{\mathrm{mean}}$ represents an average of the inner and outer radius.

Vertical transmissibility

The vertical transmissibility is given by:

$$\mathrm{TRANZ}_i = \frac{\mathrm{CDARCY} \cdot \mathrm{TMLTZ}_i}{\frac{1}{T_i} + \frac{1}{T_j}} \qquad \text{[EQ 72.18]}$$

with

$$T_i = \frac{\mathrm{PERMZ}_i \cdot \mathrm{D}\theta_i \cdot (R_2^2 - R_1^2)}{\mathrm{DZ}_i}$$

where

R_1 is the inner radius,

R_2 is the outer radius.

T_j is similar.

Note that the positions of the cells in a radial run may be entered by using COORD/ZCORN or INRAD/DR/DTHETA/DZ.

In both cases the transmissibility expressions above may be used. There are some restrictions that exist on corner point positions for the above expressions to be sensible. Cells with the same j index should have a common DTHETA, and cells with the same i index should have a common DR. If COORD/ZCORN input is used, four corners of each cell should have a common inner radius, and four a common outer radius.

Similarly, there should be a single upper and lower theta value. (This implies vertical coordinate lines in the radial case). The depths of cell corners are, however, arbitrary, and may be used to obtain dipping radial geometry.

Variations on the transmissibility calculation

ECLIPSE 100

The transmissibility calculation using either OLDTRAN, OLDTRANR or NEWTRAN employs an effective average permeability for the face between two adjacent blocks, which ECLIPSE 100 calculates from the permeabilities and dimensions of these blocks.

Options are available, however, that allow you more control over the average permeability employed in these calculations. You may input the average permeabilities directly (see keyword LINKPERM), or request alternative averaging formulations (see keyword PERMAVE).

The options LINKPERM and PERMAVE may be useful for comparison with other simulators. It is also possible to override the transmissibilities calculated by ECLIPSE 100 by supplying transmissibilities directly in the EDIT section, for comparison with other simulators.

Use of linking permeabilities in the transmissibility calculation

ECLIPSE 100

The LINKPERM keyword may optionally be used to instruct ECLIPSE 100 to treat the grid block permeabilities (input with PERMX etc.) as applying to the cell faces instead of cell centers.

ECLIPSE 100 then uses the face permeabilities directly as the “linking permeabilities” in the transmissibility calculation, instead of calculating the linking permeabilities as an average of the adjacent grid block permeabilities.

For example, if PERMX_l is chosen to represent the linking permeability in the X- direction between connecting cells I and J, then PERMX_l simply replaces both PERMX_i and PERMX_j in the above expressions for the transmissibility in each of the OLDTRAN, OLDTRANR and NEWTRAN cases. The PERMX_l values will be input as either PERMX_i or PERMX_j according to the option selected in LINKPERM.

Note The grid block values of PERMX, PERMY and PERMZ are required to calculate well connection factors (see "The connection transmissibility factor" on page 1189). If linking permeabilities are used to apply PERMX etc. to block faces, the input values may be inappropriate for calculating connection factors. In this case, connection factors could be set explicitly in COMPDAT or connection factor multipliers could be applied using WPIMULT.

Alternative permeability averaging schemes

ECLIPSE 100

The PERMAVE keyword may optionally be used to select alternative formulations for calculating the average permeability for the cell faces in the transmissibility calculation. The different formulations can best be compared by considering the formulae employed above in the default cases (when neither LINKPERM nor PERMAVE is specified).

Firstly, consider the OLDTRAN case. The X-transmissibility calculation outlined above can be expressed equivalently as:

$$\mathrm{TRANX}_i = \frac{\mathrm{CDARCY} \cdot \mathrm{TMLTX}_i \cdot A \cdot \mathrm{DIPC}}{B} \qquad \text{[EQ 72.19]}$$

with B rewritten as:

$$B = \frac{\mathrm{DX}_i + \mathrm{DX}_j}{2 \cdot \mathrm{PERMX}_{\mathrm{av}}}, \qquad \text{[EQ 72.20]}$$

and the average permeability is given by a harmonic average:

$$\mathrm{PERMX}_{\mathrm{av}} = \frac{(W_i + W_j)}{\left(\frac{W_i}{\mathrm{PERMX}_i} + \frac{W_j}{\mathrm{PERMX}_j}\right)} \qquad \text{[EQ 72.21]}$$

with weights

$$\begin{aligned} W_i &= \mathrm{DX}_i \\ W_j &= \mathrm{DX}_j \end{aligned} \qquad \text{[EQ 72.22]}$$

This is just a special case of the power law average given by:

$$\mathrm{PERMX}_{\mathrm{av}} = \left[\frac{W_i \cdot \mathrm{PERMX}_i^{\mathrm{POW}} + W_j \cdot \mathrm{PERMX}_j^{\mathrm{POW}}}{W_i + W_j}\right]^{\frac{1}{\mathrm{POW}}} \qquad \text{[EQ 72.23]}$$

where $\mathrm{POW} = -1$.

Other forms of average may be obtained by varying the value of POW. A weighted arithmetic average is obtained if $\mathrm{POW} = 1$, and a geometric average results in the limit as $\mathrm{POW} \rightarrow 0$, that is

$$\mathrm{PERMX}_{\mathrm{av}} = [\mathrm{PERMX}_i^{W_i} \cdot \mathrm{PERMX}_j^{W_j}]^{1/(W_i + W_j)}. \qquad \text{[EQ 72.24]}$$

Secondly, consider the `OLDTRANR` case. The formulation for `OLDTRANR` is analogous with `OLDTRAN`, but now the weights are given by

$$\begin{aligned} W_i &= \frac{\mathrm{DX}_i}{A_i} = \frac{\mathrm{DX}_i}{\mathrm{DY}_i \cdot \mathrm{DZ}_i \cdot \mathrm{RNTG}_i} \\ W_j &= \frac{\mathrm{DX}_j}{A_j} = \frac{\mathrm{DX}_j}{\mathrm{DY}_j \cdot \mathrm{DZ}_j \cdot \mathrm{RNTG}_j} \end{aligned} \qquad \text{[EQ 72.25]}$$

Finally, consider the `NEWTRAN` case. The weights corresponding to the default harmonic average of block permeabilities are given by:

$$\begin{aligned} \frac{1}{W_i} &= \mathrm{RNTG}_i \cdot \frac{A \cdot D_i}{D_i \cdot D_i} \\ \frac{1}{W_j} &= \mathrm{RNTG}_j \cdot \frac{A \cdot D_j}{D_j \cdot D_j} \end{aligned} \qquad \text{[EQ 72.26]}$$

and the alternative permeability averages are readily expressed in the same manner as above.

Tracer Tracking

Chapter 73

Introduction

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

The ECLIPSE Tracer Tracking option is a general facility to follow the movement of 'marked' fluid elements during a simulation run.

The Tracer Tracking option has a wide variety of reservoir modeling applications. In the case of tracers defined to exist in the water phase, it may be used, for example, to determine the movement within the reservoir of water injected into any number of injection wells or to predict the variations in salinity or concentration of other chemical species in the water produced from the reservoir.

ECLIPSE 100

Tracers may also be defined to exist in a hydrocarbon phase or component, and the ECLIPSE 100 Tracer option models the exchange of the tracer fluid elements between the free and solution states whenever mass transfer takes place in the associated hydrocarbon phase. The Tracer option is therefore useful in predicting the movement of the initial free gas or solution gas during the evolution of reservoir.

ECLIPSE 300

The ECLIPSE 300 `TRACK` option allows a division of hydrocarbons in the original reservoir to be tracked through into recovered surface volumes.

The tracer concentrations can be initialized on a region basis using a depth table method of data input. This method of data input is especially suited to tracking fluid which initially exists within a particular region of the reservoir. In the case of gas condensate reservoirs, it is thus possible to track the oil which initially exists in the vapor phase in separate regions of the reservoir and to determine its subsequent movement through the reservoir grid. The Tracer option could also be used to predict the concentration of chemical impurity species (such as the sulphur content of the oil) in the hydrocarbon production streams.

The ECLIPSE Tracer Tracking option allows up to 50 tracers to be defined in a single run. Each tracer exists in a particular stock tank phase although mass exchange of the tracer fluid may occur between the free and solution states of each phase. This ensures that if, for example, a tracer is defined to be the dry gas injected into an undersaturated oil reservoir the solution of the injected tracer gas into the oil is modeled accurately. Re-injection of tracer gas cannot, however, be modeled in the current Tracer option.

The current ECLIPSE Tracer Tracking option assumes that the presence of tracers does not affect the PVT properties of the phases in which they are embedded. The tracers are thus to be regarded as passive. The tracer equations are a set of conservation equations for each tracer species with an exchange term for hydrocarbon phase tracers to model mass transfer effects. The tracer concentrations are updated fully-implicitly at the end of each time step after the oil, water and gas flows have been computed.

Control of numerical diffusion

ECLIPSE 100

The differential equations governing the evolution of the concentration of tracer species within each grid cell are a series of linear convection equations. ECLIPSE solves the tracer transport equations at each time-step using a fully implicit time discretization. Such a system of difference equations is particularly prone to exhibiting the effects of numerical diffusion errors, particularly in regions of the grid where tracer concentrations vary most rapidly. The smearing of tracer interfaces tends to be most apparent in single phase regions (for example gas injection into a gas cap, water injection into aquifers).

There are two options in ECLIPSE 100 to reduce the effects of numerical diffusion:

Cascade algorithm

ECLIPSE 100

This option is invoked by setting item 5 to 'DIFF' in keyword TRACERS in the RUNSPEC section. At present this option applies to tracers embedded in phases where no mass transfer effects can take place.

The tracer concentrations are calculated in flow order (the cascade). In cells in which the phase throughput ratio is less than one, the explicit tracer concentrations are used. However, if the throughput ratio exceeds one, the tracer concentration cascades on to the downstream cells. Hence the algorithm is stable for all time-step sizes.

In a serial run, the cascade algorithm is always used for **active** tracers whenever no mass transfer is associated with these tracers, whether or not 'DIFF' is enabled in the RUNSPEC section keyword TRACERS. It is possible to stop the cascade algorithm being used by employing the NOCASC keyword in the RUNSPEC section. This forces the linear solver algorithm to be used for solving all single phase tracers.

This algorithm is not available for parallel runs.

Note Brine is also an active tracer and prior to the 2004A release was also solved automatically with the cascade method. Since the 2004a release, a brine tracer has been solved with the user-selected method. The cascade solver cannot be used for the Multi-component brine option.

Flux limiting scheme

ECLIPSE 100

The explicit flux limiting scheme is invoked by setting item 5 to 'DIFF' in keyword TRACERS in the RUNSPEC section, and in addition specifying the TRACTVD keyword in the PROPS section. The implicit flux limiting scheme is invoked by specifying the TRACITVD keyword in the PROPS section. These options only apply to tracers embedded in phases where no mass transfer effects can take place.

The scheme is second order in smoothly varying regions, but uses a limiting function to prevent unphysical oscillations near sharp fronts.

For the explicit flux limiting scheme, tracer concentrations are updated explicitly using a mini time-step to prevent instability. As the tracer equation is linear, these explicit time-steps are very cheap, and hence the method can be applied to most problems (see [Ref. 69]).

For the implicit flux limiting scheme, the tracer concentration is computed using the same time stepping control as the standard fully implicit solver.

Lagrangian tracer solver (E300)

ECLIPSE 300

The Lagrangian tracer solver is activated with the LAGTRAC keyword in the RUNSPEC section and the LAGTRUPD keyword in the SCHEDULE section.

This solver is used in place of the standard implicit tracer solver for all tracers when activated. It uses FrontSim technology (e.g. the Pollock method) to trace a set of streamlines based on the flows computed by ECLIPSE. The tracer solver uses these streamlines to find the source of fluid in each cell and updates the tracer concentration in each cell according to the tracer concentration in the source cells.

This technique is not fundamentally conservative and does require that the streamline set accurately represents the flow of fluid in the reservoir. Assuming that this is the case, this method can give a faster, less numerically diffusive tracer solution. To ensure that the streamlines accurately describe the reservoir flow it is important that there are enough streamlines and that the streamlines are updated if the reservoir flow changes due to changes to well controls, movement of fronts and so on. The updates occur according to the data entered in the LAGTRUPD keyword.

The tracer concentration at the time that the streamline is traced is stored on the streamline; it is this value that is used, along with the time since this value was stored, to determine tracer concentration at the destination cell. This allows a solution with less numerical diffusion. The user can choose to update these concentrations without retracing the streamlines using LAGTRUPD. In any event these values are updated at each report step.

Lagrangian tracer solver speed and accuracy

The Lagrangian tracer solver is very quick at calculating tracer distributions once a streamline set has been traced. Tracing a set of streamlines is a slow process. As a result there is a trade-off between speed and accuracy which may be controlled by judicious use of the LAGTRAC and LAGTRUPD keywords.

The third argument of the LAGTRAC keyword should be used to determine the number of streamlines to be traced; more streamlines will slow down both the streamline tracing and the solution of the tracer concentration. The default value has been chosen to give a reliably accurate solution, but this can be reduced in most cases to a value of 0.05.

The LAGTRUPD keyword may be used to control the frequency with which streamlines are retraced. The default values cause streamlines to be retraced every time step; this is safe but slow. In the majority of cases the retrace frequency can be reduced dramatically, with a retrace roughly every report step being reasonable, and a value of at least 0.1 for argument 3 also being safe in most cases.

One final point to consider is that a set of streamlines can be used for more than one tracer. This means that multiple tracers which have the same carrier (i.e. tracers with the same argument 2 in TRACER and all LUMPING tracers) are all solved using one streamline set. As the majority of the cost of the Lagrangian tracer option is associated with tracing the streamlines, this means that every tracer after the first for a given carrier has a very marginal cost in memory and CPU time. This contrasts sharply with the standard tracer solver where the cost of calculation is almost proportional to the number of tracers.

Lagrangian tracer solver in parallel

The Lagrangian tracer solver is compatible with the PARALLEL keyword. This is implemented by gathering the grid and flow information necessary to trace a streamline onto the master processor and then copying this information to all processors. Each processor then traces some of the streamlines required, and the streamlines are finally mirrored across processors. This allows the tracer solver to run without any cross-processor communication being necessary except when tracing streamlines.

Lagrangian tracer summary

The LAGTRAC keyword is used to activate the Lagrangian Tracer Solver. The LAGTRUPD keyword is then used to define the solver method and the streamline update strategy. The RPTSLN keyword can be used to output the streamlines produced by the Lagrangian solver as SLN format files. LTRACE and NOLTRACE keywords are also available to allow the user to fine tune the update of the streamlines.

Using the ECLIPSE tracer tracking facility

ECLIPSE 100

The Tracer Tracking facility in ECLIPSE 100 is activated by means of three integers NOTRAC, NWTRAC, NGTRAC set in keyword TRACERS in the RUNSPEC section. These three integers represent the maximum number of tracers to be defined in each of the three stock tank phases: water, oil and dry gas. It is recommended that the precise number of tracers to be tracked should be specified in keyword TRACERS in the RUNSPEC section in order to minimize the memory requirements of the model.

ECLIPSE 300

The Tracer Tracking facility in ECLIPSE 300 does not require the keyword TRACERS in the RUNSPEC section; it is activated by entering the TRACER keyword in the PROPS section of the input data.

The names of each tracer and their associated stock tank phases are defined using the TRACER keyword in the PROPS section of the input data. In ECLIPSE 100 the maximum number of rows of data is (NOTRAC+NWTRAC+NGTRAC). Each tracer name is, at present, limited to be a name of up to 3 characters in length. The output of TRACER data is controlled by the TRACER mnemonic in the RPTPROPS keyword.

ECLIPSE 100

In the case of tracers defined to exist in the gas phase when DISGAS is specified in the RUNSPEC section, or tracers defined in the oil phase when VAPOIL is specified in RUNSPEC, the full tracer solution consists of a concentration in both the free and solution states of its associated phase. ECLIPSE 100 automatically allocates space for tracers which may exist in the free or solution state, and issues error messages whenever insufficient data is provided to calculate the initial tracer conditions.

The initial tracer concentrations may be supplied in one of two ways. The most direct method is enumeration using the keyword TBLK in the SOLUTION section of the input data. An alternative method is to define a series of tracer concentration versus depth tables in the SOLUTION section, together with the associated region numbers in the REGIONS section of the input data. The data is entered by means of the keywords TVDP and TNUM respectively, and control of the depth table and region data output is achieved using the RPTSOL and RPTREGS keywords. The keywords TBLK, TNUM and TVDP must be used in conjunction with the character F or S (free or solution), together with the tracer name, to make a composite keyword of up to 8 characters in length. For example, to initialize a water phase tracer whose name is 'WTR', the keywords required for the region/depth table method of input would be TVDPFWTR and TNUMFWTR.

It is possible to restart a run with more tracers defined than in the original base run; this allows a tracer run to be restarted from a non-tracer base case. The extra tracers must be initialized using either TBLK for TVDP keywords as described above. If the TBLK or TVDP keywords are used for tracers that have read from the RESTART file, then the data is ignored.

You may specify the initial concentration of a water phase tracer in each of the analytic aquifers in a simulation run. This can be achieved using the AQANTRC keyword in the SOLUTION section of the input data. Output of the status of each water tracer for each analytic aquifer is obtained at each report time using the mnemonic AQUCT or AQUFET in the RPTSCHED keyword.

Tracer injection control is achieved by means of the WTRACER keyword in the SCHEDULE section. This keyword allows the specification of the concentration of a particular tracer in the injection stream for each well. A default injection concentration value of zero is assumed for tracers not defined under the WTRACER keyword. The WTRACER keyword optionally allows the injected tracer concentration to be a function of the total cumulative fluid injected, which can be useful for modeling H_2S generating processes.

Note Injection/production of tracers in wells having significant cross flow may lead to tracer mass balance errors due to current limitations in the tracer model. The mass balance errors can be examined in the tracer-in-place reports in the print file, enabled by mnemonic FIPTR in keyword RPTSCHED. For non-linear tracers, for example environmental tracers with adsorption enabled, the issue with cross flow in wells may lead to convergence problems. The problem can be resolved by disabling cross flow (see item 10 of the WELSPECS keyword).

ECLIPSE 100

Reporting of the tracer concentration in each grid cell is activated using the mnemonic TBLK or TRACER in the RPTSCHED keyword. Output of tracer volumes within each reporting region and the inter-region tracer flows are available using the mnemonic FIPTR in the RPTSCHED keyword. Grid block arrays of tracer concentration are reported under a mnemonic consisting of the tracer name together with the character F and/or S. For example, if IGS is a gas phase tracer in a run with dissolved gas (DISGAS is specified in the RUNSPEC section), then two grid block concentration arrays are produced under the mnemonics IGSF and IGSS which represent the local concentration of IGS in the free and dissolved gas respectively.

ECLIPSE 300

Each tracer sets up a solution array, with its own name, which may be reported using RPTSOL and RPTSCHED, or output to graphics using RPTRST. For example:

```
RPTSOL
PRESSURE SWAT TR1 TR2 /
RPTRST
PRESSURE SWAT TR1 TR2 /
```

Completion, well, group and field reports for production/injection rates and cumulative production/injection of each tracer are provided with the well reports in the PRINT file, which may be requested by using the mnemonic WELLS in the RPTSCHED keyword.

ECLIPSE 100

Tracers can be injected in rivers using the keyword RIVTRACE in the SCHEDULE section.Tracers can be injected in a specified reach of the river and through the upstream end of a river branch. The RIVTRACE keyword optionally allows the injected tracer concentration to be a function of the total cumulative water flow.

Tracking original fluids

Introduction

ECLIPSE 300

This section describes the operation of the ECLIPSE 300 TRACK option, which enables a division of hydrocarbons in the original reservoir to be tracked through into recovered surface volumes. The option is enabled by entering the TRACK keyword with appropriate arguments in the RUNSPEC section. To use the enhanced Tracking facility, with user defined regions, it is also necessary to define the maximum number of regions to be used in the REGDIMS keyword in the RUNSPEC section.

Formulation

Suppose that we divide the hydrocarbons in a reservoir into those lying above the GOC (gas-oil contact) and those lying below it. As the reservoir is produced these components will flow through the field to the production wells. Generally, each component will move along a different path. For example, the light components such as methane will fraction into the reservoir vapor, and tend to rise to the top of the reservoir. The heavy components will tend to remain in the oil and stay at the bottom of the reservoir.

At a given producer, each component will have a fraction from below the GOC and a fraction from above it. We can thus define two streams, one of components originating from below the GOC and one from above. If these are run separately through the separator system we can obtain stock tank oil and stock tank gas volumes attributable to each of these streams.

With the enhanced Tracking option it is possible to divide the reservoir in any way the user desires.

Usage

The Tracking option is turned on using the TRACK keyword. This allows the initial fluid to be split at the GOC or at a given depth. The following text discusses splitting at the GOC; splitting at a specified depth is analogous.

In the tracking model, production volume rates originating from hydrocarbons above and below the GOC are reported separately in the PRINT file and when using SUMMARY section mnemonics.

Summary items are as follows:

- For wells:

  The SUMMARY mnemonic for the oil surface volume production rate of a given well is WOPR. The oil surface volume production rates from above and below the GOC are WOPRA and WOPRB respectively.

  These are totaled over time separately, so that the total production from the well is WOPT and the totals from above and below the GOC are WOPTA and WOPTB.

  WGPRA, WGPRB, WGPTA and WGPTB are similar items for well gas rates and totals.

- For groups:

The well contributions are summed over the group structure.

GOPRA and GOPRB are the group rates from above and below the GOC. These are totaled over time, so that GOPTA and GOPTB are the group totals from above and below the GOC. GGPRA, GGPRB, GGPTA and GGPTB are similar items for group gas rates and totals.

- For the field:

  The field items follow those of the well and groups.

  FOPRA and FOPRB are the field rates from above and below the GOC; FOPTA and FOPTB are the field totals from above and below the GOC.

  FGPRA, FGPRB, FGPTA and FGPTB are similar items for field gas rates and totals.

Enhanced TRACKing

To enable enhanced tracking it is also necessary to use the REGDIMS keyword in the RUNSPEC section to define the maximum number of user defined regions. It is strongly recommended that this maximum does not exceed the actual maximum as it can represent a considerable unnecessary overhead. An additional option in the TRACK keyword is the 'REGONLY' keyword that can be used in place of 'GOC' or 'DEPTH' if region tracking is the only facility required.

To define the regions the TRACKREG keyword in the REGIONS section must be used.

Fluid in place reports

When the fluid in place in the reservoir at surface conditions is reported, the track option includes extra reports for the surface volumes obtained from the fluid in the reservoir which originated above and below the gas oil contact. These volumes may be obtained for the entire reservoir or by fluid in place reporting region.

Tracer structure

Internally, the TRACK option sets up tracer variables. These are called A_n, for component n above the GOC and B_n for component n below the GOC. Suppose a tracer is attached to the methane. Then a tracer A_1 will be set up, initialized to 1 for cells above the GOC and 0 for cells below the GOC. The tracer B_1 will be conversely initialized to 0 for cells above the GOC and 1 for cells below the GOC. (Cells which intersect the GOC are given appropriate fractional values.)

Although the A_n and B_n tracers are set up automatically, you can still display their concentration profiles in the usual way by requesting output using RPTRST.

It might be thought that one tracer per component would be sufficient. However, the problem is with injection hydrocarbon, which cannot be identified with either the 'above' or 'below' original components, so that we cannot assume that components not belonging to one set must belong to the other.

For the most accurate solution we should track all of the components in the reservoir, as each will generally form a different distribution in the reservoir. However, this may be expensive, as $2N_c$ tracers will be required. An alternative is to track a smaller number and interpolate or extrapolate to find the concentrations of the missing tracers. At least one light component (generally C1) and one heavier component (for example C7) should be tracked. The TRACK keyword allows you to select which components are to be tracked, or to select ALL for the full set.

Enhanced tracking

The naming of the tracers is done in a similar manner. If the 'REGONLY' option is used then the tracer A_ refers to region 1, B_ to region 2, etc. If the 'GOC' or 'DEPTH' option has been used the A_ and B_ tracer names are reserved for those, with region 1 tracers being denoted with C_, region 2 tracers with D_, etc. A table is output to the print file detailing the names of the tracers and which regions they refer to.

Note that the comments about expense given above are multiplied here if a significant number of regions are defined.

The normalization option

When the fluid in place or the production from a well is obtained the tracers are used to define a stream from above the GOC and a stream from below the contact. These are then flashed separately so that the volumes from above and below can be reported separately. The sum of these two volumes will not generally be exactly equal to the total volume produced by the well, for three reasons:

1. Injection gas, which is not from original reservoir components.
2. The separator flash is not linear, so the volume of a oil and gas resulting from a mixture is not the same as the sum of the volumes of two sub-mixtures.
3. The tracer solution will disperse like any numerical solution, so that the sum of the A and B tracer concentrations will not generally equal exactly one.

Note The NORM option is provided so that you can request that the volumes be normalized to equal the total rate of the well.If normalization with the TRACK keyword (argument 3) is activated then FGPRA, FGPRB, FGPTA and FGPTB and similar items for well and group gas rates are calculated as:

$$FGPRA = \frac{FGPR}{FGPRA + FGPRB} FGPRA$$

Alternative production reporting

An alternative method for calculating the production from above and below the GOC is to directly split a well's volumetric flow rate according to the tracer concentrations above and below the contact. So, for example, the well oil production from fluids above the GOC, $WOPR1$, is:

$$WOPR1 = WOPR \cdot \sum_{i=1}^{N} WXMF_i \cdot T_i \qquad \text{[EQ 73.1]}$$

where:

$WOPR$	is the well volumetric oil production rate
$WXMF_i$	is the well liquid mole fraction for component i
T_i	is the above GOC tracer concentration for component i
N	is the number of components.

Similar equations can be written for the well oil production from fluids below the GOC and for the gas production rates. This way of splitting a well's production ensures that the sum of the volume from above and below the GOC equal the total volume produced by the well in the absence of injection gas and subject to the numerical accuracy of the simulation.

This method is known as the molar volumetric split and the resulting field, group and well rates and totals are calculated in addition to the standard TRACK option production quantities.

The molar volumetric split quantities may be reported using the following SUMMARY section mnemonics:

- For wells:

  W(O,G)P(R,T)1 gives the oil and gas surface volume production rates and totals from fluids above the GOC

  W(O,G)P(R,T)2 gives the oil and gas surface volume production rates and totals from fluids below the GOC

- For groups:

  G(O,G)P(R,T)1 gives the oil and gas surface volume production rates and totals from fluids above the GOC

  G(O,G)P(R,T)2 gives the oil and gas surface volume production rates and totals from fluids below the GOC

- For the field:

  F(O,G)P(R,T)1 gives the oil and gas surface volume production rates and totals from fluids above the GOC

  F(O,G)P(R,T)2 gives the oil and gas surface volume production rates and totals from fluids below the GOC

Following the tracers

Tracer variables are concentrations, which normally lie in the range 0 to 1. Each tracer is associated with a carrier component. The equation defining the change in the tracer solution over a time step is then:

$$R_i = M_i^{T+\Delta T} \cdot C_i^{T+\Delta T} - M_i^T C_i^T + \Delta T \sum_n F^{i \to n} C_{in}^{T+\Delta T} + \Delta T C_i^{T+\Delta T} Q_{M_i} = 0 \qquad \text{[EQ 73.2]}$$

where:

M_i is the moles of carrier in cell i

C_i is the tracer concentration in cell i

$F^{i \to n}$ is molar carrier flow from cell i to neighboring cell n

Q_{M_i} is the carrier molar production from well completion in block i

C_{in} is the upstreamed tracer concentration from cell i to neighboring cell n.

Note Note that the tracer conservation equation is linear in C, and thus solves in one iteration of Newton's method.

Example of the use of this method

As an example, we consider the third SPE benchmark problem [Ref. 70], which is a small gas condensate study. The top of the reservoir is at 7315 feet and the bottom of the reservoir is at 7475 feet. We modify the problem slightly to place a gas-oil contact at 7400 feet, and produce the field using the standard gas cycling production strategy.

The `TRACK` option is selected as follows:

```
TRACK
GOC 1* U C1 C7+2 /
```

This requests initialization with respect to the GOC, no normalization of the output volumes and that tracking use tracers on the C1 and C7+2 components only.

In the `SUMMARY` section we request output of the field oil production rate, and the contributions from hydrocarbons originally above and below the GOC:

```
FOPR
FOPRA
FOPRB
```

The resulting production profiles are:

Figure 73.1 Total oil production rate and contributions from above and below GOC

Note Note that as the production well is completed below the GOC, initially all the production comes from here. This changes quickly as fluid movement drive by gas injection moves hydrocarbon down from cells above the GOC.

Delumping and lumping

Detailed compositional simulation using tracers

ECLIPSE 300

This feature, which is available in ECLIPSE 300, is a straightforward implementation of paper SPE 49068 by Leibovici et al. [Ref. 80]. The basic idea is to do a full simulation with as small a number of components as possible and then to use multiple tracers carried on each of these 'lumped' components to represent the detailed composition of that component. The rationale and details of the approach are explained in the above paper; here we will concentrate on how to apply this procedure.

The facility is switched on with the `LUMPDIMS` keyword in the `RUNSPEC` section. This gives the number of lumped components (not necessarily as many as the number of components in the 'lumped' simulation) and a maximum for the number of detailed components.

The detailed composition of the fluid is described in the `PROPS` section. The `LUMPING` keyword assigns the number of detailed components to each lumped hydrocarbon component and also allows names to be attached to them. **The order in which the detailed components are entered is important as it will be assumed by later keywords.** The initial detailed state of the reservoir is specified with the `DETAILMF` or `DETAILVD` keyword, which determines the proportion of the detailed components making up the lumped components. This can vary with depth. A table is entered for each tracer region (if more than one), defined with keyword `TNUM`.

Various physical properties of the detailed components can also be specified in the `PROPS` section with the `ACFDET`, `MWDETAIL`, `TCRITDET`, `PCRITDET`, `OMEGAADE`, and `OMEGABDE` keywords. These are identical in nature to their 'normal' simulation counterparts, though as in a normal simulation, not all the `*CRITDET` keywords are needed in a given simulation.

The detailed composition of an injected fluid is given by using the `WINJEDET` keyword in the `SCHEDULE` section. This must be given for each injection well. For a given well the make-up of the detailed fluid is described in an analogous way to the `DETAILVD` keyword.

Output relies on the standard tracer output with the well, group and field tracer concentration (summary keywords `WTPC`, `GTPC` and `FTPC`) now representing the mole fraction of that detailed component within the overall fluid from that well, with the names of the tracers being those given in the `LUMPING` keyword. No other summary output is available; in particular the "Traced Mole Fraction" summary mnemonics `FTMF`, `GTMF` and `WTMF` will not report useful quantities as these are the product of the tracer concentration with the carrier molar rate.

Diffusion and dispersion

ECLIPSE 100

Prior to 2004A diffusion was only available for environmental tracers. With the 2004A release diffusion was made available for all tracers. Also a form of velocity and concentration dependent dispersion was added, see the keywords `DISPDIMS`, `TRDIS` and `DISPERSE`.

In 2005A mechanical dispersion (see [Ref. 108]), keyword `HDISP`, was also introduced. Note that mechanical dispersion is sometimes referred to as hydraulic dispersion, or streamwise diffusion, or streamwise dispersion.

Treating passive tracers as non-linear

Convergence problems in ECLIPSE 100

ECLIPSE 100

Sometimes problems can occur with passive tracers, which show as solutions values unacceptably below the minimum, or above the maximum, value input. Whilst the simulator should solve the passive tracer problem exactly (which in some sense it does do) minute mass balance errors in the underlying reservoir simulation get massively magnified in the tracer calculation leading to obviously incorrect results. Two-phase tracers seems particularly sensitive. The only way to overcome this problem is to solve the underlying reservoir simulation to a greater accuracy; usually well beyond where it appears to have any effect on the reservoir simulation. This unavoidably leads to large, often very large, increases in CPU times. The idea of treating passive tracers as if they were non-linear is an attempt to try to automate the process of adjusting the convergence tolerances for the simulation.

Hint It would always be better to use `TUNING` to tighten the tolerances before the tracers notice any difficulty. It is always possible that it is too late by the time the tracers fail to converge.

Having chosen to treat a passive tracer as a non-linear tracer, with item 8 of the `TRACERS` keyword, ECLIPSE 100 then performs checks on the convergence of the tracer. If it notices a problem the time step is chopped and the whole solution (including the underlying reservoir simulation) is repeated with the new time-step. The first time this occurs `MATCORR` is switched on (if it is not already on). Also the linear and non-linear convergence tolerances are tightened by the one-off application of the factors in items 9 and 10.

Hint We suggest the choice of a factor of 0.01-0.001 for item 9 and leaving item 10 defaulted. This is because increasing the linear iterations has a direct effect on material balance and is relatively cheap. For items 11 and 12 values of 0.1-0.3 and 3-4 seem reasonable.

Note It is important not to allow the tolerances to be tightened too much as the simulator will reach a point where the tolerances are below the rounding error and it becomes impossible to satisfy the conditions.

Units

Chapter 74

The unit conventions

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

There are four unit conventions currently used:

- METRIC units
- FIELD units
- LAB units
- PVT-Metric units (in ECLIPSE 300 only)

The `LAB` convention is intended for use when simulating laboratory-scale experiments. It is based on the original Darcy units, except that the unit of time is the hour rather than the second. The units for each data quantity are given in Table 74.1.

Table 74.1 A table of the units used for four conventions

Quantity	Metric	Field	Lab	PVT-M
Length, depth, radius	m	ft.	cm	m
Time	day	day	hr	day
Density	kg/m^3	lb/ft^3	gm/cc	kg/m^3
Pressure (absolute)	Barsa	Psia	Atma	Atma
Pressure (difference)	Bars	Psi	Atm	Atm
Temperature (absolute)	K	$°R$	K	K
Temperature (difference)	$°C$	$°F$	$°C$	$°C$
Compressibility	1/Bars	1/Psi	1/Atm	1/Atm
Viscosity	cpoise	cpoise	cpoise	cpoise
Permeability	MDarcy	MDarcy	MDarcy	MDarcy
Liquid surface volume	sm^3	stb	scc	sm^3
Gas surface volume	sm^3	Mscf	scc	sm^3

Table 74.1 A table of the units used for four conventions (Continued)

Quantity	Metric	Field	Lab	PVT-M
Reservoir volume	rm^3	RB	rcc	rm^3
Liquid surface volume rate	sm^3/day	stb/day	scc/hr	sm^3/day
Gas surface volume rate	sm^3/day	MScf/day	scc/hr	sm^3/day
Reservoir volume rate	rm^3/day	RB/day	rcc/hr	rm^3/day
Formation volume factor (liquid)	rm^3/sm^3	RB/stb	rcc/scc	rm^3/sm^3
Formation volume factor (gas)	rm^3/sm^3	RB/Mscf	rcc/scc	rm^3/sm^3
Gas-oil ratio	sm^3/sm^3	Mscf/stb	scc/scc	sm^3/sm^3
Oil-gas ratio	sm^3/sm^3	stb/Mscf	scc/scc	sm^3/sm^3
Transmissibility	cP.m^3/day/Bar	cP.RB/day/Psi	cP.cm^3/hr/Atm	cP.m^3/day/Atm
Mass	kg	lb	gm	kg
Moles	kg-mol	lb-mol	gm-mol	kg-mol

A note regarding the definition of unit of **mol**:

The standard definition of 1 mole of an element or compound is in g-mol (or gram-moles) and is in terms of the number of atoms or molecules. There are $6.022.10^{23}$ atoms in 1 mole of any element. Everything is relative to Carbon-12 so that 1 mole of Carbon-12 weighs 12 grams.

In order to work in different unit systems, the number of moles is reported in kg-mol (Kg-M), lb-mol (lb-M) or g-mol (gm-M). They are defined for Carbon-12 so that 1 kg-mol weighs 12 kg and 1 lb-mol weights 12 lb. This means that there is 1000 times more atoms in 1 kg-mol than in 1 g-mol.

Note The multiplier M (used for the volume outputs) stands for a million in all units except for the FIELD units, where it means a thousand.

Constants

Table 74.2 gives the values of some principal constants in the four unit conventions.

Table 74.2 Constants used in the four unit conventions

Quantity	Metric	Field	Lab	PVT-M
Gravity constant (ECLIPSE 100)	0.0000981	0.00694	0.000968	
Gravity constant (ECLIPSE 300)	0.0000980665	0.00694444	0.000967841	0.0000967841
Darcy constant (ECLIPSE 100)	0.008527	0.001127	3.6	

Table 74.2 Constants used in the four unit conventions (Continued)

Quantity	Metric	Field	Lab	PVT-M
Darcy constant (ECLIPSE 300)	0.00852702	0.00112712	3.6	0.00864
Atmospheric pressure	1.01325	14.6959	1.0	1.0
Density of air	1.2232	0.076362	0.0012232	1.2232
Density of water	999.014	62.3664	0.999014	999.014
Gas constant, R	0.083143	10.732	820.55776	0.08205576

Standard conditions are taken as one atmosphere and $60°F$.

Conversion factors

Table 74.3 gives some useful conversion factors between the unit systems.

Table 74.3 Some useful conversion factors

Quantity	Metric	Field	Other units
Length	1 m 0.3048 m	=3.28084 ft. =1 ft.	
Volume	1 m^3 0.02831685 m^3	=35.31466 cf =1 cf	=6.289811 bbl =0.1781076 bbl
Mass	1 kg 0.45359237 kg	= 2.204623 lb =1 lb	=0.001 (metric) tons =0.0005 (short) tons
Density	1 kg/m^3 16.01846 kg/m^3	=0.06242797 lb/ft.^{3} =1 lb/ft.^{3}	=10^{-3} g/cm^3 =0.01601846 g/cm^3
Pressure	1 bar 0.06894757 bar	=14.50377 psi =1 psi	=0.986923 Atm =0.068046 Atm
Gas-liquid ratio	1 m^3/m^3 178.1076 m^3/m^3	$=5.614583 \cdot 10^{-3}$ mcf/Mscf =1 mcf/bbl	
Temperatures	TK	$=(T/1.8)°$R	

A note regarding the use of the **ton** as a unit of mass:

- In metric-based systems, a ton (also called a **metric ton** or **tonne**) is 1,000 kg.
- In imperial-based systems, a **short ton** is 2,000 lbs (this is also known as a **net ton** and is the common definition of a **ton** in the U.S.) while a **long ton** is 2,240 lbs
- Prior to 2007.1, when working in `FIELD` units, ECLIPSE 300 reported both mass rates and mass totals (for example, the `WOMR` and `GWMT` summary vectors) in **long tons**. This was changed to **short tons** at 2007.1. The previous behavior can be recovered by setting `OPTIONS3` item 112 to a non-zero value. An '`L`' (for long) or '`S`' (for short) has been added to the name of the unit in the output from 2007.1 to clarify which type of ton is being used.

API gravity

API gravity = (141.5 / liquid gravity) - 131.5

Conversion for an ideal gas (Z=1)

For an 'ideal gas' Z-factor of unity, we can use the ideal gas equation:

$$PV = nRT \qquad \text{[EQ 74.1]}$$

where:

P is pressure

V is volume

n is number of moles

R is the gas constant

T is temperature

Using [EQ 74.1], we can calculate the following:

- When $P = 14.7\text{psia}$ and $T = 519.67°\text{R}$,
  - the volume occupied by one mole of gas is:

$$V_{\text{molar}} = 379.39445\text{ft}^3 \quad \text{[EQ 74.2]}$$

  - the number of moles in unit volume is:

$$n_{\text{moles}} = 0.00263578\text{mol} \quad \text{[EQ 74.3]}$$

- When $P = 1.013\text{Bar}$ and $T = 288.15\text{K}$,
  - the volume occupied by one mole of gas is:

$$V_{\text{molar}} = 23.650203 \times 10^{-6}\text{m}^3 \quad \text{[EQ 74.4]}$$

  - the number of moles in unit volume is:

$$n_{\text{moles}} = 0.04228293\text{mol} \quad \text{[EQ 74.5]}$$

Vertical Equilibrium

Chapter 75

Introduction

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL

This chapter describes the ECLIPSE 100 Vertical Equilibrium option.

Vertical equilibrium (VE) is the assumption that equilibrium is established in a cell in a time short compared to that characteristic of horizontal flows. This implies that the phase hydrostatic potential is independent of depth within a cell. In the absence of capillary pressure effects, the saturation distribution is then a step function depending on the fluid contact depths. This saturation distribution is taken account of when ECLIPSE calculates relative permeability values for the cell faces. In the case shown (Figure 75.1), for example, water may flow out of the bottom of the cell, but not out of the top, at which no mobile water exists. The water flow through the sides reflects the fraction of the cell faces over which a mobile saturation exists. If the VE assumption of approximate hydrostatic equilibrium is valid, such a segregated saturation distribution may provide a more accurate description of fluid flow than a default dispersed option in which the fluids are assumed to be evenly distributed over the grid block.

Figure 75.1 A cell with approximate hydrostatic equilibrium

The effect of corner point geometry upon the VE model is to introduce different depths for cell faces. For example, water may only flow out of an up-dip cell face if the contact level is sufficiently high.

In such a model hysteresis can occur if, for example, oil invades the water zone and retreats, leaving a residual oil saturation.

An option exists in the ECLIPSE VE implementation that enables you to specify the fractions of distributed and segregated saturation distribution functions which are to be used.

The use and formulation of the ECLIPSE VE option is discussed in more detail below.

Using the VE option

The Vertical Equilibrium option in ECLIPSE 100 is enabled by including the keyword VE in the RUNSPEC section.

There is a set of four keywords (VEFRAC, VEFRACV, VEFRACP, VEFRACPV) which are specific to VE. These keywords enable you to specify either dispersed (rock curve) or segregated (VE) saturation functions, or a weighted average of the two, for calculating relative permeabilities and capillary pressures. VEFRAC gives the VE relative permeability fraction, and VEFRACP the VE capillary pressure fraction, for the whole field. VEFRACV and VEFRACPV allow the specification of VEFRAC and VEFRACP values on a cell-by-cell basis. The values of VEFRAC and VEFRACP should lie in the range 0.0 to 1.0. For example, using the following:

```
VEFRAC
 0.8 /
VEFRACP
 0.8 /
```

will specify 80% VE and 20% rock curves for both relative permeability and capillary pressure. The default is a VEFRAC and VEFRACP value of 1.0. If the VE option is not required, omit the keyword VE from RUNSPEC, rather than use a VEFRAC value of 0.0 everywhere.

If VE is specified, and the ninth argument of the EQUIL keyword data is set to zero, then a method of equilibration specific to VE is used, which ensures that the initial contact depths are exactly matched by the initial saturations, and which yields a static initial solution. If this ninth argument is not zero, then a normal ECLIPSE fine grid equilibration is performed using the rock curves.

The latter method is only likely to be useful if a non default VEFRAC value is used; for a pure VE run, the former option is best.

To obtain a printed output of the contact depths at each report step, use the mnemonics OWC and/or GOC in the RPTSCHED keyword. These control the output of the oil water and gas oil contact depths respectively.

The Vertical Equilibrium option can not be used in combination with the Hysteresis option in the same run.

The Endpoint Scaling option/initial mobile fluid end point correction should not be used with the Vertical Equilibrium option.

The cell saturation distribution

The saturation distribution in a cell assumed by the VE model is based on the assumption that the phase hydrostatic potential is independent of depth. In the absence of capillary pressure effects, the saturation distribution becomes a step function. The form of this depends on the phase option used.

The oil-water case

The distribution is derived from the water saturation value, S_w. If Z_c is the current oil water contact depth, and Z_l is the lowest contact depth which has occurred in the run, the distribution is as shown below. This has three depth intervals, as follows:

Region 1

Above the current oil water contact. The water saturation takes the connate value, S_{wco}, and the oil saturation is $1\text{-}S_{wco}$.

Region 2

Above the lowest oil water contact, but below the current. In this region water has displaced oil, so the critical oil saturation remains, S_{ocr}, whilst the water saturation is $1\text{-}S_{ocr}$.

Region 3

Below the lowest oil water contact. Only the connate oil saturation exists, S_{oco}, which is usually zero, and the water saturation is $1 - S_{oco}$.

If the current contact attempts to fall below Z_l, oil invading the water zone, then Z_l is set to the new Z_c value. If the contact again rises Z_l retains its value, so that a critical oil saturation remains in the invaded region.

Figure 75.2 Oil-water vertical equilibrium

The oil-gas case

The distribution is derived from the gas saturation value, S_g. If Z_c is the current oil gas contact depth, and Z_h is the highest contact depth that has occurred in the run, the distribution is as shown below. This has three depth intervals, as follows:

Region 1

Above the highest oil gas contact. The connate oil saturation exists, S_{oco},which is often zero, and the gas saturation is 1-S_{oco}.

Region 2

Below the highest oil gas contact, but above the current. In this region gas has displaced oil, so the critical oil saturation remains, S_{ocr}, whilst the gas saturation is 1-S_{ocr}.

Region 3

Below the current oil gas contact. The gas saturation is the connate value, S_{gco}, which is usually zero, and the oil saturation is 1-S_{gco}.

If the current contact attempts to rise above Z_h, oil invading the gas zone, then Z_h is set to the new Z_c value. If the contact again falls Z_h retains its value, so that a critical oil saturation remains in the invaded region.

Figure 75.3 Oil-gas vertical equilibrium

The gas-water case

The distribution is derived from the gas saturation value, S_w. If Z_c is the current water gas contact depth, and Z_l is the lowest contact depth that has occurred in the run, the distribution is as shown below. This also has three depth intervals, as follows:

Region 1

Above the current water gas contact. The water saturation has the connate value, S_{wco}, and the gas saturation is 1-S_{wco}.

Region 2

Above the lowest water gas contact, but below the current. In this region water has displaced gas, so the critical gas saturation remains, S_{gcr}, whilst the water saturation is 1-S_{wcr}.

Region 3

Below the lowest water gas contact. The gas saturation is the connate value, S_{gco}, which is usually zero, and the water saturation is 1-S_{gco}.

If the current contact attempts to fall below Z_l, gas invading the water zone, then Z_l is set to the new Z_c value. If the contact again rises Z_l retains its value, so that a critical water saturation remains in the invaded region.

Figure 75.4 Gas-water vertical equilibrium

The three phase case

In this case S_w and S_g are distinct variables. The usual distribution is shown in Figure 75.5. There are five depth intervals, as follows:

Region 1

Above the highest gas oil contact. The water saturation has the connate value, S_{wco}, and the oil saturation is the connate with respect to gas, S_{ocog}. The gas saturation is thus $1-S_{wco}-S_{ocog}$.

Region 2

Below the highest oil gas contact, but above the current. Gas has displaced oil, so the critical oil saturation with respect to gas remains, S_{ogcr}, with $S_w=S_{wco}$ and $S_g=1-S_{ogcr}-S_{wco}$.

Region 3

Below the current oil gas contact and above the current oil water contact. The water and gas saturations take their connate values, (S_{wco}, S_{gco}), and the oil saturation is $1-S_{wco}-S_{gco}$.

Region 4

Below the current oil water contact, but above the lowest oil water contact. Water has displaced oil, so the critical oil saturation with respect to water remains, S_{owcr}, with $S_w=1-S_{owcr}-S_{gco}$.

Region 5

Below the lowest oil water contact. The gas saturation is connate, S_{gco}, and the oil saturation is the connate value with respect to water, S_{ocow}. The water saturation is $1-S_{ocow}-S_{gco}$.

In any of the phase combinations, one or more of the regions may not exist, depending on the conditions in a cell.

Figure 75.5 Three-phase vertical equilibrium

In the 85/7 version of ECLIPSE the oil-water contact was simply not permitted to be above the oil-gas contact. In later versions the treatment of the three-phase VE case was extended to cover the case in which the oil zone becomes immobilized. When this occurs, the oil zone becomes a residual saturation over a certain depth interval, and the water may rise above this zone, leading to a gas-water contact.

The immobilized oil state can occur with the gas-water contact above, below or in the immobilized oil. The first situation would arise if the water contact rose up into the original oil zone. In the normal three phase oil case, the residual oil saturation would then have to exist from the original contact depth to the current. This is possible as long as sufficient oil saturation exists. If, however, the contact continues to rise, a point will be reached at which all the oil exists at residual saturation. If the contact rises further, the following immobilized oil distribution will exist (assuming connate oil and gas saturations are zero):

Figure 75.6 Three phase equilibrium with immobilized oil

ECLIPSE now models this immobilized oil state correctly, and returns to the mobile oil state once a sufficient oil saturation exists.

The various connate and critical saturations used in the above are obtained from the `PROPS` section data with the appropriate saturation table number, as follows:

Connate water saturation, S_{wco}. The smallest saturation value in the `SWFN` table.

Critical water saturation, S_{wcr}. The smallest saturation value for which a non-zero k_{rw} value exists.

Connate gas saturation, S_{gco}. The smallest saturation value in the `SGFN` table.

Critical gas saturation, S_{gcr}. The smallest saturation value for which a non-zero k_{rg} value exists.

Connate oil saturation in water, S_{ocow}. This is $1-S_{wmax}-S_{gco}$, where S_{wmax} is the maximum saturation value in the `SWFN` table.

Connate oil saturation in gas, S_{ocog}. This is $1-S_{gmax}-S_{wco}$, where S_{gmax} is the maximum saturation value in the `SGFN` table.

Critical oil saturation in water, S_{owcr}. This is the lowest oil saturation for which a non zero k_{ro} value exists, with $S_g=S_{gco}$ (that is, from the second column of the `SOF3` table in the three phase case or from `SOF2` in the two phase case).

Critical oil saturation in gas, S_{ogcr}. This is the lowest oil saturation for which a non zero k_{ro} value exists, with $S_w=S_{wco}$ (that is from the third column of the `SOF3` table in the three phase case or from `SOF2` in the two phase case).

VE relative permeabilities and capillary pressures

In the ECLIPSE VE algorithm, the current best solution to the non-linear fluid flow equations is held as a set of cell saturations. The current contact levels are calculated from the saturations using the models for the segregated fluid distributions described in the previous section. (The lowest and highest levels, Z_l and Z_h, are stored as arrays, and will effect the relationship between the saturation and current contact values). Once the contact depths have been obtained, the relative permeability values over the cell faces and a 'pseudo capillary pressure' can be evaluated.

These calculations require a knowledge of the cell geometry, obtained by an analysis at the start of the run. For each cell a table of fractional volumes below each distinct depth is stored, a distinct depth being one at which one or more cell corners exist. These fractional volumes are obtained using a linear interpolation of the horizontal cross-sectional areas of the cell at each distinct depth. A similar table of fractional areas below depths is kept for each cell face. With these tables it is possible to obtain effective relative permeability values by integration over depth, weighting the appropriate relative permeability values implied by the saturation distributions in each interval. The different depths of block faces are thus correctly accounted for in corner point geometry.

The 'pseudo capillary pressure' is due to the difference in hydrostatic head between two phases, and is a function of the distance between the cell center and the current contact depth. In the oil water case it is given by:

$$P_{cow} = g(\rho_w - \rho_o)(Z_{owc} - Z_{\text{centre}}) \qquad \text{[EQ 75.1]}$$

where ρ_w and ρ_o are the phase densities and g is the gravity constant.

Similarly, for the gas phase, the pseudo capillary pressure is given by:

$$P_{cog} = g(\rho_o - \rho_g)(Z_{goc} - Z_{\text{centre}})$$

The densities are calculated at the end of each time step and then held constant over the next time step. If water is not present in a cell, or has a saturation value lower than connate, then the water-oil contact Z_{owc} is set to the lowest depth in the cell. This effectively introduces a cut-off in the pseudo capillary pressure, although in cases where this occurs, the water would be immobile. In a similar manner, if gas is absent, the contact Z_{goc} is set to the top of the cell.

Residual flow model

When water sweeps through a cell containing oil, a residual oil saturation as defined in the SATNUM saturation table will remain immobile in the cell. In some cases this 'residual' oil may in fact be slightly mobile, and over a period of time some of this oil will flow, reducing the oil saturation to a new residual value. This situation can be modeled in a very simple manner using the residual flow option.

You specify a second set of saturation tables, using the RESIDNUM keyword in the REGIONS section. An extra component of relative permeability is then added to the flows, which is calculated by looking up the relative permeability on the RESIDNUM tables using the prevailing cell saturation.

$$K_{r\ \mathrm{tot}} = K_{r\ \mathrm{VE}} + K_{rr}(S_o) \qquad \text{[EQ 75.2]}$$

where

$K_{r\ \mathrm{tot}}$ is the total relative permeability

$K_{r\ \mathrm{VE}}$ is the VE relative permeability calculated from the contact depth

$K_{rr}(S_o)$ is the relative permeability from the RESDINUM table

S_o is the oil saturation.

The RESDNUM tables will typically have very small oil K_r values, with zero K_r values for phases with no residual flow.

For example, consider a system with the following relative permeability behavior:

Figure 75.7 Relative permeability behavior for a residual flow model

This relative permeability curve could be partitioned into a curve for VE (specified using SATNUM) and a curve for the residual flow specified using RESIDNUM.

Figure 75.8 V.E Curves (SATNUM)

Figure 75.9 Residual flow (RESIDNUM)

Miscellaneous points

RESTART files produced by VE runs include the lowest and highest contact depths, as required by the phase combination used. It is not possible to restart a VE run from a non-VE run.

Although the saturation distributions assumed by the VE algorithm are based on zero capillary pressures, non-zero values may be entered in the PROPS data section, to be used if the VEFRAC mixing parameter is not unity; the VE fraction will ignore rock capillary pressures.

The VE model described above assumes a three zone saturation distribution. It is possible to extend this model to allow four zones. In such a case, considering the oil water phase combination, there would be a critical water saturation up to a highest contact depth, and a (possibly different) connate saturation above this. This extension to a 'four cycle' VE model could be provided in future if demand exists.

If twisted sloping faults are used, the intersection of a cell face with a plane of constant depth becomes slightly curved. This curvature is ignored when setting up the cell geometry data for VE. However, the fractional volumes below distinct depths are then re-normalized to ensure that the volume fraction below the highest distinct depth is 1.

If the reservoir geometry is entered using TOPS, DX, DY etc., rather than with corner point geometry (COORD, ZCORN etc.), the top and bottom of the cells will be flat. In this case an unrealistic step function change in the relative permeability would be obtained when the fluid contact reaches the top of a cell, which would cause problems in solving the non-linear flow equations. To avoid this, the cell top is 'spread' slightly, over 10% of the cell depth, so that the flow relative permeability rises from zero to its maximum value as the fluid contact traverses this interval.

In a dipping reservoir specified using TOPS, DX etc. the VE algorithm will prevent flow to the adjacent cell if the water level is below the bottom of the adjacent cell. A better method of specifying a dipping reservoir would be to use corner point geometry (COORD, ZCORN etc.). However the full face of each cell can be specified for flow by using the OPTIONS keyword, setting OPTIONS(19)=1. A second option can be used by specifying OPTIONS(19)=2, in which case the flows are calculated assuming a corner point approximation to the original grid. The pore volume and transmissibilities are preserved, however the VE flow calculations are performed assuming the cells to be part of a continuous layer. This corner point approximation will be output to the GRID file and hence can be plotted using a pre-processor such as PETREL. This latter option is used in Local Grid Refinement cases if the grid is entered as a block centered grid.

Consider a reservoir specified using corner point geometry. When there is a gap of non-zero thickness between a pair of adjacent active cells in a column which are connected together (such as for a pinchout of non-zero thickness due to use of MINPV), contact depths cannot be resolved in this dead space. In this case, setting OPTIONS(52)=2 instructs ECLIPSE to modify corner-point depths on the bottom face of the top-most cell in the pair to be equal to those for the top face of the bottom-most cell. If the Local Grid Refinement option is used, ECLIPSE always modifies the corner point depths in local grids to avoid gaps between local and global grids where there is a connection.

In the assumed segregated fluid distributions described previously, the connate saturation of a phase was always assumed to be present throughout a cell. As soon as this is exceeded, a contact forms, and flow is generally possible through the top or bottom of the cell. This will usually occur at a lower saturation than the critical value. For example, the connate gas saturation is often zero. If any gas saturation exists in a cell it will be assumed to be concentrated at the top, and thus be mobile through the top of the cell, even if this is below the critical gas saturation. This is a straightforward consequence of the assumption of vertical equilibrium, but may not be

appropriate in some cases. For example, if gas appears in a cell through bubble point crossing, it would be evenly distributed through the cell, and could not concentrate at the top until the critical, rather than the connate, saturation is reached. Arguments 4 and 5 of the `OPTIONS` keyword (used in the `SCHEDULE` section) enable the critical water and gas saturation values to replace the connate in the simulation if required. Note, however, that this may result in the VE equilibration option not being static, as this always uses connate saturations.

Compressed vertical equilibrium option

The compressed Vertical Equilibrium option is a facility to allow the modeling of a reservoir in which the entire reservoir section is assumed to be in vertical equilibrium. If this assumption holds, then gravity forces dominate the viscous forces and we can assume that the phases will be distributed with the less dense phases above the denser phases.

The option takes full account of the reservoir heterogeneity by allowing the engineer to input a full 3-dimensional model of the reservoir, which may include many geological layers with different porosities, permeabilities and relative permeabilities. The individual layers are automatically merged in the Z-direction into compressed VE cells, which use internal tables to account for the variation in rock properties across the width of the merged layers.

The physical model assumed in the compressed VE option contrasts with the standard (uncompressed) VE description, in that vertical equilibrium is assumed to exist across the entire section, rather than within each original cell (representing a geological layer or sub-layer) individually. Note that if the assumption that the entire reservoir section is in VE holds, then the two models should give similar descriptions of the initial fluid distribution.

Features such a shale breaks and non-VE layers can be modeled. In these cases the merging of layers is only carried out over particular sections of the reservoir, rather than across the whole width. Non-neighbor connections are generated automatically to allow the modeling, for example, of intermittent shales.

Using the option

The option is initiated by setting item 1 to 'COMPRESS' in keyword VE in the RUNSPEC section. The 3D model will automatically be modified into a compressed VE grid having a number of larger merged cells. In the absence of any shale breaks (zero Z-transmissibilities or inactive cells between layers), a three dimensional run will reduce to a two dimensional areal model.

The COLLAPSE keyword is available in the GRID section to allow you to prevent certain cells being merged into compressed VE cells. Also, the CRITPERM keyword can be used to specify a minimum value of the Z-direction permeability below which a cell will not be merged with its lower neighbor in the column.

The top cell of each column of merged cells is taken as the representative cell. In the printed output these representative cells are output with a “T” in place of the decimal point; the original cells below the top cell are output with a “V”.

Breaking the compressed VE cells

Within a column of cells, each cell will be merged with the next active cell below, provided that none of the following situations occur:

- You have given the cell a collapse integer of zero in the COLLAPSE keyword, to prevent it from being merged.
- TRANZ = 0, representing an impermeable barrier. The assumption of vertical equilibrium breaks down if the transmissibility is zero.
- PERMZ ≤ the critical value entered in CRITPERM, if this keyword is present. If the CRITPERM keyword is absent, the critical permeability defaults to 0.

- There is an inactive cell with DZ > 0 between the two active cells. If an inactive cell has DZ > 0 then it must form an impermeable barrier.
- Either cell has been refined using RADFIN or CARFIN (LGR option).

Any of these situations will prevent the column being merged into a single collapsed VE cell. The column will then be represented by two or more collapsed VE cells. Non-neighbor connections will be generated wherever required.

Tabular description of the cell

Internal tables are set up to describe the compressed VE cells. These tables consist of a set of data items tabulated against distinct depths. The distinct depths are the corner points of the original cells that make up the compressed VE cell. If too many of these distinct depths occur together, some will be removed to improve efficiency.

There are tables for the cell volume and for each face. The tabulated data is as follows:

Cell volume table:

DEPTH

VOLUME Normalized volume tables, including porosity and NTG values

SOWCR Oil saturation at critical water (residual oil to water)

SOGCR Oil saturation at critical gas (residual oil to gas)

SWCR Critical water saturation

SGCR Critical gas saturation

SOWMX Oil saturation at maximum water saturation

SOGMX Oil saturation at maximum gas saturation

SWCO Connate water saturation

SGCO Connate gas saturation

KRO1 K_{ro} at SWCO, SGCR

KRO2 K_{ro} at SWCO, SGCO

KRO3 K_{ro} at SWCR, SGCR

KRW1 K_{rw} at SOWCR, SGCO

KRW2 K_{rw} at SOWMX, SGCO

KRG1 K_{rg} at SOGMX, SWCO

KRG2 K_{rg} at SOGCR, SWCO

Cell face tables:

DEPTH

AREA Normalized surface area tables

KRO2 K_{ro} at SWCO, SGCO

KRW1 K_{rw} at SOWCR, SGCO

KRW2 K_{rw} at SOWMX, SGCO

KRG1 K_{rg} at SOGMX, SWCO

KRG2 K_{rg} at SOGCR, SWCO

These internal tables can be examined by using the VEDEBUG keyword. For each cell requested, they can either be output as tables to the debug file or they can be output in GRAF user file format (again to the debug file). The GRAF user file can be read into the GRAF program to enable plots to be made of the tables described above.

Use with local grid refinement

The compressed VE option can be used in conjunction with Local Grid Refinement. Any cells that have been refined are not placed in compressed VE merged cells, as a more detailed description of the reservoir is required in these areas. Compressed VE cells will be generated adjacent to the refinement and will be automatically connected by non-neighbor connections.

Water components

Chapter 76

Introduction

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

In ECLIPSE, water is traditionally modeled using a single component. Density and viscosity variations caused by salinity changes can be modeled using the Brine Tracking option, see "Brine Tracking" on page 103. This option uses a tracer to track the salt movement in the reservoir. This single-component model can be applied to many practical cases. However, there are some situations where more than one water component is required:

- As an alternative brine model. When density variations are large, the tracer method may become unstable, since the tracer equations are solved after the flow equations.
- To model brine in multi-segment wells, see "Multi-segment Wells" on page 635. At present the Brine Tracking option cannot be used at with the Multi-segment Well option.
- To model reactions involving water borne components such as bacteria or dissolved solids, see "Chemical Reactions" on page 147.

Water component keywords

RUNSPEC section keywords

The RUNSPEC keywords are

Table 76.1 Water RUNSPEC keywords

Keyword	Description
COMPW	Enables the water component model and specifies the number of water components

PROPS section keywords

The PROPS keywords are:

Table 76.2 Water RUNSPEC keywords

Keyword	Description
CREFW	Compressibility of water components at the reference pressure
DREFW	Density of water components at the reference pressure
MWW	Molecular weight of water components
PREFW	Reference pressure for water component density and viscosity calculations
VREFW	Viscosity and viscosibility for water components at the reference pressure
WNAMES	Water component names

SOLUTION section keywords

The SOLUTION keywords are:

Table 76.3 Water SOLUTION keywords

Keyword	Description
AQSTREAW	Set composition of water analytic aquifer (can also be used in the SCHEDULE section)
WI	Specifies overall initial water composition (can also be used in the PROPS section)
WMF	Specifies initial water composition explicitly
WMFVD	Specifies initial water composition with respect to depth (can also be used in the PROPS section)

SCHEDULE section keywords

The SCHEDULE keywords are:

Table 76.4 Water SCHEDULE keywords

Keyword	Description
WELLSTRW	Sets composition of the injection water stream
WINJW	Specifies the nature of the injection water

OUTPUT keywords

The mnemonics WMF and MWAT can be used with the keywords RPTRST, RPTSCHED and RPTSOL to output water mole fraction and molar concentrations.

For summary block quantities, such as BMLSC, the water component index starts at $N_c + 1$ and ends at $N_c + N_w$ where N_c is the number of components given by COMPS and N_w is the number of water components given by the COMPW keyword.

Water component properties

Density

The molar density of water is given by

$$b_w = \frac{1}{V_w} \qquad \text{[EQ 76.1]}$$

where V_w is the specific volume given by:

$$V_w = \sum_c w_c \cdot \frac{MW_c}{\rho_{ref_c} \cdot (1 + X_c + 0.5X_c^2)} \qquad \text{[EQ 76.2]}$$

In [EQ 76.2] the sum is over the water components where:

w_c is the water component fraction in the water phase

ρ_{ref_c} is the reference density, specified with keyword `DREFW`

MW_c is the water component molecular weight, specified by the keyword `MWW`

The compressibility terms X_c in [EQ 76.2] are given by

$$X_c = c_{P_c} \cdot (P - P_{ref_c})$$

where

c_{P_c}

is the water component compressibility, specified with keyword `CREFW`

P_{ref_c} is the reference pressure, specified with keyword `PREFW`

Viscosity

The viscosity of water is given by

$$\mu_w = \sum_c w_c \cdot \frac{\mu_{ref_c}}{(1 + Y_c + 0.5Y_c^2)} \qquad \text{[EQ 76.3]}$$

where

w_c is the water component fraction in the water phase

μ_{ref_w} is the reference viscosity, specified with keyword `VREFW`

The “viscosibility” terms Y_c in [EQ 76.3] are given by

$$Y_c = -c_{V_c} \cdot (P - P_{ref_c})$$

where

c_{V_c} is the “viscosibility”, specified with keyword `VREFW`

P_{ref_c} is the reference pressure, specified with keyword `PREFW`

Surface properties

The surface density of water is still defined by the DENSITY keyword. This value is used to calculate the properties of injected water.

Aquifer properties

By default, an analytic water aquifer defined by AQUCHWAT, AQUCT, AQUFETP or AQUFLUX contains only the last water component. The AQSTREAW keyword may be used to give the composition of a specified aquifer. Any water component is allowed to leave the reservoir or enter it, depending on the aquifer composition. If the item 131 of OPTIONS3 is set to 1, this reverts to the pre-2008.1 behavior where the water entering / leaving the reservoir is made up entirely of the last water component. Additionally, the water properties used in the analytic aquifers are calculated as described above. If the item 119 of OPTIONS3 is set to 1, this reverts to the pre-2008.1 behavior where the PVT tables were used. See "Aquifer Modeling Facilities" on page 73.

Converting PVTW data to water component data

In standard simulations, water is modeled as a single component, with properties specified by the PVTW and DENSITY keywords. This case can also be modeled using the multiple component water option, with a single water component. Table 76.5 gives values for the water component keywords, derived from the PVTW and DENSITY data.

Note Note however, that PVTW and DENSITY data are specified for each PVT region, while water component data is specified for each EOS region, so the two models can produce different answers if the regions do not match.

Table 76.5 Water component values

Keyword	Value
MWW	18.015
PREFW	Item 1 of PVTW
DREFW	$DENS_w/BW_{ref}$ where $DENS_w$ is Item 2 of DENSITY BW_{ref} is Item 2 of PVTW
CREFW	Item 3 of PVTW
VREFW	Items 4 and 5 of PVTW

Converting PVTWSALT data to water component data

Water with varying salinity can already be modeled using the Brine Tracking Option, see "Brine Tracking" on page 103. The Multiple Component Water provides an alternative mechanism, with one component used to model freshwater and another component used to represent brine. In the Brine Tracking Option, water properties are defined using the `PVTWSALT` keyword. This can be converted to component data in a similar manner to the `PVTW` data, described above, except that the density term must now include the weight of salt

$$DREFW = (DENS_w + SAL_w)/BW_{ref}$$

where SAL_W is the salt concentration.

Examples

The following data sets come with the ECLIPSE installation:

Table 76.6 Multi-component water example files

File	Description
`MCW0.DATA`	Brine model (using tracers, for comparison)
`MCW1.DATA`	Brine model
`MCW2.DATA`	MSW Brine model
`MCW3.DATA`	Brine + solids + reactions
`MCW.GRF`	GRF to compare runs

Limitations

The multi-component water model can be used in place of the single component water model in most standard simulations (black oil, EoS or K-value). However, it can not be used at present in certain simulations where water has special properties. Therefore, options that **cannot** be used with the multi-component water model include:

- The `THERMAL` option, where water is allowed in the gas phase (see "Thermal Option" on page 1023).
- The `GASWAT` option, where water is modeled using an equation of state (see "The GASWAT Option" on page 405).
- The `CO2SOL` option, where CO_2 dissolves in water (see "CO_2/H_2S Solution in the Aqueous Phase" on page 143).
- Flux boundary conditions (see "Flux Boundary Conditions" on page 309).
- Satellite groups (see `GSATPROD`).
- `IMPSAT`.

In addition, there are currently some limitations when specifying water source terms:

- The water entering / leaving a multi-segment well using the source / sink term (`WSEGEXSS`), is made up entirely of the last water component.

The Wellbore Friction Option

Chapter 77

Introduction

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL

The Wellbore Friction option is a special extension to ECLIPSE 100; it is not available in ECLIPSE 300. It models the effects of pressure loss due to friction in the well tubing along the perforated length, and between the perforations and the bottom hole reference point of the well. The facility is primarily intended for use with horizontal and multi-lateral wells, in which frictional pressure losses may be significant over the horizontal section of the well-bore and in the branches.

In the standard well model, frictional pressure losses between the well connections and the bottom hole reference point are neglected entirely. The difference between the pressure at the connections and the bottom hole pressure is purely the hydrostatic head of the fluid mixture in the wellbore acting over the difference in true vertical depth between these locations. In wells completed vertically through the formation, the frictional pressure drop acting over the relatively short length of the perforated section is generally negligible.

In horizontal wells, however, the perforated section may extend over many hundreds of feet. The frictional pressure drop over this length may have a significant effect on the behavior of the well. The pressure in the well-bore towards the far end of the perforations (that is, away from the wellhead) will be higher than the pressure at the near end, so the drawdown will vary over the perforated length. This may cause the productivity per unit length of perforations to fall off towards the far end. Furthermore, the lower wellbore pressure at the near end of the perforations may cause localized water or gas coning to occur in this region, thus reducing the effectiveness of the horizontal well in overcoming coning problems. It is therefore important to consider the effects of frictional pressure losses when deciding on the optimum length and diameter of a horizontal well.

Calculating the frictional pressure loss

The frictional pressure drop over a length L of tubing is

$$\delta P_f = 2f \cdot \frac{L}{D} \cdot \rho \cdot v^2 \qquad \text{[EQ 77.1]}$$

where

f is the Fanning friction factor

D

is the tubing inner diameter

ρ is the fluid density

v is the fluid velocity

Transforming the flow velocity into the local volumetric flow rate Q (in rb/day or m^3/day) and taking into account the unit conversion factors, the pressure drop becomes

$$\delta P_f = \frac{C_f \cdot f \cdot L \cdot \rho \cdot Q^2}{D^5} \qquad \text{[EQ 77.2]}$$

where

C_f = 2.956E-12 in `FIELD` units

(L and D in ft., ρ in lb/ft.^{3}, Q in rb/day)

= 4.343E-15 in `METRIC` units

(L and D in m, ρ in kg/m^3, Q in rm^3/day)

= 2.469E-13 in `LAB` units

(L and D in cm, ρ in g/cm^3, Q in rcc/hr)

The Fanning friction factor depends upon the Reynolds number Re. For laminar flow ($\mathrm{Re} < 2000$),

$$f = \frac{16}{\mathrm{Re}} \qquad \text{[EQ 77.3]}$$

For $\mathrm{Re} > 4000$ we use Haaland's formula ([Ref. 71]),

$$\sqrt{\frac{1}{f}} = -3.6 \log_{10}\left(\frac{6.9}{\mathrm{Re}} + \left(\frac{e}{3.7D}\right)^{10/9}\right) \qquad \text{[EQ 77.4]}$$

where

e is the absolute roughness of the tubing in the same units as D

In the "uncertain region" (2000 < Re < 4000) we use a linear interpolation between the values at Re = 2000 and Re = 4000. The above scheme has the advantage of providing a direct calculation for values of f that are continuous in Re and are in reasonable agreement with the friction factor chart (see [Ref. 72], for example). It should be noted, however, that these formulae do not take account of fluid inflow through the perforations. Its influence on the effective roughness of the tubing is largely unknown at present, and the engineer is advised to vary the roughness to gauge the sensitivity of the results to this parameter.

The Reynolds number is

$$\mathrm{Re} = \frac{\rho v D}{\mu} \qquad \text{[EQ 77.5]}$$

where

μ is the viscosity of the fluid.

Converting v to the volumetric flow rate Q and including the unit conversion factors, this becomes

$$\mathrm{Re} = \frac{C_r \rho Q}{D \mu} \qquad \text{[EQ 77.6]}$$

where

C_r = 0.1231 in `FIELD` units

(ρ in lb/ft.^{3}, Q in rb/day, D in ft., μ in cP

= 0.01474 in `METRIC` units

(ρ in kg/m^3, Q in rm^3/day, D in m, μ in cP)

= 0.03537 in `LAB` units

(ρ in g/cm^3, Q in rcc/hr, D in cm, μ in cP)

If there is more than one free phase flowing in the well, we assume the flow along the perforated length is homogeneous (that is, no slip between the phases). The homogeneous mixture density (mass flow rate / Q) is used in the equations for Re and δP_f, and the viscosity is the volumetric flow weighted average of the phase viscosities.

Wellbore friction is included in ECLIPSE by treating the connection head terms as additional variables. These are solved implicitly at the same time as the well solution variables. An implicit solution technique is necessary to avoid numerical instabilities. A pressure balance equation is solved between each pair of adjacent connections, equating the difference in their head terms with the frictional (and hydrostatic, if they are at different depths) pressure losses. The fluid flowing into a connection is treated as entering the connection at the middle of its perforated interval within the grid block. Thus the frictional pressure drop between a connection and its downstream neighbor depends on the production rates of that connection and all connections upstream, and the distance between the central points of the connection and its downstream neighbor.

Using the wellbore friction option

The maximum number of friction wells (NWFRIC) in the problem must be set using the FRICTION keyword in the RUNSPEC section.

You can designate one or more wells to be friction wells, with the keyword WFRICTN. This keyword also sets the data required by the friction calculation, for example, the tubing diameter and roughness, and the locations of the start and end of perforations within each connection. The WFRICTN keyword can only be used on one well at a time, so the keyword must be specified more than once if there is more than one friction well. Other "frictionless" wells can also exist in the run.

An alternative keyword WFRICTNL is provided for designating friction wells that are completed in local refined grids, when using the Local Grid Refinement option. The keyword is similar to WFRICTN, except that the local grid name should be specified in each record of connection data.

Friction wells can have the same set of controls and limits as ordinary wells. If a friction well has an associated VFP table for THP calculations, it is recommended to make the bottom hole reference point of the VFP table the same as that of the well (in WELSPECS item 5). This will ensure that all friction losses upstream of the VFP table's bottom hole reference point will be included in the well's connection head terms. A suitable position for the well's bottom hole reference point is the start of the perforations (that is the end nearest the wellhead).

At present the Wellbore Friction option cannot be run with Multi-segment Wells or the Polymer, Brine, Miscible, Solvent or GI model options in ECLIPSE 100.

Multi-lateral wells

Wells with branches emanating from the main stem can also be modeled with the Wellbore Friction option. The data structure is general enough even to allow for the case where a branch itself may have branches.

To reserve memory for multi-lateral wells it is necessary to set the second item of the FRICTION keyword in the RUNSPEC section. This item is the maximum number of branches (and for this purpose the main stem counts as a branch) which emanate from any one grid block connection. For a standard well that consists of only a main stem, we could enter

```
FRICTION
 1  1 /
```

as there is a maximum of 1 branch (that is the main stem itself) that emanates from any one connection. Item 2 is set equal to 1 by default. For a "trunk and branch" well as shown in Figure 77.1, we would enter

```
FRICTION
 1  2   /
```

Figure 77.1 A trunk and branch multi-lateral well

For the "bird's foot" type well shown in Figure 77.2, we would enter

```
FRICTION
 1   3    /
```

Figure 77.2 A bird's foot multi-lateral well

For a problem that involved all three of these wells then the maximum overall value should be used for the second argument,

```
FRICTION
 3   3   /
```

The geometry of a multi-lateral well is defined by the order in which the connections are entered in the keyword `WFRICTN` or `WFRICTNL`. `WFRICTNL` only differs from `WFRICTN` in that a local grid name is required in addition to the grid block coordinates to define each connection.

The first record of `WFRICTN` identifies the well, and sets the default tubing diameter, the roughness and the flow scaling factor.

The subsequent records deal with the connections making up the well, and have two purposes. The first is to define certain properties associated with the connections, including the tubing length from the well's bottom hole reference point to the start of the perforations in the grid block, and similarly the tubing length to the end of the perforations. It is possible to give the connection a different tubing diameter to the default value specified in the first record. Connections may be defined singly or in ranges. (A range is a contiguous row or column of grid blocks that are fully penetrated by the well.) Further details can be obtained from the keyword documentation.

The geometry of the well is defined by the order in which the connections are entered. A connection entered for the first time is assumed to be situated upstream (i.e. further from the heel of the well) of the previously entered connection. So, for a single branched well the connections should simply be ordered from the heel towards the toe.

If a connection is referenced again in `WFRICTN` after its first entry, either singly or as the **first** connection of a range, then it is taken to indicate that the subsequent connections form a branch that joins the well at this location. The connections in this branch should be ordered from the join out towards its toe. Thus whenever a connection is referenced for a second or subsequent time in `WFRICTN`, its entry is essentially a "pointer" to where the next connection joins the well. As its properties have already been defined in its first entry, it is only necessary to supply items 1-3 (or items 1-4 when using `WFRICTNL`) when a connection is re-entered as a pointer. If the remaining items in the record are specified (as they must be if the pointer is the first connection in a range describing a new branch), then they will only be applied to the connections being entered for the first time, and will not overwrite the properties of the pointer connection.

Note that there must therefore be a connection defined at each branch point. However, if the well is not actually perforated within these grid blocks the connection should be declared as `SHUT` in keyword `COMPDAT`.

As an example of how multi-lateral well geometry is defined, consider the well shown in Figure 77.3. We assume that k = 1 for all the connections. The order of defining this well is by no means unique, but one way is shown as follows:

Figure 77.3 Defining multi-lateral well geometry

```
 WFRICTN
-- well name   tubing diam      Roughness     Scale Fac
   'HORIZ'        0.25            1.0E-3         1.0    /
-- I  J  K   Tlen1   Tlen2  Direction  RangeEnd  Tubing diam
--Define the main stem
   6  2  1    2*              'Y'        3        1*    /
   6  5  1    400     500      3*                       /
   6  6  1    2*              'Y'        7        1*    /
--Pointer back to where first branch leaves main stem.
--(This connection was first entered as part of the range (6, 2-
3, 1)
   6  3  1                                              /
--Define this branch
   5  3  1    2*              'X'        4        0.17  /
   2  3  1    700     900      2*                 0.17  /
--Pointer back to where second branch leaves the main stem
   6  5  1                                              /
--Define this branch
   9  5  1    600     800      2*                 0.18  /
/
```

Remember that each of these connections must previously have been defined in keyword COMPDAT (or COMPDATL in a local grid). Note that connections must be defined at the branch points, blocks (6,3,1) and (6,5,1) in this example. If these are not perforated they should be declared as SHUT.

This method of defining multi-lateral well geometry is flexible enough to allow branches to have sub-branches. As a second example, consider the well shown in Figure 77.4.

Figure 77.4 A multi-lateral well with sub-branches

We again assume that k = 1 for all the connections, and we will also assume that the well is completed in every grid block. One way of defining the connections in WFRICTN is shown as follows:

```
 WFRICTN
-- well name   tubing diam      Roughness     Scale Fac
   'HORIZ2'       0.25            1.0E-3         1.0    /
-- I  J  K   Tlen1   Tlen2  Direction  RangeEnd  Tubing diam
--Define the main stem (9, 1-8, 1)
   9  1  1    2*               'J'         8        1*  /
--Pointer to lower branch point on main stem
   9  7  1                                              /
--Define connections in the left branch (8-6, 7, 1)
   8  7  1    2*               'I'         6        1*  /
--and the same for the right branch (10-12, 7, 1)
   9  7  1                                              /
  10  7  1    2*               'I'        12        1*  /
--Pointer to the upper branch point on the main stem
   9  3  1                                              /
--Define the right branch (10-13, 3, 1)
  10  3  1    2*               'I'        13        1*  /
--and the same for the left branch (8-2, 3, 1)
   9  3  1                                              /
   8  3  1    2*               'I'         2        1*  /
--Pointer to the branch point off this branch
   3  3  1                                              /
--Define the single connection in this sub-branch
   3  2  1    1000    1020      3*                      /
/
```

This definition of the well can be shortened by incorporating the pointers into the range of connections:

```
 WFRICTN
-- well name   tubing diam      Roughness     Scale Fac
   'HORIZ2'       0.25            1.0E-3         1.0    /
-- I  J  K   Tlen1   Tlen2  Direction  RangeEnd  Tubing diam
--Define the main stem (9, 1-8, 1)
   9  1  1    2*               'J'         8        1*  /
--Define the lower left branch (9-6, 7, 1)
   9  7  1    2*               'I'         6        1*  /
--and the same for the lower right branch (9-12, 7, 1)
   9  7  1    2*               'I'        12        1*  /
--Define the upper right branch (9-13, 3, 1)
   9  3  1    2*               'I'        13        1*  /
--and the same for the upper left branch (9-2, 3, 1)
   9  3  1    2*               'I'         2        1*  /
--Pointer to the branch point off this branch
   3  3  1                                              /
--Define the single connection in this sub-branch
   3  2  1    1000    1020      3*                      /
/
```

Occasionally when opening a friction well, and in particular a multi-lateral friction well, the well solution may have difficulty in converging. This is because the initial “guess” of the pressures used to start off the Newton iteration may be poor. This can often be solved by just allowing the well to take more iterations, that is, increasing MXWSIT in the third record of the TUNING keyword. However, this may not be effective in all cases, and when this is so the OPTIONS keyword switch 55 may be tried. This damps the Newton iteration, improving its convergence properties. It should be noted that after the problem has been solved, that is at the next time-step, it is very unlikely that the Newton method will need damping. Now the use of the 55th OPTIONS switch will hinder convergence rather than help it. Hence the switch should be set to zero as soon as possible after the well has been successfully opened.

Workovers

When a workover with an action ‘+CON’ is performed on a friction well, that is ‘close the worst-offending connection and all below it’ (see Item 7 of the WECON keyword, for example), this is interpreted as meaning ‘close the connection and all connections **upstream** of it’. Thus instead of selecting the connections to close according to their depth, they are selected according to their length down the tubing - all connections further from the wellhead are closed. In multi-lateral wells, the branch topology is taken into account so that only branches subordinate to the worst-offending connection are closed off.

Graphical presentation of friction effects in horizontal wells

A useful plot for showing up the effects of friction along the perforated length of a horizontal well is a graph of

> inflow rate across a length L of perforated tubing, starting from the downstream end of the perforations (nearest the wellhead) and extending a length L towards the far end

versus

> the length L of perforated tubing considered.

The graph obviously starts from the origin (zero inflow over zero length of tubing), and ends at the point: x = total perforated tubing length, y = well's production rate.

For a well without friction, assuming the conditions in the reservoir are uniform across the perforated length, the graph will be approximately a straight line. (In fact the inflow rate per unit length will be slightly higher at the ends than in the middle.) The effect of friction is to decrease the inflow rate per unit length at distances further from the start of the perforations. This causes the graph to curve concave-downwards. In extreme cases the graph could level off at large tubing lengths, indicating that a portion of the tubing at the far end of the perforations is not contributing much to the productivity of the well.

Figure 77.5 shows the graph for three values of the tubing diameter in the horizontal section. The well is under tubing head pressure control. The well bore diameter (and thus the connection transmissibility factors) and the tubing diameter between the wellhead and the start of the horizontal section have been kept constant, so the difference between the results is purely due to friction in the perforated length. The `NOFRIC` case was run with a tubing diameter large enough to make the friction effects negligible. The `MEDFRIC` case (run with 2 inch diameter tubing) shows friction effects just becoming noticeable. The extreme `HIFRIC` case (run with 1.25 inch diameter tubing) shows that friction is severely degrading the performance of the well.

Figure 77.5 Friction effects in a horizontal well

The GRAF program has been adapted to produce graphs of this type. To make one of these graphs, carry out the following steps:

1 In the `SUMMARY` section of the ECLIPSE `DATA` file, make sure you have included the flow rates of **all** the connections in the horizontal well. For example, to plot the oil flow rate from a horizontal well called HORIZ with 6 connections use:

```
SUMMARY
COFR
 'HORIZ' 3 5 6 /
 'HORIZ' 4 5 6 /
 'HORIZ' 5 5 6 /
 'HORIZ' 6 5 6 /
 'HORIZ' 7 5 6 /
 'HORIZ' 8 5 6 /
/
```

The position of each connection (that is the length down the tubing from the bottom hole reference point) is automatically written to the `SUMMARY` file.

2 When the SUMMARY files produced from the run are loaded into GRAF, you are prompted:

```
The summary files contain Horizontal Well data
Do you wish to process this data?
Y/N (RETURN for N)
```

Reply Y to process the data for this type of graph.

```
Define times for which vectors are required
R-Report steps only A-All ministeps
(RETURN for R)
```

Reply R if you want to plot this data at report times only, or

A if you require this data at every mini-step in the SUMMARY file.

```
Select horizontal well vectors
I-Inflow along well bore
C-Connection vectors A-All
(RETURN for I)
```

Reply I to prepare data for this type of plot (integrated inflow rate versus length of perforations)

C to prepare data for plots of other connection quantities (for example, connection flow rate or connection pressure) versus length from bottom hole reference point

A to prepare data for all types of plot

```
Does data include multilateral wells? Y/N
(RETURN for N)
```

Reply Y if multi-lateral wells are present in the model (see "Multi-lateral friction well plots in GRAF" on page 1175 for more information)

N if all wells are single-borehole wells

```
Enter well name or type A-All, H-Help
S-Select from list or Q-Quit (RETURN for A)
```

Reply with the name of the horizontal well you wish to plot, or

A to prepare data for all the wells in the SUMMARY file

H to see a list of wells available

S to be prompted (Y or N) for each well

Q to abandon processing the horizontal well data.

3 GRAF then generates a new set of vectors from the data in the SUMMARY files.

The length down the tubing from the bottom hole reference point to each connection in the well is stored in the vector WOFR_L.

The inflow rate between the start of the perforations and the end of each connection is stored in the vectors WOFR0, WOFR1, WOFR2 etc., for time steps 0, 1, 2 etc. (up to 999). This is essentially the sum of the connection flows from the start of the perforations down to each connection.

To generate a horizontal well plot showing the conditions at time step 5 (say):

Type `WOFR_L` when prompted for the x-axis vector, and

`WOFR5` when prompted for the y-axis vector.

The actual time associated with each time step is shown in the index of vectors (option 0.2.1).

Plots of other connection data quantities versus length from the bottom hole reference point can be produced if you replied with `C` or `A` to the prompt "`Select horizontal well vectors`" described above. Any connection quantity (beginning with `C`) in the summary file can be selected by preceding it with a `W` and ending it with the time step number. The corresponding length vector ends with `_L`. For example, to plot the connection pressure (`CPR`) at the second time step versus length from the bottom hole reference point:

Type `WCPR_L` when prompted for the x-axis vector, and

`WCPR2` when prompted for the y-axis vector.

Similarly a plot of connection water cut (`CWCT`) at time step 4 vs. length would have x-axis = `WCWCT_L` and y-axis = `WCWCT4`.

If the well is situated in a Local Grid Refinement, the above steps will only produce data for the coarse grid connections. To obtain data for the refined grid connections, the `SUMMARY` section must contain the keywords to extract the flows of all the connections in the local refined grid. For example, if the well HORIZ has 6 connections in the local grid called LGR1, to extract the connection oil flow rates use:

```
LCOFR
 'LGR1' 'HORIZ' 3 5 6 /
 'LGR1' 'HORIZ' 4 5 6 /
 'LGR1' 'HORIZ' 5 5 6 /
 'LGR1' 'HORIZ' 6 5 6 /
 'LGR1' 'HORIZ' 7 5 6 /
 'LGR1' 'HORIZ' 8 5 6 /
/
```

When local grid data is processed for horizontal well plots, the vector names are the same as those for the global grid quantities, but preceded with an `L`. Thus the tubing length data for each connection in the local grid is stored in the vector `LWOFR_L`, and the integrated flow data at each global time step is stored in the vectors `LWOFR0`, `LWOFR1`, etc. up to `LWOFR999`. To generate a horizontal well plot showing the conditions at global time step 5:

Type `LWOFR_L` when prompted for the x-axis vector, and

`LWOFR5` when prompted for the y-axis vector.

Similarly a plot of local grid connection pressure (`LCPR`) at time step 2 versus length would have x-axis = `LWCPR_L` and y-axis = `LWCPR2`.

Multi-lateral friction well plots in GRAF

The horizontal well option in GRAF has been extended to allow you to split a multi-lateral well into branches.

Each branch is defined by a unique name (which is treated like a well name in GRAF, for vector identification purposes) and a list of connections. Branches in amalgamated LGRs can have connections in more than one LGR, although GRAF cannot mix global and local connections on the same branch.

Interactive input

You can choose whether to treat wells as multi-lateral in response to the prompt:

```
Does data include multilateral wells? Y/N
(RETURN for N)
```

If the response is `YES`, you are prompted to select wells as described above. After selecting a well, the prompts are as follows:

```
Is this a multilateral well? Y/N (RETURN for Y)
```

If the response is `NO`, the well is handled as a standard single-branch horizontal well. Otherwise, GRAF reports the number of connections found and prompt for branches:

```
Enter branch name or Q-Quit (RETURN for bbb)
```

where bbb is a default branch name made up from the current well name and the branch count, for example `WELLA_3` for the third branch of `WELLA`.

Enter `Q` to stop defining branches for this well.

The branch name is checked for uniqueness before it is accepted. If LGR data are available for the well, you can choose whether to define the branch by global or local connections. Connections are selected by responding to the prompts:

```
Add connection i j k LGR lll to branch?
Y/N or E to end list for this branch (RETURN for Y)
```

If a connection has already been used for another branch of the same well, the default response is changed to `NO`. However, a connection can be added to any number of branches, as required. (For example you may wish to plot connections for the entire route along the well from each branch tip.)

GRF input

For input from a `GRF`, the load commands are of the form:

```
LOAD SUMMARY sss  ORIGIN ooo  HORIZ MULTI  HTIME REPORT  HVEC ALL
LOADOPT  MLWELL www1  BRANCH bbb1
  CONNECTION   i1 j1 k1   [ LGR lll ]
  .....
LOADOPT  MLWELL www1  BRANCH bbb2
  CONNECTION   ii1 jj1 kk1
  .....
LOADOPT  HWELL www2
```

Use of multi-lateral wells is indicated in the LOAD SUMMARY command by setting the HORIZ option to MULTI and each branch is then defined by a LOADOPT command and a list of connections. Single-borehole wells can be specified by using LOADOPT HWELL as usual.

Plotting

The vectors for plotting inflow versus length are assembled in the same way as for horizontal well plots, for example WOFR_L, WOFR0, WOFR1... The branch names are used in place of the well name for each vector.

Production flow rates from upstream connections

ECLIPSE can output vectors (L)C(O,W,G)FRU, representing the oil, water and gas production flow rates from all connections upstream of (and including) a specified connection.

The upstream flow vectors COFRU, etc. cannot be used to create inflow plots in GRAF, as they already contain summed flows. GRAF identifies these vectors and suppresses the summation operation automatically, if you request Inflow vectors.

The naming convention for the vectors created from COFRU etc. gives new vector names in the form WOFRU_L, WOFRU1, etc.

Plotting WOFRU1 versus WOFRU_L (for example) will give a plot of upstream flow versus the distance of the connection from the bottom hole reference point. This shows maximum production total at distance zero along the well, unlike the economic plots obtained by constructing inflow vectors in GRAF.

Summary of keywords

RUNSPEC section

The keywords are:

Table 77.1 RUNSPEC section

Keyword	Description
FRICTION	Enables the Wellbore Friction option and sets array dimensions.

SUMMARY section

The following items are relevant to the Wellbore Friction model:

Table 77.2 Wellbore friction SUMMARY output controls

Connection	Information
COFRU	Sum of connection oil flow rates upstream of, and including, this connection
CWFRU	Sum of connection water flow rates upstream of, and including, this connection
CGFRU	Sum of connection gas flow rates upstream of, and including, this connection
LCOFRU	As COFRU but for local grids
LCWFRU	As CWFRU but for local grids
LCGFRU	As CGFRU but for local grids

The local grid keywords (LCOFRU etc.) should be used if the well is situated in a local grid.

SCHEDULE section

The keywords are:

Table 77.3 SCHEDULE section

Keyword	Description
OPTIONS	Switch 55 may be used to damp the Newton iterations if well convergence problems arise.
WFRICTN	Designates a well as a friction well, and defines its tubing properties and the position in the tubing of the well connections with the grid cells.
WFRICTNL	Performs the same task as WFRICTN, but should be used if the well is situated in a local grid.

Example problem

This dataset describes the 7th SPE Comparative Solution Project [Ref. 73] - a horizontal well with friction. The dataset describes CASE 4B, having a 2100 ft. horizontal well completed across 7 grid blocks in the x-direction.

RUNSPEC

```
RUNSPEC
TITLE
 7th SPE Benchmark, Case 4b
DIMENS
 9    9    6  /
NONNC
OIL
WATER
GAS
DISGAS
FIELD
TABDIMS
 1    1   15   15    1   15  /
WELLDIMS
 2    9    1    2 /
FRICTION
 1 /
START
1 'JAN' 1990  /
NSTACK
 50 /
FMTOUT
FMTIN
UNIFOUT
UNIFIN
```

GRID

```
GRID        ========================================================
=
DXV
  9*300  /
DYV
  620  400  200  100  60  100  200  400  620  /
DZ
  81*20  81*20  81*20  81*20  81*30  81*50  /
PERMX
  486*3000 /
PERMY
  486*3000 /
PERMZ
  486*300 /
PORO
  486*0.2 /
EQUALS
'TOPS'  3590  1 9  1 9  1 1  /
/
```

PROPS

```
PROPS     ==========================================================
-- Use Stone's Method 2 for 3-phase oil relative permeability
STONE2

SWOF
-- Sw    Krw    Krow     Pcow
  0.22   0.0    1.0000   6.30
  0.3    0.07   0.4000   3.60
  0.4    0.15   0.1250   2.70
  0.5    0.24   0.0649   2.25
  0.6    0.33   0.0048   1.80
  0.8    0.65   0.0      0.90
  0.9    0.83   0.0      0.45
  1.0    1.0    0.0      0.0    /
SGOF
-- Sg    Krg    Krog     Pcgo
  0.0    0.0    1.00     0.0
  0.04   0.0    0.60     0.2
  0.1    0.022  0.33     0.5
  0.2    0.1    0.10     1.0
  0.3    0.24   0.02     1.5
  0.4    0.34   0.0      2.0
  0.5    0.42   0.0      2.5
  0.6    0.5    0.0      3.0
  0.7    0.8125 0.0      3.5
  0.78   1.0    0.0      3.9    /
```

```
PVTW
-- Pref    Bw       Cw       Vw    Viscosibility
   3600  1.00329   3.0E-6   0.96         0  /
ROCK
-- Pref     Cr
   3600    4.0D-6   /
DENSITY
--  oil   water   gas
    45.0  62.14   0.0702  /
PVDG
-- Pgas   Bg    Vg
    400  5.9   0.013
    800  2.95  0.0135
   1200  1.96  0.014
   1600  1.47  0.0145
   2000  1.18  0.015
   2400  0.98  0.0155
   2800  0.84  0.016
   3200  0.74  0.0165
   3600  0.65  0.017
   4000  0.59  0.0175
   4400  0.54  0.018
   4800  0.49  0.0185
   5200  0.45  0.019
   5600  0.42  0.0195   /
PVCO
-- Pbub   Rs     Bo     Visco    Co     CVo
    400  0.165  1.012   1.17   1.0E-5   0.0
    800  0.335  1.0255  1.14     2*
   1200  0.500  1.038   1.11     2*
   1600  0.665  1.051   1.08     2*
   2000  0.828  1.063   1.06     2*
   2400  0.985  1.075   1.03     2*
   2800  1.130  1.087   1.00     2*
   3200  1.270  1.0985  0.98     2*
   3600  1.390  1.11    0.95     2*
   4000  1.500  1.12    0.94     2*
   4400  1.600  1.13    0.92     2*
   4800  1.676  1.14    0.91     2*
   5200  1.750  1.148   0.9      2*
   5600  1.810  1.155   0.89     2*
  /
PMAX
 6000  /
RPTPROPS
 'SOF2'   'SWFN'   'SGFN'   'PVTO'
 'PVTW'   /
```

SOLUTION

```
SOLUTION  ========================================================
EQUIL
-- Dep   Pres  Dwoc  Pcwoc  Dgoc  Pcgoc  Rsvd  Rvvd  Fipc
   3600  3600  3700   0.0   3590   0.0    1      0     0  /
RSVD
-- Dep   Rs
   3500  1.41
   3750  1.41  /
RPTSOL
 'PRES'  'SOIL'  'SWAT'  'SGAS'  'RS'  /
```

SUMMARY

```
SUMMARY   ========================================================
FOPR
FGPR
FWPR
FOPT
FGPT
FWPT
FWCT
FGOR
WBHP
'PROD' /
CPR
'PROD'   6 5 1 /
/
FPR
WIR
FWIT
RUNSUM
```

SCHEDULE

```
SCHEDULE   ========================================================
RPTSCHED
 'WELLS=2'  'SUMMARY=2'  'WELSPECS'  /
```

Specify producer and injector

```
WELSPECS
-- Name  Group  I  J  Dbh   Phase
  'PROD'  'G'   8  5  3600  'OIL'  /
  'INJ'   'G'   8  5  3725  'WAT'  /
/
```

Specify well connections

```
COMPDAT
-- Name   I  J  K1-K2  Op/Sh  SatTab  ConFact  Diam  KH S D  Dirn
  'PROD'  2  5  1  1  'OPEN'   2*              0.375   3*    'X'  /
  'PROD'  3  5  1  1  'OPEN'   2*              0.375   3*    'X'  /
  'PROD'  4  5  1  1  'OPEN'   2*              0.375   3*    'X'  /
  'PROD'  5  5  1  1  'OPEN'   2*              0.375   3*    'X'  /
  'PROD'  6  5  1  1  'OPEN'   2*              0.375   3*    'X'  /
  'PROD'  7  5  1  1  'OPEN'   2*              0.375   3*    'X'  /
  'PROD'  8  5  1  1  'OPEN'   2*              0.375   3*    'X'  /
  'INJ'   1  5  6  6  'OPEN'   1*     243.4  /
  'INJ'   2  5  6  6  'OPEN'   1*     243.4  /
  'INJ'   3  5  6  6  'OPEN'   1*     243.4  /
  'INJ'   4  5  6  6  'OPEN'   1*     243.4  /
  'INJ'   5  5  6  6  'OPEN'   1*     243.4  /
  'INJ'   6  5  6  6  'OPEN'   1*     243.4  /
  'INJ'   7  5  6  6  'OPEN'   1*     243.4  /
  'INJ'   8  5  6  6  'OPEN'   1*     243.4  /
  'INJ'   9  5  6  6  'OPEN'   1*     243.4  /
/
```

Specify the friction data for the producer.

```
WFRICTN
-- Name   Diam   Roughness
  'PROD'  0.375  3.75E-4  /
-- I J K  Tlen1  Tlen2
   8 5 1   0.0    300  /
   7 5 1   300    600  /
   6 5 1   600    900  /
   5 5 1   900   1200  /
   4 5 1  1200   1500  /
   3 5 1  1500   1800  /
   2 5 1  1800   2100  /
/
```

Specify the production well constraints

```
WCONPROD
-- Name   Op/Sh  CtlMode  Orat  Wrat  Grat  Lrat  Resv  Bhp
  'PROD'  'OPEN'  'LRAT'   3*               9000  1*    1500  /
/
```

Specify the injection well constraints

```
WCONINJE
-- Name  Phase   Op/Sh  CtlMode  Srat  Resv  Bhp
  'INJ'  'WAT'  'OPEN'  'RATE'   6000  1*    5000  /
/
```

Set 50 day maximum time step and increase linear iteration limit to 40

```
TUNING
1 50 /
/
2* 40 /
```

Time steps and reports

```
TSTEP
 4*100  /
RPTSCHED
 'PRES'  'SWAT'  'SGAS'  'FIP'  'WELLS=2'
 'SUMMARY=2'  'CPU=2'  'WELSPECS'  /
TSTEP
 100  /
RPTSCHED
 'WELLS=2'  'SUMMARY=2'  'WELSPECS'  /
TUNING
25 25 /
/
2* 50 /
TSTEP
 2*100  /
TUNING
50 50 /
/
2* 50 /
TSTEP
 2*100  /
RPTSCHED
 'PRES'  'SWAT'  'SGAS'  'FIP'  'WELLS=2'
 'SUMMARY=2'  'CPU=2'  'WELSPECS'  /
TSTEP
 100  /
RPTSCHED
 'WELLS=2'  'SUMMARY=2'  'WELSPECS'  /
TSTEP
 4*100  /
RPTSCHED
 'PRES'  'SWAT'  'SGAS'  'FIP'  'WELLS=2'
 'SUMMARY=2'  'CPU=2'  'WELSPECS'  /
TSTEP
 100  /
END
```

Well Inflow Performance

Chapter 78

Introduction

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

This chapter outlines the relationships that ECLIPSE uses to calculate well inflow performance.

We define the flow path between the well bore and a single reservoir grid block as a "connection". The flow rate of a phase p (oil, water or gas) across a connection j is given by the Inflow Performance Relationship.

Inflow performance relationship in ECLIPSE 100

In ECLIPSE 100 the inflow performance relationship is written in terms of the volumetric production rate of each phase at stock tank conditions:

$$q_{p,j} = T_{wj}\, M_{p,j}\, (P_j - P_w - H_{wj}) \qquad \text{[EQ 78.1]}$$

where

$q_{p,j}$ is the volumetric flow rate of phase p in connection j at stock tank conditions. The flow is taken as positive from the formation into the well, and negative from the well into the formation.

T_{wj} is the connection transmissibility factor, defined below.

$M_{p,j}$ is the phase mobility at the connection.

P_j is the nodal pressure in the grid block containing the connection.

P_w is the bottom hole pressure of the well.

H_{wj} is the well bore pressure head between the connection and the well's bottom hole datum depth. $P_w + H_{wj}$ is thus the pressure in the well at the connection j, which we call the "connection pressure".

The calculation of the terms in this relationship is described in the following sections of this chapter.

Inflow performance relationship in ECLIPSE 300

In ECLIPSE 300 the inflow performance relationship is written in terms of the volumetric production rate of each phase at reservoir conditions:

$$\hat{q}_{p,j} = T_{wj}\left(\frac{k_{rp,j}}{\mu_{p,j}}\right)(P_j - P_w - H_{wj}) \qquad \text{[EQ 78.2]}$$

where

$\hat{q}_{p,j}$ is the volumetric flow rate of phase p in connection j at reservoir conditions.

$\mu_{p,j}$ is the phase viscosity at the connection.

$k_{rp,j}$ is the phase relative permeability at the connection.

The surface volumetric rate is

$$q_{p,j} = \hat{q}_{p,j}\, b_{p,j} \qquad \text{[EQ 78.3]}$$

where $b_{p,j} = \frac{1}{B_{p,j}}$, and $B_{p,j}$ is the phase formation volume factor.

The molar rate is obtained by multiplying by the phase reservoir molar densities, $b^m_{p,j}$. The molar rate of component *i* is then:

$$q_{i,j} = x_{i,j}\, b^m_{o,j}\, q_{o,j} + y_{i,j}\, b^m_{g,j}\, q_{g,j} \qquad \text{[EQ 78.4]}$$

where:

q_i is the component i molar rate

x_i is the liquid mole fractions

y_i is the vapor mole fractions

The connection transmissibility factor

For brevity, we refer to this quantity as the "connection factor". It depends on the geometry of the connecting grid block, the well bore radius, and the rock permeability. Its value may be specified directly by the engineer, or it can be calculated by the program using one of the formulae below.

Cartesian grids

In a Cartesian grid, ECLIPSE uses the relationship

$$T_{wj} = \frac{c\theta Kh}{\ln(r_o/r_w) + S} \qquad \text{[EQ 78.5]}$$

where

c is a unit conversion factor (0.001127 in field units, 0.008527 in metric units, 3.6 in lab units).

θ is the angle of the segment connecting with the well, in radians. In a cartesian grid its value is 6.2832 ($= 2\pi$), as the connection is assumed to be in the center of the grid block. For wells located on an edge (or a corner) of a cartesian grid, e.g. in a sector model or symmetry element, use keyword `WPIMULT` after `COMPDAT` to scale the resulting connection factors by 0.5 (or 0.25).

Kh is the effective permeability times net thickness of the connection. For a vertical well the permeability used here is the geometric mean of the x- and y-direction permeabilities, $K = (K_x K_y)^{1/2}$.

r_o is the "pressure equivalent radius" of the grid block, defined below.

r_w is the well bore radius.

S is the skin factor.

The pressure equivalent radius of the grid block is defined as the distance from the well at which the local pressure is equal to the nodal average pressure of the block. In a cartesian grid we use Peaceman's formula, which is applicable to rectangular grid blocks in which the permeability may be anisotropic. The well is assumed to penetrate the full thickness of the block, through its center, perpendicularly to two of its faces.

$$r_o = 0.28\frac{\left[D_x^2\left(\frac{K_y}{K_x}\right)^{1/2} + D_y^2\left(\frac{K_x}{K_y}\right)^{1/2}\right]^{1/2}}{\left(\frac{K_y}{K_x}\right)^{1/4} + \left(\frac{K_x}{K_y}\right)^{1/4}} \qquad \text{[EQ 78.6]}$$

where D_x and D_y are the x- and y- dimensions of the grid block, and K_x and K_y are the x- and y- direction permeabilities.

[EQ 78.5] and [EQ 78.6] are intended for vertical wells. Horizontal wells may penetrate the block in either the x- or y- direction, and the appropriate components of permeability and block dimensions are substituted in these equations. For a well penetrating in the x-direction, for example, the quantities K_y, K_z, D_y, D_z will be used in [EQ 78.6], and in [EQ 78.5] $Kh = D_x(K_y K_z)^{1/2}$.

ECLIPSE 300

ECLIPSE 300 requires the HWELLS keyword to be entered in the RUNSPEC section if there are well connections penetrating in a direction other than z; this is not necessary in ECLIPSE 100.

A net-to-gross ratio applied to a grid block containing a horizontal well will not affect Kh, but will still affect r_o through the reduction in the value of D_z.

Radial grids

In radial geometry, the formula used to calculate T_{wj} depends on the location of the connecting grid block.

When the connection is situated in one of the inner-most grid blocks (i = 1), it is assumed to lie on the inner boundary of the block. The connection factor is

$$T_{wj} = \frac{c\theta Kh}{\frac{r_2^2}{r_2^2 - r_w^2}\ln(r_2/r_w) - 0.5 + S} \qquad \text{[EQ 78.7]}$$

where

θ is the segment angle of the grid block in radians,

r_2 is the block's outer radius.

If a connection is situated in one of the outer-most blocks (i = NDIVIX), it is assumed to occupy the outer boundary of the block for the purpose of applying boundary conditions. The corresponding connection factor is

$$T_{wj} = \frac{c\theta Kh}{\frac{r_1^2}{r_2^2 - r_1^2}\ln(r_1/r_2) + 0.5 + S} \qquad \text{[EQ 78.8]}$$

where r_1 and r_2 are the inner and outer radii of the block.

This enables you to use a BHP-controlled well to apply constant pressure outer boundary conditions. If on the other hand you wish to model a real well in an outer block, the calculated connection factor will be much too large. In this case, you should either supply an appropriate connection factor value in item 8 of the COMPDAT keyword, or add an extra ring of inactive cells so that the well is no longer in an outer block.

If a well is completed in any other block (1 < i < NDIVIX), its connection factor is calculated as if it were in a cartesian grid block with $D_x = r_2 - r_1$ and $D_y = \theta(r_1 + r_2)/2$

Fractured completions

ECLIPSE 300

When using one of the built-in formulae above to calculate the connection factor, the engineer can supply a constant value for the skin factor *S*, to represent the effects of formation damage or stimulation, partial penetration, well bore deviation etc. However, ECLIPSE 300 provides an additional formula to calculate the connection factor for cases where a vertical fracture penetrates the full width of the cell in the x- or y- direction.

The fractured completion formula is selected by entering FX or FY in item 13 of COMPDAT. The FWELLS keyword must be entered in the RUNSPEC section if any wells use this option.

In the case of FX, the completion connection factor is

$$T_{wj} = \frac{8cK_yD_xD_z}{D_y + SD_x/\pi} \qquad \text{[EQ 78.9]}$$

In the case of FY, the completion connection factor is

$$T_{wj} = \frac{8cK_xD_yD_z}{D_x + SD_y/\pi} \qquad \text{[EQ 78.10]}$$

In both cases we assume uniform flow into the fracture at the center of the cell. The mean distance of flow is $D_y/4$ or $D_x/4$, over an area of $2D_xD_z$ or $2D_yD_z$.

The flow-dependent skin factor

When ECLIPSE uses one of the built-in formulae above to calculate the connection factor, you can supply a constant value for the skin factor S, to represent the effects of formation damage or stimulation, partial penetration, well bore deviation etc. In addition, a flow-dependent contribution can be included to model non-Darcy effects in gas flow near the well,

$$S \rightarrow S + D|q_{fg}| \qquad \text{[EQ 78.11]}$$

q_{fg} represents the flow rate of free gas through the connection. The D-factor is usually obtained from measurements on the well, and is multiplied by the well's free gas flow rate Q_{fg} to give the non-Darcy contribution to the skin factor. But in reality the effect of non-Darcy flow at a connection depends on the free gas flow rate through the connection itself rather than the well as a whole. Accordingly ECLIPSE calculates the non-Darcy skin from the connection flows, as in [EQ 78.11], rather than the well flow. If the well has more than one connection, the D-factor must be scaled to apply to the connection flows instead of the well flow. The scaling is performed by the program; the engineer specifies the measured D-factor for the well (with keyword `WDFAC`) and the program will scale it for each open connection. The scaling process aims to give each connection initially the same non-Darcy skin, by setting the D-factors in inverse proportion to the gas flow rate. The calculation assumes that initially the free gas mobility and the drawdown are the same in each connection, so that

$$D_j = D_w \frac{\sum_i T_{wi}}{T_{wj}} \qquad \text{[EQ 78.12]}$$

where

D_j is the scaled D-factor for the connection j,

D_w is the measured D-factor for the well, and the

$\sum_i T_{wi}$ term denotes the sum of the connection transmissibility factors for all the connections in the well that are open at the time the D-factor is specified.

If you do not wish the D-factors to be scaled, you may enter them **with a negative sign** in item 12 of the `COMPDAT` keyword. The program will simply reverse the sign without applying scaling. (Entering a positive value in this item was a former method of specifying the D-factor for the whole well in ECLIPSE 100, before the `WDFAC` keyword was implemented; so positive values are scaled as if they were entered with `WDFAC`.)

ECLIPSE 300

D-factors may also be calculated by the simulator for each connection in a well using an expression defined in the `WDFACCOR` keyword. This expression is based on a correlation for the coefficient of inertial resistance, β, in terms of the permeability and porosity of the connected grid blocks. Please refer to the keyword for more details.

The program calculates the connection factor without the contribution from the D-factor, and the D-factor is transformed into a multiplying factor D_m to be applied to the free gas mobility

$$D_m = \frac{G}{G + D|q_{fg}|} \qquad \text{[EQ 78.13]}$$

where G is a geometric factor:

- For the Cartesian case:

$$G = \ln\left(\frac{r_o}{r_w}\right) + S \qquad \text{[EQ 78.14]}$$

- For the inner cell radial case:

$$G = \frac{r_2^2}{r_2^2 - r_w^2}\ln\left(\frac{r_2}{r_w}\right) - 0.5 + S \qquad \text{[EQ 78.15]}$$

- For the outer cell radial case:

$$G = \frac{r_1^2}{r_2^2 - r_1^2}\ln\left(\frac{r_1}{r_2}\right) + 0.5 + S \qquad \text{[EQ 78.16]}$$

- For the fractured completion cases:
  - Fracture in x-direction:

$$G = \pi\frac{\Delta_y}{\Delta_x} + S \qquad \text{[EQ 78.17]}$$

  - Fracture in y-direction:

$$G = \pi\frac{\Delta_x}{\Delta_y} + S \qquad \text{[EQ 78.18]}$$

Since the multiplying factor itself depends upon the free gas flow rate, ECLIPSE uses a calculation that provides a completely self-consistent pair of values for these two quantities. The modified inflow performance relationship for the well connection can be written as

$$q_{fg} = T_w D_m M_{fg}\,\Delta P \qquad \text{[EQ 78.19]}$$

where ΔP is the drawdown into the connection.

Substituting [EQ 78.13] for D_m gives the quadratic equation

$$T_w M_{fg}\,\Delta P = q_{fg} + \left(\frac{D}{G}\right) q_{fg}\left|q_{fg}\right| \qquad \text{[EQ 78.20]}$$

that may readily be solved for $\left|q_{fg}\right|$, thus enabling D_m to be obtained.

ECLIPSE 300

Note that ECLIPSE 300 uses the reservoir gas phase flow rate converted to surface volume, rather than the stock tank gas rate for the completion. The latter would involve a separator flash within the above calculations, and be dubious physically - the flow into the well should not depend on the subsequent separation process. At near critical conditions the oil phase may be sufficiently close to the gas phase that a D-factor should also be applied to oil inflow - this is not currently done in ECLIPSE 300 as oil does not generally show non-Darcy flow effects.

The phase mobilities

The term $M_{p,j}$ in [EQ 78.1] represents the mobility of the phase p at the connection. In producing connections, where the flow is from the formation into the well bore, the mobility depends on the conditions in the grid block containing the connection. The mobility of a free phase (free oil, water, or free gas) is given by

$$M_{fp,j} = k_{p,j}\,\lambda_{p,j} \qquad \text{[EQ 78.21]}$$

where

$k_{p,j}$ is the relative permeability of the phase,

$\lambda_{p,j}$ is defined by

$$\lambda_{p,j} = \frac{1}{B_{p,j}\,\mu_{p,j}} \qquad \text{[EQ 78.22]}$$

$\mu_{p,j}$ is the phase viscosity, and

$B_{p,j}$ is the phase formation volume factor.

The quantities in [EQ 78.21] and [EQ 78.22] are determined from the pressure and saturations in the grid block. However, the relative permeability can be obtained using a separate saturation functions table that the engineer can supply to take account of the distribution of fluids and the location of the perforations within the grid block.

ECLIPSE 100

The total gas and total oil mobilities are obtained by including the dissolved gas and vaporized oil content of the phases

$$\begin{aligned} M_{o,j} &= M_{fo,j} + R_{v,j} M_{fg,j} \\ M_{g,j} &= M_{fg,j} + R_{s,j} M_{fo,j} \end{aligned} \qquad \text{[EQ 78.23]}$$

If a flow-dependent skin factor is used, the free gas mobility is multiplied by the factor D_m defined in [EQ 78.13].

$$M_{fg,j} \rightarrow D_m M_{fg,j} \qquad \text{[EQ 78.24]}$$

Special inflow equations

ECLIPSE 100

There is an option to use one of three special inflow equations to provide a more accurate model of the flow of gas into the well: the Russell Goodrich equation, the dry gas pseudo-pressure equation and the generalized pseudo-pressure method. These are all methods of taking into account the pressure-dependence of $\lambda_{g,j}$ between the grid block pressure and the well bore pressure. The generalized pseudo-pressure equation alters both the gas and oil mobilities, and takes account also of the effects of condensate dropout. The 8th item of keyword `WELSPECS` selects the method to use for each well.

Russell Goodrich

ECLIPSE 100

The familiar form of the Russell Goodrich equation shows the flow rate to vary as the difference between the squares of the grid block and well bore pressures. But it can be written in the form of [EQ 78.1] if the term $\lambda_{g,j}$ defined in [EQ 78.22] is evaluated at the average of the grid block pressure and the well bore pressure at the connection, that is $(P_j + P_w + H_{wj})/2$. The relative permeability is evaluated at the grid block conditions.

Dry gas pseudo-pressure

ECLIPSE 100

In the pseudo-pressure equation, the gas flow rate varies as the difference between the "real gas pseudo-pressure" values at the grid block and well bore pressures. This also can be written in the form of [EQ 78.1] by setting $\lambda_{g,j}$ equal to its integrated average value between the pressures P_j and $P_w + H_{wj}$. This representation avoids the need to calculate tables of pseudo-pressure vs. pressure. The integration is performed by applying the trapezium rule between each intermediate pair of nodal pressure values in the gas PVT table. This essentially treats λ_g as a linear function of P between adjacent nodal pressure values. The relative permeability is evaluated at the grid block conditions.

At low values of the drawdown the Russell Goodrich and pseudo-pressure equations will give approximately the same results. When the drawdown is large enough to span a significantly non-linear segment of the graph of λ_g versus P, the pseudo-pressure equation will give more accurate results than the Russell Goodrich equation.

This option is not available in gas condensate runs. In these cases the generalized pseudo-pressure method may be used instead.

Generalized pseudo-pressure in ECLIPSE 100

ECLIPSE 100

The generalized pseudo-pressure method is intended for use by gas condensate producers. It provides a means of taking account of condensate dropout, as well as compressibility, in the calculation of the mobility integral. It is based on the method described by Whitson and Fevang [Ref. 74].

At the beginning of each time step the integral of the total oil and gas mobility is evaluated between the grid block pressure and the well bore pressure at the connection. This is compared with the total oil and gas mobility at grid block conditions multiplied by the drawdown, and the ratio of the two quantities is stored as a "blocking factor" for each grid block connection in the well,

$$\beta_j = \frac{\int_{(P_w + H_{wj})}^{P_j} (M_g + M_o) dP}{(M_{g,j} + M_{o,j})\,(P_j - (P_w + H_{wj}))} \qquad \text{[EQ 78.25]}$$

The integrand is fundamentally a function of two independent variables: the pressure P and the gas saturation S_g. (The oil saturation is equal to $1 - (S_g + S_w)$ and S_w is regarded as fixed at the grid block value.) However, S_g is eliminated as an independent variable, making it a function of P at pressures below the dew point by requiring that the local total mobility ratio should be the same as the total mobility ratio at grid block conditions,

$$M_o / M_g = M_{o,j} / M_{g,j} \qquad \text{[EQ 78.26]}$$

This requirement assumes that within the grid block the flows are in steady state and there is no zone of immobile dropped-out oil.

The integral in equation [EQ 78.25] is evaluated by applying the trapezium rule to a set of pressure values between the grid block and connection pressures. The user can control the distribution of these pressure values, and hence the accuracy of the integration, using keyword PICOND. At each pressure value below the dew point, the gas saturation is determined by solving equation [EQ 78.26] using Newton's method.

The blocking factor is then used to multiply both the oil and gas mobilities in the inflow performance relationship, equation [EQ 78.1]. Note that the free oil mobility is modified by this treatment, whereas the Russell Goodrich and dry gas pseudo-pressure treatments leave this unchanged. At present this calculation is not performed in injecting connections; the blocking factor is set equal to 1.0. The value of the blocking factor can be written to the Summary file by using the SUMMARY section keyword CDBF.

The blocking factor for each connection is retained for the duration of the time step, and recalculated at the beginning of each subsequent time step. In case this degree of explicitness should cause oscillations, these may be damped by averaging the calculated blocking factor with its value at the previous time step,

$$\beta_{\text{used}} = f\,\beta_{\text{new}} + (1 - f)\,\beta_{\text{previous}} \qquad \text{[EQ 78.27]}$$

where the weighting factor f is specified in the PICOND keyword.

Generalized pseudo-pressure in ECLIPSE 300

ECLIPSE 300

The generalized pseudo-pressure method of Whitson and Fevang [Ref. 74] is also available in ECLIPSE 300. The option can be requested for all wells with the PSEUPRES keyword, which should be entered early in the SCHEDULE section before any well operations or time stepping keywords are specified. Alternatively this option can be activated for individual wells by entering GPP in item 8 of keyword WELSPECS. Calculation controls are set with keyword PICOND. The calculation and its implementation are described below.

Consider the flow of phase j = (oil, gas), in a radial homogenous media of height h and permeability K. Darcy's law describes the velocity of the fluid at radius r in the equation:

$$v_j = \frac{Q_j}{2\pi rh} = \frac{KK_{rj}}{\mu_j}\frac{dP}{dr} \qquad \text{[EQ 78.28]}$$

where Q_j is the in-situ volumetric flow rate, K_{rj} and μ_j are the phase relative permeability and viscosity and dP/dr is the pressure gradient. The phase volumetric flowrate Q_j is related to phase molar flowrate n_j by:

$$Q_j = n_j V_{mj} \qquad \text{[EQ 78.29]}$$

where V_{mj} is phase molar volume. Combining [EQ 78.28] and [EQ 78.29] and re-arranging gives:

$$n_j \frac{dr}{r} = 2\pi Kh \frac{K_{rj} b_j}{\mu_j} dp \qquad \text{[EQ 78.30]}$$

where $b_j = 1/V_{mj}$, the phase molar density. For component, i, the component phase molar rate will be:

$$n_{ij} \frac{dr}{r} = 2\pi Khx_{ij} \frac{K_{rj} b_j}{\mu_j} dp \qquad \text{[EQ 78.31]}$$

where x_{ij} is the mole fraction of the ith component in the jth phase.

Integrating [EQ 78.31] between the Peaceman pressure equivalent radius r_B and the well radius r_W gives:

$$n_{ij} \int_{r_W}^{r_B} \frac{dr}{r} = 2\pi Kh \int_{p_W}^{p_B} x_{ij} \frac{K_{rj} b_j}{\mu_j} dp \qquad \text{[EQ 78.32]}$$

Summing component phase molar rates in the oil and gas phases for component i:

$$(n_{io} + n_{ig})\left[\ln\left(\frac{r_B}{r_W}\right) + S\right] = 2\pi Kh \int_{p_W}^{p_B} \left[x_i \frac{K_{ro} b_o}{\mu_o} + y_i \frac{K_{rg} b_g}{\mu_g}\right] dp \qquad \text{[EQ 78.33]}$$

or:

$$n_i = T \int_{p_W}^{p_B} M_i(p) dp \qquad \text{[EQ 78.34]}$$

where the well connection factor, T, including the skin factor, S, is given by:

$$T = \frac{2\pi Kh}{\ln\left(\frac{r_B}{r_W}\right) + S} \qquad \text{[EQ 78.35]}$$

and the Component Generalized Molar Mobility (CGMM) is given by:

$$M_i = x_i \frac{K_{ro} b_o}{\mu_o} + y_i \frac{K_{rg} b_g}{\mu_g} \qquad \text{[EQ 78.36]}$$

Normally, we do not consider the pressure dependency in [EQ 78.34]. Instead, we compute the CGMM at the block pressure:

$$\int_{p_W}^{p_B} M_i(p)dp \rightarrow M_i(p_B)[p_B - p_W] \qquad \text{[EQ 78.37]}$$

that we know can be a poor approximation for a gas condensate where, although the block pressure may be well above the dew point and hence the gas relative permeability is high, the well pressure may be well below the dew point in which case the gas relative permeability can be substantially reduced. In this case, we want perform the integral in [EQ 78.33].

To prevent accumulation of moles in the completion cell, the ratio of CGMM for component i to the Total Generalized Molar Mobility (TGMM) must be a constant:

$$z_{pi} = \frac{M_i}{M_T} \qquad \text{[EQ 78.38]}$$

where:

$$M_T = \sum_i M_i \qquad \text{[EQ 78.39]}$$

Therefore, [EQ 78.34] can be written:

$$n_i = Tz_{pi}\int_{p_W}^{p_B} M_T(p)dp \qquad \text{[EQ 78.40]}$$

where:

$$M_T = \frac{K_{ro}b_o}{\mu_o} + \frac{K_{rg}b_g}{\mu_g} \qquad \text{[EQ 78.41]}$$

If the block pressure p_B exceeds the dew point pressure, the *production composition* z_{pi} is simply the total hydrocarbon composition in the block. If the block pressure is less than the dew point pressure, the production composition is computed from CGMM for oil and gas.

When the Generalized Pseudo-Pressure (GPP) model is applied in ECLIPSE 300, rather than re-writing the appropriate parts of the well model, we simply modify the conventional well inflow equation through the addition of the component dimensionless *flow blocking* factor, F_{Bi}, to give:

$$n_i = TF_{Bi}M_i(p)[p_B - p_W] \qquad \text{[EQ 78.42]}$$

where:

$$F_{Bi} = \frac{1}{M_i(p_B)}\frac{1}{(p_B - p_W)}z_{pi}\int_{p_W}^{p_B} M_T(p)dp \qquad \text{[EQ 78.43]}$$

Because of [EQ 78.38], we can write:

$$M_i(p_B) = z_{pi}M_T(p_B) \qquad \text{[EQ 78.44]}$$

therefore, we define the total dimensionless flow blocking factor, F_B, (DFBF)

$$F_B = \frac{1}{M_T(p_B)}\frac{1}{(p_B - p_W)}\int_{p_W}^{p_B} M_T(p)dp \qquad \text{[EQ 78.45]}$$

and hence the modified inflow equation becomes:

$$n_i = TF_B M_i(p)[p_B - p_W] \qquad \text{[EQ 78.46]}$$

Note `OPTIONS3` item 121 can be used to impose an upper limit (greater than or equal to unity) on the value of the dimensionless blocking factor in order to prevent convergence issues in the simulation.

The final form of the GPP inflow equation, [EQ 78.46], differs only from the normal inflow equation by the additional multiplication factor, F_B, which we calculate from [EQ 78.45]. The block pressure, p_B and the TGMM at block conditions, $M_T(p_B)$ is already known prior to entering the well routines. The integral in [EQ 78.45] appears to be a function of two variables:

$$Q(p_B, p_W) = \int_{p_W}^{p_B} M_T(p)dp \qquad \text{[EQ 78.47]}$$

but in fact it can be constructed as the difference of two integrals, namely:

$$Q(p_B, p_W) = R(p_B) - R(p_W) = \int_{p_L}^{p_B} M_T(p)dp - \int_{p_L}^{p_W} M_T(p)dp \qquad \text{[EQ 78.48]}$$

where p_L is some suitable *lower* or base pressure, say the user-specified bottom hole pressure.

The contributions of the oil and gas phases to the TGMM are clearly saturation dependent through the respective relative permeability functions, (K_{ro}, K_{rg}). The requirement that the produced composition be constant is equivalent to saying that the produced Gas-Oil-Ratio (GOR) or the produced gas-oil molar ratio be constant. Therefore,

$$\frac{M_o(S_o)}{M_g(S_g)} = \frac{L}{V} \qquad \text{[EQ 78.49]}$$

where L and V are the total liquid and vapor moles at the pressure $p_L \le p \le p_B$ such that $L + V = 1$ and the gas saturation $S_g = 1 - S_w - S_o$ where (S_w, S_o) are the water and oil saturations, respectively. The phase molar mobility, M_j is simply:

$$M_j = \frac{K_{rj} b_j}{\mu_j} \qquad \text{[EQ 78.50]}$$

[EQ 78.49] is a simple non-linear equation in one variable, the oil saturation. The single integral in [EQ 78.48] is constructed by trapezium rule, following the suggestions made by Whitson and Fevang [Ref. 74]. In practical terms, the integral is constructed at the beginning of each time step, only if the block conditions have changed significantly.

Note As described on page 107 of [Ref. 127], and also illustrated in [Ref. 129], an over-coarse grid will lead to an underestimate of well deliverability. It is important that the grid size containing the well should be small enough to model the region 1 steady-state flow using GPP. This will be of the order of 50-100m for lean gas condensate and 100-200m for rich gas condensate reservoirs, although sensitivity studies should be carried out to determine the required well grid.

Velocity Dependent Relative Permeability effects

The generalized pseudo-pressure model may include the effects of velocity dependent relative permeabilities (see item 4 of keyword `PICOND`). The pseudo-pressure integral [EQ 78.45] and the phase molar mobility, M_j [EQ 78.50] require the calculation of oil and gas relative permeabilities. At each pressure in the pseudo-pressure integration, the radius at which this pressure occurs is estimated using the following equation:

$$\frac{\ln r - \ln r_w}{\ln r_o - \ln r_w} = \frac{m(p) - m(p_w)}{m(p_B) - m(p_w)} \qquad \text{[EQ 78.51]}$$

When the radius is known, it is possible to derive the Darcy velocity and Capillary Number at the given pressure. The equations for calculation of Capillary Number are presented in "Velocity-dependent relative permeabilities" on page 716. The ‘Pipe-Equivalent’ velocity is calculated from:

$$v_g = \frac{n_g}{2\pi r h \phi (1 - S_{wc}) b_g} \qquad \text{[EQ 78.52]}$$

The Capillary Number relative permeabilities may be further modified by a turbulent, Forchheimer effect. A description of the models available for the Forchheimer parameter is presented in "Forchheimer models" on page 714. This effect is modeled as a non-Darcy flow factor as described in "Non-Darcy Flow" on page 711.

Note If a cell containing a well connection drops below dew point, and velocity dependent effects are strong, rapid variations in PI are occasionally observed. These effects are caused by gas condensate effects being modeled by both the GPP algorithm and the grid block discretization simultaneously. If the connected cell is coarse enough that all of regions 1and 2 (as described in [Ref. 74] and [Ref. 127]) are contained within the pressure-equivalent radius then these anomalies will be eliminated.

Injecting connections

In injecting connections, an alternative expression for the mobility is required to reflect the upstream saturation conditions in the well bore. In the absence of well bore crossflow, there would be single-phase conditions at all injecting connections. In these circumstances it is standard practice to replace the term $k_{p,j} / \mu_{p,j}$ in [EQ 78.21] and [EQ 78.22] with its sum over all phases, to give in ECLIPSE 100

$$M_{p,j} = \frac{\frac{k_{o,j}}{\mu_{o,j}} + \frac{k_{w,j}}{\mu_{w,j}} + \frac{k_{g,j}}{\mu_{g,j}}}{B_{p,j}} \qquad \text{[EQ 78.53]}$$

and in ECLIPSE 300

$$M_{p,j} = \frac{k_{o,j}}{\mu_{o,j}} + \frac{k_{w,j}}{\mu_{w,j}} + \frac{k_{g,j}}{\mu_{g,j}} \qquad \text{[EQ 78.54]}$$

where p here refers to the injected phase. The connection injectivity is thus a function of the total fluid mobility in the grid block. In ECLIPSE 300, the fluid in the wellbore is flashed at the bottom hole pressure to determine the wellbore fluid molar density. For each connection, this wellbore molar density is used with the mobility in [EQ 78.54], the pressure drawdown and the connection factor to form connection molar injection rates. (These are summed to give the total molar injection rate of the well, which is multiplied by a surface volume per mole factor to give the reported injection volume rate.) The ECLIPSE 300 calculation thus relies on quantities at both wellbore conditions (fluid molar density) and grid block conditions ([EQ 78.54]) to determine the injection flow for a given drawdown. ECLIPSE 100, in contrast, only relies on quantities at the grid block conditions ([EQ 78.53]). For gas injectors with very large drawdowns, the difference between the wellbore and grid cell pressures will result in small differences in gas injection flow rates between the simulators for BHP controlled wells.

If gas or water is injected into a grid block initially containing oil, this relationship will cause the well's injectivity to vary until the grid block is completely flooded out with the injected phase. In reality however, most of the pressure drop occurs over a region close to the well, and when this region has flooded out the injectivity will stay approximately constant. If the grid block size is much larger than this region, the calculated injectivity will be incorrect until the whole grid block has flooded out. For cases like this, an alternative approach is available that allows the user to supply a constant value of the injected phase relative permeabilities. In ECLIPSE 100 the keyword `COMPINJK` can supply a single relative permeability of the injected phase for non-crossflowing injectors. In ECLIPSE 300 the keywords `COMPKRI` and `COMPKRIL` supply relative permeabilities for all three phases, which can be applied to injectors regardless of their crossflow status; alternatively `COMPMOBI` and `COMPMBIL` can supply a fixed value for the injection mobility.

ECLIPSE 100

If crossflow occurs through the well bore, or if the well is a multi-phase injector, it is possible for the well to inject a mixture of phases into the formation through one or more connections. The injection mobilities must be adapted to represent the fluid mixture present in the well bore. ECLIPSE 100 determines the proportion of each phase present in the mixture entering the well bore, and ensures that the phases flow in the same proportions through all the injecting connections. The treatment of crossflow is thus based on the assumption that the mixture of phases is uniform throughout the well bore at formation level. To determine the injection mobilities, a set of intermediate quantities is first calculated, which we call the "well volume factors". The well volume factor α_p of a given phase p is defined to be the stock tank volume

of the phase that is contained within a unit volume of the well bore at formation level. Its value depends on the well's bottom hole pressure and the mixture of phases entering the well bore. The injection mobilities are set in proportion to the well volume factors. Assuming that the total injectivity of the connection is still proportional to the total fluid mobility within the grid block, the injection mobilities are given by

$$M_{p,j} = \alpha_p \left[\frac{\frac{k_{o,j}}{\mu_{o,j}} + \frac{k_{w,j}}{\mu_{w,j}} + \frac{k_{g,j}}{\mu_{g,j}}}{\alpha_{fo} B_{o,j} + \alpha_w B_{w,j} + \alpha_{fg} B_{g,j}} \right] \qquad \text{[EQ 78.55]}$$

The expression reduces to the standard injection mobility, [EQ 78.53], when the well bore contains only one free phase.

If a flow-dependent skin factor is used, the mobility reduction factor D_m is calculated as in [EQ 78.13]. It is applied to every phase flowing in the injecting connection, so that the injection rates remain in proportion to the well volume factors.

If the Russell Goodrich or pseudo-pressure equations are applied, they affect the mobility of free gas within the grid block containing the connection, unlike the skin factor whose influence is restricted to the immediate vicinity of the well bore. Accordingly the term $k_{g,j}/\mu_{g,j}$ in [EQ 78.53] or [EQ 78.55] is replaced by the term $k_{g,j}\lambda_{g,j}B_{g,j}$, where $\lambda_{g,j}$ is evaluated in the form appropriate to each equation, as described above for producing connections.

High mobility injection wells

ECLIPSE 300

It is known that injector wells can cause convergence problems when the mobilities are high in the connecting grid cells. Specially in thermal simulation this might be the case when steam is injected at high pressure resulting in fracturing near the well. The high mobility injection well use the expression

$$\hat{q}_{p,j} = T_{wj} M_{pj} \left(\frac{\sum_i M_{pi}(P_i - H_{wi})}{\sum_i M_{pi}} - P_w \right)$$

for the volume injected by a completion j. The mobilities are the total mobilities as given above. See equation [EQ 78.1] and above for the standard model.

Using the high mobility injection well option, set by item 16 of the keyword `WELSPECS`, an average completion pressure-term is calculated. This gives a fixed drawdown for all the completions of the well. However the total volume injected at a given reservoir pressure and bottom hole pressure is maintained by the model. It is important to note that this model does not reflect the instantaneous flow that would occur on a short time scale as the model is designed to average out injections oscillating between completions and thus avoiding convergence problems for larger time steps.

The well bore head term

The well bore head term H_{wj} is the pressure difference in the well bore between the connection j and the well's bottom hole datum depth. Friction effects are usually small within the well bore at formation level, and they are neglected. H_{wj} therefore represents the hydrostatic head through the mixture in the well bore between the connection depth and the bottom hole datum depth.

If we treat the mixture as uniform throughout the well bore at formation level, the hydrostatic head depends on the average mixture density. This can be readily calculated from the well volume factors,

$$\bar{\rho} = \rho_{o,\mathrm{surf}}\alpha_o + \rho_{w,\mathrm{surf}}\alpha_w + \rho_{g,\mathrm{surf}}\alpha_g \qquad \text{[EQ 78.56]}$$

where the subscript 'surf' refers to the phase densities at stock tank conditions.

However, the assumption that the mixture density is uniform throughout the well bore at formation level may result in inaccuracy. Consider a well having two connections, in which oil enters through the lower connection and a high G.O.R. mixture enters through the upper connection. In reality the density of the fluid in the well bore between the two connections would be close to the density of the oil in the lower connection, whereas [EQ 78.56] would predict a lower density corresponding to a mixture of higher G.O.R. This could give inaccurate results if the head between the two connections were a significant fraction of the drawdown. To overcome this, ECLIPSE has an option to calculate the head between each pair of connections separately, according to the local mixture density. The well bore is segmented into sections beginning and ending at adjacent connections. Within each section, the local mixture density is calculated from the accumulated upstream connection flows, assuming there is no slip between the phases,

$$\rho_{\mathrm{local}} = \frac{\rho_{o,\mathrm{surf}}Q_o + \rho_{w,\mathrm{surf}}Q_w + \rho_{g,\mathrm{surf}}Q_g}{Q_{fo}B_o + Q_wB_w + Q_{fg}B_g} \qquad \text{[EQ 78.57]}$$

Q here represents the accumulated flow through the upstream connections, and the formation volume factors B are evaluated at the average pressure in the well bore section. The expression on the denominator is the local volumetric flow rate of the mixture in the section. Within the present structure of the program, it is not possible to perform the segmented head calculation implicitly. It is therefore performed explicitly, based on the conditions at the end of the previous time step. The explicit treatment of this term is not likely to cause instability, and in view of its greater accuracy it has been made the default option in **both simulators**.

The productivity index

Engineers have traditionally used the "productivity index" to define the degree of communication between a well and the reservoir. Unlike the connection factor, it is a quantity that can be obtained directly from field measurements. But whereas the connection factor remains constant over the lifetime of the well (unless the skin factor changes because of silting or stimulation, for example), the productivity index will vary with the fluid mobilities at the well.

The steady-state productivity index is defined as the production rate of a chosen phase divided by the drawdown. The drawdown here is the difference between the well's bottom hole pressure and the pressure in the reservoir at the edge of the well's zone of influence. The extent of the well's influence is known as the "drainage radius", r_d. Thus the productivity index, J, is defined as

$$J = \frac{Q_p}{P_d - P_w} \qquad \text{[EQ 78.58]}$$

where Q_p is the production rate of the chosen phase, and P_d is the pressure at the drainage radius. [EQ 78.58] also defines the corresponding quantity for injection wells, the "injectivity index", if injection rates are treated as negative.

A relationship between J and the connection factors of the well can be derived, assuming steady radial Darcy flow exists with uniform mobility throughout the region bounded by the drainage radius:

$$J = \sum_j \left[T_{w_j} M_{p,j} \left(\frac{\ln(r_o/r_w) + S}{\ln(r_d/r_w) + S} \right) \right] \qquad \text{[EQ 78.59]}$$

where Σ_j denotes a summation over all the connections j belonging to the well. The simulator calculates the productivity index of each well from this expression, and can optionally print it out at each well report. If the chosen phase p is gas and the connections have a non-zero D-factor, the non-Darcy skin is added to S in [EQ 78.59].

The simulator's calculation of the productivity index should not be used for a horizontal well, as the assumptions used in [EQ 78.59] will not be valid. [EQ 78.59] requires that a steady radial flow regime perpendicular to the well bore exists out to the drainage radius. For a horizontal well, this flow regime will rapidly be disrupted by the top and bottom boundaries of the formation, and the ultimate flow regime may be linear or pseudo-radial depending on the geometry of the well and its drainage region. For horizontal wells, therefore, you should calculate the productivity index "manually" from [EQ 78.58]. The reservoir pressure at the drainage radius could be obtained from a representative grid block, or the average pressure of the field or a suitable fluid-in-place region could be used.

The drainage radius r_d is often known only to a poor degree of accuracy. The engineer must supply its value for each well in the input data, but there is a default option which sets its value for each connection equal to the value of the pressure equivalent radius of the connecting grid blocks. In this case Equation [EQ 78.59] simply reduces to

$$J = \sum_j [T_{wj} M_{p,j}] \qquad \text{[EQ 78.60]}$$

and the PI should be regarded as the PI of the well within its connecting grid blocks rather than within its drainage region.

Additional forms of the well PI can be output to the Summary File using the SUMMARY Section mnemonics WPI1, WPI4, WPI5 and WPI9. These use [EQ 78.58] where P_d is replaced with a weighted average pressure within a 1-block, 4-block, 5-block and 9-block region around each connecting grid block. The weighting formula is specified with keywords WPAVE and WWPAVE

ECLIPSE 100

An alternative option is triggered by the engineer setting a negative value for the drainage radius. In this case the well's production/injection potential is printed in the well reports instead of its productivity index. The potential is defined as the flow rate that the well could sustain in the absence of any rate limits, that is if it were constrained only by the supplied limiting values of the bottom hole pressure and the tubing head pressure, if set. When this option is selected, the well PIs can still be output to the SUMMARY file using the SUMMARY section mnemonic WPI.

Note The PI of the preferred phase of a well is written to the SUMMARY file as a quantity without units. This is because the SUMMARY file is unable to handle a change in the units if the preferred phase of the well changes between gas and liquid.

Well Modeling Facilities

Chapter 79

Well completions

Multiple connections

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

Each well can be completed in several layers of the reservoir. A communication between the well bore and a single grid block is defined as a **connection**. A given well may have any number of connections, as long as it does not exceed the maximum number of connections per well, which is specified in keyword `WELLDIMS` in the `RUNSPEC` section. (Any of the `WELLDIMS` maximum values may be increased on a restart, and in ECLIPSE 300 it is also possible to increase them during a run using the `CHANDIMS` keyword.)

The connections belonging to a particular well need not all lie in the same vertical column of grid blocks, so it is possible to model highly deviated wells. Moreover a well can be completed in several grid blocks within the same layer, enabling the simulator to model horizontal wells. This feature is also useful in 3D radial coning studies where the well connects with all the innermost, wedge-shaped, blocks in any given layer.

Inflow performance

The flow rate of a phase (oil, water or gas) through a connection is proportional to the product of three quantities:

1. the transmissibility between the grid block and the well bore
2. the mobility of the phase in the grid block at the perforations
3. the pressure drawdown between the grid block and the well bore.

These quantities are discussed in more detail in "Well Inflow Performance" on page 1187.

In certain situations the drawdown in one layer may have the opposite sign to the drawdown in the other completed layers. The simulator will allows **crossflow** to take place between the reservoir layers through the well bore. A material balance constraint is observed within the well bore at formation level, so that the mixture of phases flowing out of the well bore corresponds to the average phase or component mixture entering the well bore. A special case in which crossflow may occur is when the well is plugged off above the formation while the completions remain open to the reservoir. In ECLIPSE 100 the crossflow facility can be turned off if you do not wish this to occur, using a switch in the keyword `WELSPECS`.

Relative permeabilities in well connections

The relative permeability of a phase in a well connection is a function of the saturations in the grid block containing the connection. By default, the relative permeabilities are calculated using the Saturation Function tables associated with the grid block containing the connection. However, the engineer can supply additional relative permeability tables for calculating the fluid mobilities at the well connections. These tables could take into account the effects of coning and/or partial completion within the grid block, or they could be well **pseudo** tables derived from an earlier fine-grid simulation. The association between the relative permeability tables and the well connections is defined using the keyword `COMPDAT`.

An alternative means of allowing for coning or partial completion within a grid block is to re-scale the Saturation Function tables to different end-point saturation values. The engineer can optionally supply alternative sets of end-point saturation values for use in particular well connections. These end-point values are used to transform the grid block saturations into the saturations 'seen by' the well connections. The alternative end-point saturation values are entered using the keyword `COMPRP`.

ECLIPSE 100

In simulation runs which use the Vertical Equilibrium option, the keyword `COMPVE` should be used instead of `COMPRP`, to supply depth values for the top and bottom of the perforated interval. The `COMPVE` keyword may be used to specify partial penetration data in non-VE runs also; ECLIPSE automatically translates the top and bottom depths of the perforations into `COMPRP` type scaled end-points.

Special facilities for gas wells

There are two facilities for improving the accuracy of modeling gas inflow. Firstly, the non-Darcy component of the pressure drawdown resulting from turbulence near the well bore is modeled by a flow-dependent contribution to the skin factor. The turbulence skin factor for a well connection is equal to the product of the free gas flow rate in the connection and the connection's D-factor. Well D-factors are defined using the keyword `WDFAC` or they may be calculated from a correlation using the keyword `WDFACCOR`.

The second facility is the option to use a special inflow equation to provide a more accurate model of the flow of gas into the well. ECLIPSE 100 offers a choice of three models: the Russell Goodrich equation, the dry gas pseudo-pressure equation and the generalized pseudo-pressure method. These are all methods of taking into account the pressure-dependence of the gas properties between the grid block pressure and the well bore pressure. The generalized pseudo-pressure equation also takes account of the effects of condensate dropout. They are described in more detail in "Special inflow equations" on page 1195. You can select the desired equation by setting a switch in the keyword `WELSPECS`. The generalized pseudo-pressure equation is also available in ECLIPSE 300, where it may be activated for specific wells using the `WELSPECS` keyword, or it may be activated for all production wells using the keyword `PSEUPRES`.

Scale deposition model

A scale deposition model is available, which accounts for the cumulative effects of scale deposited around the well connections and the resulting degradation of the productivity index.

The rate of scale deposition in each well connection is proportional to the flow rate of water into it and also varies with the amount of impurities dissolved in the water. Formation water and injected sea water contain different impurities and thus the overall rate of scale deposition is dependent on the relative proportion of sea water and formation water.

This scale deposition process is modeled by a table that defines the rate of scale deposition per unit water flow rate as a function of the fraction of sea water present in the water flowing into a connection (see keyword `SCDPTAB`). The sea water fraction is represented by a passive water tracer configured by the user (see keyword `SCDPTRAC`).

A second scale damage table defines how the current amount of deposited scale alters the PI of the well connections (see keyword `SCDATAB`).

The calculation for the effect of scale damage on a well is performed explicitly. Thus, at the end of each time step the mass of scale deposited around each connection is incremented according to the flow of water during the time step and the current sea water concentration. This is translated into a PI reduction factor for each connection, that is used during the following time step.

You may supply multiple versions of each of the two table types. The `WSCTAB` keyword must be supplied to select which tables are to be used by each well for the calculation.

The `WSCCLEAN` keyword may be used to reduce the amount of scale deposited around well connections after a scale cleanup operation. (The `WSCCLENL` keyword should be used for wells in local grid refinements.)

Well controls and limits

Well control options

Individual wells can operate at a target value of any of the following quantities:

1 Oil flow rate

2 Water flow rate

3 Gas flow rate

4 Liquid flow rate (oil and water)

5 Linearly combined rate (user-specified linear combination of oil, water and gas)

6 Reservoir fluid volume (or voidage) rate

Note Voidage rate is the reservoir volume that the produced or injected fluids would occupy at reservoir conditions. The associated reservoir pressure is usually the average hydrocarbon pressure in the field. However, this may be set to be the average pressure in a nominated fluid in place region using argument 13 in keyword `WELSPECS`. By obtaining voidage rates at a single pressure for all wells we avoid the effects of higher pressures at injectors and lower pressures at producers; setting equal injection and production voidage rates will then yield approximate pressure maintenance.

7 Bottom hole pressure

8 Tubing head pressure

ECLIPSE 300 only

9 Wet gas rate (which is the volume that the hydrocarbon mixture would occupy at standard conditions if it were an ideal gas)

ECLIPSE 300 only

10 Total molar rate

Note The total molar rate includes the molar rate of water.

ECLIPSE 300 only

11 Calorific value rate.

One of these quantities is selected as the main control target, while limiting values can be supplied for any of the remaining quantities. The limiting flow rates are treated as upper limits, while the pressure limits are treated as lower limits in production wells and upper limits in injectors. The well will automatically change its mode of control whenever the existing control mode would violate one of these limits. The target quantity in the old control mode will then become a limit in the new control mode.

For example, consider a production well operating at a target oil flow rate of 1000 stb/day, with a lower limit of 3000 PSIA on the bottom hole pressure. The well will produce 1000 stb/day of oil until the bottom hole pressure falls below 3000 PSIA. The well's control mode will then change automatically to maintain a constant bottom hole pressure of 3000 PSIA, and the oil production rate will decline. If subsequently the oil production rate were ever to exceed 1000 stb/day (resulting for example from stimulation of the well, or the opening of an extra connection), then the control mode will automatically switch back to the original target oil production rate.

In addition to the control quantities listed above, a maximum allowable drawdown can be specified for each production well, using the keyword WELDRAW. This is converted into a maximum liquid or gas production rate at each time step (depending on the well's preferred phase), using the current value of the liquid or gas mobility in the well's connections. Thus the liquid or gas rate limit is used to apply constraints on the drawdown.

An alternative method of control can be applied to wells during the history matching process, when their observed oil, water and gas production rates are known. The observed rates are entered with the WCONHIST keyword. A well can be made to produce at a rate which matches one of the phases, or at the equivalent liquid or reservoir volume rate of the observed production. The latter option is useful for making the well produce the correct amount of total fluid from the reservoir before the mobility ratios are fully matched. Thus the rate of pressure decline should be approximately correct. The observed rates and production ratios can be written to the summary file, for graphical comparison with the calculated rates. Observed rates for injection wells may be entered with the WCONINJH keyword

Economic limits

There is an additional class of limits that can be applied to production wells. The engineer can specify values of the oil or gas production rate, the water cut, the gas-oil ratio and the water-gas ratio which represent the limits of economic operation of the well. The violation of one of these limits will result in the well, or one or more of its connections, being automatically closed. The economic limit options are:

1 Lower limits for the oil, gas and liquid production rates for each well. The well is closed if any of these limits are broken.

ECLIPSE 100

Alternatively, if the well still has one or more connections placed on automatic opening (see "Automatic opening of wells and well connections" on page 1214), one of these connections will be opened.

2 Upper limits for the water cut, the gas-oil ratio, the gas-liquid ratio and the water-gas ratio of each well. There is a choice of actions to be taken if a well violates one of these limits:

- The worst-offending connection (that is the connection with the highest water cut, gas-oil ratio or water-gas ratio) in the well is closed.
- The worst-offending connection and all connections below it in the well are closed. Here, "below" is interpreted as "further from the wellhead according to the connection ordering". The connection ordering may be specified in the COMPORD keyword.
- The well itself is closed.

ECLIPSE 100

- The well is plugged back, i.e. the length of perforations in either the top or bottom open connection or the worst-offending connection is reduced by an amount specified in keyword WPLUG.

Keyword WECON is used to set the limiting values and define the choice of workover action.

Note If a well which is closed still has two or more connections open to the reservoir, it may be possible for cross-flow to occur between the completed layers through the well bore. To distinguish a well that can still make its presence felt in this way after being closed, we refer to it as **stopped**. A well that is not allowed to crossflow after being closed is referred to as **shut**. Thus a **stopped** well can still affect the reservoir, whereas a **shut** well is effectively ignored by the simulator. Wells can be opened, stopped or shut manually. The engineer can also stipulate whether a well is to be shut or stopped after it violates an economic limit, using item 9 in the WELSPECS keyword.

Economic limits can also be set for individual connections in production wells. A connection will be closed if it exceeds its limit on water cut, gas-oil ratio or water-gas ratio, or if its oil or gas production rate falls below a specified value. Connection economic limits are set using the keyword CECON.

ECLIPSE 100

A minimum economic rate can also be applied to injection wells, using keyword WECONINJ. This allows injection wells to be automatically closed if they are not injecting at a high enough rate.

Workovers are normally performed on individual well connections, but it is possible to lump sets of connections together into 'completions' for simultaneous closure, using the keyword COMPLUMP. When a well with lumped connections is worked over (for example, after exceeding a water cut limit set in WECON), the worst-offending completion is identified (by summing the flow rates of its connections) and all the connections belonging to that completion will be shut together.

The economic limits are checked at the end of each time step, and the appropriate action is taken if any are violated. The action is taken at the end of the time step during which the limit was violated. Thus a well may produce 'uneconomically' for one time step before remedial action is taken. However, the engineer can specify a **tolerance fraction** for these limits. If a limit is broken by proportionately more than this tolerance, the time step will be repeated after the remedial action has been taken. Thus the action will take effect from the beginning of the time step in which the limit would otherwise have been violated. For example, consider a well which has a maximum economic gas-oil ratio of 10 Mscf/stb, and assume a tolerance fraction of 0.2 has been specified. If the well's gas-oil ratio increases to 11 Mscf/stb, then remedial action is taken at the end of that time step, and will be effective for the following time step. If however the gas-oil ratio reaches above 12 Mscf/stb, the time step will be repeated after the remedial action has been taken. The tolerance fraction is set with the keyword WLIMTOL. Note that each time step will be repeated only once. Thus if a well still violates an economic limit at the end of the repeated time step, that time step is 'accepted'. Further remedial action will applied to the well before starting the next time step.

Wells or connections that are closed after violating an economic limit can be tested periodically to see if they can flow economically again. If they can flow without violating a limit, they will be re-opened. This facility can, for example, allow a watered out well to produce again after its cone has subsided sufficiently while it was closed. Instructions for periodic testing of closed wells are entered using the keyword WTEST.

There is an option to make the simulation end automatically if all the production wells in the field, or in a particular group, have been shut or stopped. This saves the simulator from using computer time needlessly when nothing further of interest is going to happen. The option can be switched on using the GECON keyword. In addition, selected wells can be flagged to make the simulation stop if any of them are closed during the run for any reason, using a switch in the WECON keyword. If for example a flagged well is shut because it has exceeded its economic

water cut limit, the run will stop and an appropriate message will be printed. The engineer can then examine the state of the reservoir at that time, make any necessary changes to the well completions, and re-start the run. This 'semi-interactive' mode of working allows the engineer to carry out workover procedures that are more complex than the built-in automatic workover options.

An additional option allows the engineer to nominate a well to be opened when another well is closed automatically for any reason (for example on violating an economic limit). This facility can be used, for example, to open an injector when a producer closes after becoming uneconomic. If the injector has the same completions as the producer, the effect will be equivalent to converting the producer into an injector, although in this case they are treated as two separate wells. The option is activated with item 9 in keyword WECON.

Automatic cut-back of flow rate

There is an indirect way of controlling the following production ratios of individual wells subject to gas or water coning:

1 Water cut

2 Gas-oil ratio

3 Gas-liquid ratio

4 Water-gas ratio.

Upper limits for one or more of these quantities can be set using the keyword WCUTBACK. Whenever one of these limits is exceeded, the well's production rate is cut back to a specified fraction of its current value. The rate will be cut back successively at each time step until the limit is no longer exceeded. The rate cut-back is performed at the end of the time step in which the limit is exceeded. However, if the limit is exceeded by proportionately more than the tolerance fraction entered with keyword WLIMTOL (see above), the time step will be repeated with the well flowing at the reduced rate. The time step will only be repeated once; if the limit is still exceeded at the end of the repeated time step, the well's flow rate will be reduced again for the next time step.

This facility is useful for determining the production rate necessary to keep water or gas coning in a well down to an acceptable level.

The WCUTBACK keyword can also supply a minimum pressure value for the grid blocks in which the well is completed. If the pressure falls below this value, the well's production rate is cut back to the specified fraction of its current value. This facility can be useful for limiting the decline of grid block pressure at the wells.

For each of the five cut-back limits, there is a corresponding limit that reverses the cut-back process. For example, the gas-oil ratio could have a cut-back limit of 5.0 Mscf/stb and a cut-back reversal limit of 3.0 Mscf/stb. If the GOR increases above 5.0 Mscf/stb, the production rate will be cut back until the GOR falls below this limit. If subsequently the GOR falls below 3.0 Mscf/stb while the well is operating at the reduced flow rate, the rate will be allowed to increase again, reversing the cut-back process. Cut-back limits and their corresponding reversal limits are therefore a useful way of making wells operate within a given band of conditions.

The WCUTBACK keyword can also apply to injection wells, in which case the production ratio limits are ignored, the cut-back pressure limit is a maximum value, and the cut-back reversal pressure limit is a minimum value.

Efficiency factors

Wells can be given 'efficiency factors' to take account of regular downtime (see keyword WEFAC). For example, if a well is 'down' for 10 percent of the time, its efficiency factor will be 0.9.

The well rates and pressures given in the well reports describe the state of the flowing well. The full rates are also used in the well economic limit checks. The efficiency factors are applied when updating the cumulative well and connection flows.

The efficiency factors are also applied when summing the well rates to obtain the group and field flow rates. Thus the group and field flow rates represent the average rates over the time step, rather than the full rates reached when all the wells are operating.

The WEFAC keyword is best used when wells are brought down individually on a regular basis for short periods of maintenance. But if all the wells in a group are brought down together for regular maintenance, then the efficiency factor should instead be applied to the group. Group efficiency factors are defined with the keyword GEFAC. The treatment of group downtime is entirely analogous to the treatment of well downtime. The group's flow rates reported in the PRINT and SUMMARY files reflect its full flow rate. Any flow targets and limits for the group will apply to its full flow rate. But the group rates will be multiplied by the group efficiency factor when summing the group's contribution to its parent group in the hierarchy. Thus the flows of any superior groups will reflect this group's time-averaged flow rate.

Automatic opening of wells and well connections

ECLIPSE 100

An automatic opening facility in ECLIPSE 100 allows wells to be opened in sequence at the earliest opportunity, subject to constraints on the drilling rate and the maximum number of open wells allowed in each group. Wells are placed on automatic opening by setting their OPEN/SHUT status to AUTO, in any of the WCON... keywords or WELOPEN. They will be treated as initially SHUT, and will be opened in sequence in the order in which they are first defined in keyword WELSPECS (although this order is over-ridden if some of the wells belong to groups that already have their maximum quota of open wells).

Drilling rigs can be allocated to groups with keyword GRUPRIG, and the time taken to drill each well may be specified in keyword WDRILTIM. Together these restrict the rate at which wells can be opened automatically. Wells can effectively be opened mid-way through a time step by having their efficiency factors temporarily adjusted to allow for the part of the time step in which they were still shut. Thus in order to open a series of wells at 30 day intervals, say, it is not necessary to restrict the time steps to 30 days.

Each group may be given an upper limit to the number of open wells it can contain, using keyword GECON (Item 9). When a group has its full complement of open wells, no more of its wells can be opened automatically until one is shut-in. The AUTO facility can therefore be used to keep a group operating with its full complement of wells while some wells are being closed after violating their economic limits.

Well connections can also be placed on automatic opening, by setting their OPEN/SHUT status to AUTO in keywords COMPDAT or WELOPEN. They will remain SHUT until the well is subjected to an automatic workover, as for example is performed when a well or group economic water cut limit is exceeded. A connection on AUTO will be opened each time its well is worked over. One will also be opened each time the well's production rate falls below its minimum economic limit. Within a given well, the connections will be opened in the order in which they are first defined in keyword COMPDAT.

In order to prevent wells or connections from being automatically opened in areas of the reservoir that have already been depleted, a minimum acceptable oil saturation can be declared using the keyword `WELSOMIN`. A connection will not be opened automatically in a workover, or opened when its well is opened automatically, if the connecting grid block has an oil saturation below this limit. A flag can also be set, using the keyword `WDRILRES`, that prevents a well on automatic opening from being opened if it is completed in the same grid block as a well that is already open.

Wells and individual connections can also be opened automatically from a **drilling queue** in response to a group failing to meet its production target. This facility is described in "Group Production Rules" on page 505.

Group and field control facilities

The multi-level grouping hierarchy

Each well must belong to a particular group, which is named when the well is first declared with keyword `WELSPECS`. Rate targets and limits, and economic limits, can be applied to each group and to the whole field. This provides a standard three-level (well - group - field) control hierarchy, where controls and limits can be specified independently at each level.

If controls or limits are required on more than three levels, a highly flexible multi-level hierarchy can be built up using the keyword `GRUPTREE`. This keyword declares additional groups that do not contain wells, but instead act as 'parents' to other groups at a lower level. These groups are called 'node-groups', to distinguish them from 'well-groups' which contain wells.

Figure 79.1 A five-level hierarchy

An example of a five-level hierarchy is shown in Figure 79.1. The field occupies the highest level, level 0. PLAT-A and PLAT-B are node-groups at level 1. The groups at level 2 are all well-groups except for SAT-B which is a level-2 node-group. Groups GR-S1 and GR-S2 are level-3 well-groups subordinate to SAT-B. GR-A1 and GR-A2, and their respective wells, are subordinate to PLAT-A. GR-B1 and GR-B2, and their respective wells, are subordinate to PLAT-B. GR-S1 and GR-S2, and their respective wells, are subordinate to both SAT-B and PLAT-B. When the wells are included (on levels 3 and 4), the hierarchy has five levels in total.

Group production control options

Group production rate targets - guide rate control

Selected groups at any level in the hierarchy (including the field) can be made to produce at a target value of any one of the following quantities:

- Oil production rate
- Water production rate
- Gas production rate
- Liquid (oil + water) production rate
- Linearly combined production rate (user-specified linear combination of oil, water and gas)

- Reservoir fluid volume rate

ECLIPSE 300 only
- Wet gas rate (which is the volume that the hydrocarbon mixture would occupy at standard conditions if it were an ideal gas)
- Reservoir volume production balancing fraction (which limits the group's reservoir fluid volume production rate to the specified fraction of its reservoir fluid volume injection rate).

ECLIPSE 300 only
- Surface gas volume production balancing fraction (which limits the group's surface gas volume available rate to the specified fraction of its surface gas volume injection rate).

ECLIPSE 300 only
- Surface water volume production balancing fraction (which limits the group's surface water volume production rate to the specified fraction of its surface water volume injection rate).

ECLIPSE 300 only
- Calorific value rate.

The control target is set using keyword `GCONPROD`. The group's target rate is apportioned between the individual wells in proportion to each well's specified **guide rate**. A guide rate can optionally be defined for any phase; oil, water, gas, liquid or reservoir fluid, using keyword `WGRUPCON`. If the phase differs from the phase under group control, the guide rate is translated into a guide rate for the controlled phase using the well's production ratio for the previous time step. Thus, for example, the target oil production rate of a group can be apportioned among its subordinate wells such that each well produces similar amounts of liquid. This can be done by giving each well the same guide rate for the liquid phase.

If a well's guide rate has not been specified, it is set at the beginning of each time step to a value that depends on the well's **production potential**. The production potential is defined as the instantaneous production rate that the well would initially have in the absence of any constraints on flow rate, at the current grid block conditions (see "Well potentials" on page 802). By default, the well's guide rate is set equal to its production potential of the phase under group control. This enables, for example, a group's target oil production rate to be apportioned among the individual wells such that the wells produce in proportion to their potentials. Alternatively, the keyword `GUIDERAT` can be used to specify the coefficients of a general formula for well guide rates as a function of their oil, water and gas production potentials. This can enable, for example, wells with high water cut to be given a progressively smaller guide rate as they water out.

The individual wells can be subject to their own flow and pressure constraints. For example, if a well would violate its bottom hole pressure limit when producing its full share of the group's production target, the well will operate at its bottom hole pressure limit. The production rates of the other wells will increase in proportion to their guide rates (or potentials) to meet the group's target, as long as they do not violate any of their own flow or pressure limits. A well that is producing its full share of the group's production target is said to be under **group control**, while a well constrained by its own flow or pressure limits is said to be under **individual control**. If required, certain wells can be prevented from coming under group control. These wells will continue to operate according to their individual flow or pressure constraints, regardless of any group flow target.

Drilling queue

At some stage in the simulation, it may happen that all the group-controlled wells in the group have hit one of their constraints, and have either dropped out of group control or have been closed. The group can then no longer meet its flow target, and its production rate will decline. This situation can be postponed by placing additional wells in the **drilling queue**, using either keyword `QDRILL` or keyword `WDRILPRI`. When a group can no longer meet its production target, the simulator will scan the drilling queue and open a suitable well. The well must be subordinate to the group that is unable to meet its target, but not subordinate to any lower level group that is operating at its own production target or limit for the same phase. The drilling

queue may either be a **sequential** queue, from which wells are opened in the sequence in which they were placed in the queue, or a **prioritized** queue from which wells are opened in decreasing order of their **drilling priority**. Well drilling priorities may be set manually in keyword WDRILPRI or calculated from their current potentials according to a function defined with keyword DRILPRI.

The rate of opening new wells from the drilling queue can be limited by specifying the time taken to drill each well with keyword WDRILTIM and allocating drilling rigs to groups with keyword GRUPRIG.

ECLIPSE 100

Additionally, ECLIPSE 100 can also place an upper limit on the number of open wells in each group, with keyword GECON. A well from the drilling queue is not opened if it would cause the limit to be exceeded. The restrictions set with the keywords WELSOMIN and WDRILRES described in "Automatic opening of wells and well connections" on page 1214 also apply to wells opened from the drilling queue. In addition to drilling new wells, ECLIPSE 100 can take other actions to postpone a decline in production rate. These include opening new connections in existing wells, decreasing the wells' THP limits, retubing the wells and applying artificial lift. The actions are specified with the keyword PRORDER. These are described more comprehensively in "Group Production Rules" on page 505.

Note The drilling queue should not be confused with the Automatic Opening facility described on page -1214. They are both subject to the same constraints on drilling rate, rig availability and maximum number of wells per group. But wells on AUTO are opened as soon as these constraints allow, whereas wells in the drilling queue are opened only when a group or the field cannot meet its rate target.

Group production rate limits

The engineer can impose **upper limits** on any of the quantities listed on "Group production rate targets - guide rate control" on page 1216, for selected groups and/or the field, using keyword GCONPROD. These limits can be imposed whether or not the group or field has a production target to maintain. If a group's production rate exceeds one of these limits, there is a choice of actions to follow:

1. The worst-offending connections in the worst-offending wells will be closed successively until the rate limit is honored. The 'worst-offending' well or connection here is the one that has the highest ratio of the production rate of the phase violating the limit to the production rate of the well's preferred phase.
2. As 1 above, except that all connections below the worst-offending one in the well will also be closed.
3. The worst-offending wells will be closed successively until the rate limit is honored.

ECLIPSE 100

4. The worst-offending well will be plugged back, that is, a length of perforations in the top or bottom open connection is reduced by an amount specified in keyword WPLUG.
5. The group will be made to produce the offending phase at its maximum allowed rate, according to the group control facilities described in "Group production rate targets - guide rate control" on page 1216. If the group was previously meeting a target rate for another phase, that target will no longer be met but will act as an upper limit for the production rate of that phase. The violated limit in effect becomes the new target.

Note When a group which was operating under a rate target for one phase, switches to rate control by another phase in this way, the guide rates for any children of this group are not allowed to be zero.
This is to prevent the following situation. Consider, for example, a group under gas production rate control which also has a limit on the group water production rate to control water break-through. When water breakthrough occurs in some of the group's wells and the water rate limit is violated, the group switches to water rate production control. Any wells that do not see any water breakthrough would not have any water production potential, and normally this would mean zero guide rates so these pure gas production wells would shut - which is obviously not what is required for a group which is primarily intended for gas production.
Instead, a small positive guide rate is imposed on such wells force them to stay open and operate at their potential. **Unfortunately, sometimes this means that the original group gas rate target is exceeded.** Because of this, it is strongly recommended that `RATE` is not used to control break-through in this way. Two possible alternatives are (i) use an alternative procedure such as `WELL` or `CON` under `GCONPROD` in items 11 onwards, or (ii) use a `GUIDERAT` formula that penalizes wells that see break-through.

If a reservoir fluid volume rate limit or a production balancing fraction limit is exceeded, only action 5 is performed.

As an example consider a group of oil wells experiencing water breakthrough, such that the liquid rate is approaching its maximum limit. If action 1, 2, 3 or 4 is selected, the wells or connections with the highest liquid-oil ratio (that is the highest water cut) will be successively closed or plugged back. If the group is operating at a target oil rate, this target will continue to be met until there are no wells remaining under group control. If however action 5 is selected, the group control facility will be applied to make the group produce liquid at the maximum liquid rate. The oil rate will decline steadily as the group's water cut increases.

Guide rate control for node-groups

When a node-group (or the field itself) has a rate target, its target rate will be apportioned among its subordinate wells as described in "Group production rate targets - guide rate control" on page 1216, unless a subordinate group either

- has a target rate of its own,
- has been declared unavailable for control at a higher level,
- has been given a production guide rate.

If a subordinate group has a target rate of its own, its production rates will have already been calculated. (The groups under rate control are processed level by level, from the bottom level up to the field.) For example, in Figure 79.1 let PLAT-B have a target rate of 50,000 stb/day for oil and SAT-B have a target rate of 30,000 stb/day for liquid. SAT-B's target rate may be a limit that has become a target after it had been violated. SAT-B's oil rate is subtracted from PLAT-A's target, and the remainder is shared between the wells in GR-B1 and GR-B2.

If SAT-B is required to produce 30,000 stb/day of liquid regardless of PLAT-B's target, it should be declared (in keyword `GCONPROD`) unavailable for control at a higher level. Otherwise its wells could come back under control from PLAT-B if ever SAT-B's share of PLAT-B's oil target fell below its current oil rate when producing 30,000 stb/day of liquid. SAT-B's wells would then produce their own share of PLAT-B's oil target, with a combined liquid rate of less than 30,000 stb/day. Declaring a group unavailable to respond to a higher level control will ensure that it continues to operate at its rate target or maximum production capacity independently of any higher level rate control.

A group can be given a production guide rate (in keyword GCONPROD) if it is required to produce a specific proportion of a higher level rate target. For example, if the field has an oil rate target of 100,000 stb/day and platforms PLAT-A and PLAT-B are required to produce equal quantities of liquid, then PLAT-A and PLAT-B should be given equal guide rates for the liquid phase. The liquid guide rates are translated into oil guide rates by multiplying them by each platform's oil-liquid ratio from the previous time step. Both platforms are then given oil rate targets in proportion to their oil guide rates, and placed under control from the higher (field) level. If PLAT-A is unable to provide its share of the field's oil target, it will produce what it can according to its own limits or capacity, and the remainder of the field's target will be allocated to PLAT-B.

Prioritization - an alternative to guide rate control

The prioritization option is an alternative means of applying production rate limits to groups. It can be used instead of the guide rate method described so far in "Group production control options" on page 1216. In the prioritization option, wells are turned on in decreasing order of priority, the well with the highest priority flowing first, until a group's production rate limit is exceeded. The flowing wells operate at their individual targets or limits (WCONPROD). However, the well that exceeds the group's rate limit is cut back to meet the limit, and the wells with priority lower than this subordinate to that group do not flow. The low priority wells not selected for flow are "priority-closed" until they are eventually selected.

The priority number for each well is calculated from its potentials at regular intervals, according to a general formula with user-defined coefficients. The keyword PRIORITY is used to select the prioritization option and to set the coefficients of the priority equation. The equation allows the priority to be set equal to, for example, the potential oil rate, or the reciprocal of the potential gas rate, or the reciprocal of the water cut. Individual wells may also have their priorities set independently by the user with the keyword WELPRI, overriding the general equation.

With the prioritization option, upper limits can be set for the oil, water, gas, liquid and reservoir fluid volume rates of each group. The choice of actions on exceeding a limit is the same as described in "Group production rate limits" on page 1218, except that action 5 causes the rate to be reduced by closing the low priority wells instead of cutting all the wells back to flow in proportion to their guide rates. The group production rate limits and actions must be set using the keyword GCONPRI, and not GCONPROD as in the guide rate method.

ECLIPSE 100

Optionally, a second priority equation can be defined, which may be used instead of the first equation when prioritizing wells to cut back production to obey specific rate limits. For example, if a group gas rate limit is exceeded you may wish to close wells with a high GOR, while if a water rate limit is exceeded you may wish to close wells with a high water-cut. When two priority equations are defined, the calculation procedure differs slightly from the single priority equation case. All the available wells are opened initially, then if a group rate limit is violated the wells with the lowest priority according to the formula chosen for that particular limit are closed in turn until the rate limit is no longer exceeded. The last well to be closed is then re-opened and made to produce at a rate calculated to meet the limit. If two or more group rate limits are exceeded which use different priority formulae, the lowest priority well is selected according to the formula appropriate to the limit that is exceeded by the greatest fraction. The 'active' formula may of course change as each well is closed.

It is possible to mix the two group control strategies - prioritization and guide rates - in a single run. Groups to be controlled by prioritization should have their rate limits set with keyword GCONPRI, while groups using the guide rate method have their rate limits set with keyword GCONPROD. A group can be changed from one method to the other simply by re-specifying its rate limits with the appropriate keyword. The mixing of the two methods, however, is subject to the following restriction: the guide rate method cannot be used by a group subordinate to a group using the prioritization method. Thus, for example, no groups can use the guide rate method if the field has a prioritization rate limit.

When the two group control strategies are mixed, the well rates are allocated according to the following method. First, the production wells subordinate to prioritization groups are solved, selecting the wells for flowing in decreasing order of their priorities. Next, the producers in the remaining parts of the grouping tree are solved, employing the standard guide rate method to apportion the flow rates between them to meet any GCONPROD group targets. If a prioritization group is subordinate to one with a GCONPROD rate target, the wells in the prioritization group will already have had their flow rates set, and the remaining wells will have their rates set to make up the rest of the GCONPROD group's target. Wells from the drilling queue (keyword QDRILL or WDRILPRI) that are not subordinate to a prioritization group will be opened when necessary to meet the GCONPROD group's rate target. The drilling queue is **not** used for opening wells subordinate to prioritization groups, as the prioritization algorithm itself handles the opening and closing of these wells.

Economic limits and cut-back limits

The set of economic limits described earlier for wells can also be applied to the overall production of groups and the field as a whole. If the oil or gas production rate of the group (or field) falls below a minimum value, all its production wells are closed. If the group (or field) water cut, gas-oil ratio or water-gas ratio exceeds a maximum value, then the choice of workover options described earlier for wells will be performed on successive worst- offending wells in the group (or field). The economic limit values are set using keyword GECON.

As with the well economic limits, the remedial actions for the group and field economic limits are performed at the end of the time step in which they were broken. But if a limit is broken by proportionately more than the **tolerance fraction** specified with keyword WLIMTOL, the time step will be repeated after the action has been taken.

ECLIPSE 100

There is also an option to cut back the group production rate by a specified fraction each time the group's water cut, gas-oil ratio, gas-liquid ratio, or water-gas ratio exceeds a specified value. The option is invoked with the keyword GCUTBACK. It works in a similar way to the well rate cut-back option controlled by the keyword WCUTBACK.

Group injection control options

Groups, and the field as a whole, can be given targets and limits which govern the injection rate of each phase. The injection controls can be applied independently of any production controls the group may have. The injection controls are set using the keyword GCONINJE.

A group at any level (including the field) can have its injection rate of water and/or gas (and oil in ECLIPSE 100) controlled to meet one of the following targets:

- Surface injection rate of a particular phase
- Total reservoir volume injection rate of all phases
- Re-injection fraction of a particular phase

- Total voidage replacement fraction of all injected phases.

ECLIPSE 300 only

- Wet gas rate (which is the volume that the injected hydrocarbon mixture would occupy at standard conditions if it were an ideal gas).

The first control mode gives the group a user-specified surface injection rate target. The second control mode gives the group a target reservoir volume injection rate for the phase that is equal to the user-specified total rate minus the reservoir volume injection rate of any other phases in the group. The third control mode gives the group a surface injection rate target equal to its production rate of the phase (or alternatively the production rate of another named group) multiplied by the user-specified re-injection fraction. If the phase is gas, gas consumed by the group and any subordinate groups is subtracted from the production rate, and gas imported to the group and any subordinate groups is added to the production rate, before multiplying by the re-injection fraction. (A gas consumption rate and import rate can be specified for any group using keyword GCONSUMP in ECLIPSE 100 and keywords GRUPFUEL and GADVANCE in ECLIPSE 300.) In ECLIPSE 300 gas reserved for sale (keyword GRUPSALE) is also subtracted from the production rate. The fourth control mode gives the group a target reservoir volume injection rate for the phase that is calculated such that the group's total reservoir volume injection rate equals its voidage production rate (or alternatively the voidage production rate of another named group) multiplied by the user-specified voidage replacement fraction.

If the group has a target set for reservoir volume injection rate or voidage replacement fraction for a particular phase, that phase is designated the 'top-up' phase for the run. It is used to top up the total group injection rate to the required target. Its injection target depends on the injection rates of the other phase(s) within the group, and therefore the 'top-up' phase must have its injection controls handled last of all. There can only be one 'top-up' phase at any given time in the simulation - different groups cannot have different 'top-up' phases. An example application of injection control is for a platform to re-inject all its available gas (that is, produced gas minus gas consumed for power), and to inject water so that the water and gas injection together replaces the platform's production voidage. Water is the 'top-up' phase here.

Upper limits can be imposed on the four quantities at any group or the field as a whole. If the injection rate exceeds one of a group's upper limits, the violated limit becomes a target for the group. If the group was already injecting at a target rate, the original target will no longer be met but will act as an upper limit for that quantity. For example, a group can inject water to replace its voidage production, subject to a surface rate limit that represents its injection water handling capacity. Obviously only the 'top-up' phase can have upper limits specified for quantities 2 or 4 since a limit could become a target at some stage in the simulation.

Defining the composition of injected gas

ECLIPSE 300

When gas is injected in compositional simulations, the nature of this gas must be defined. The injected gas may be provided in the following ways:

- The gas may be designated as a named wellstream, whose composition is defined explicitly in keyword WELLSTRE.
- The gas may be provided by the production stream of a named group. Gas used for fuel (GRUPFUEL) and sales (GRUPSALE) is by default subtracted from a source group's available gas supply for re-injection, but the keyword WTAKEGAS may be used to alter the priorities for re-injection, fuel and sales. The injection gas supply from the source group may be augmented by advance gas defined in keyword GADVANCE. The advance gas will be used for re-injection prior to the gas produced by this group from the reservoir.
- The gas may be provided by the production stream of a named well. The gas supply from the source well may be augmented by advance gas defined in keyword WADVANCE. The advance gas will be used for re-injection prior to the gas produced by this group from the reservoir.

- The injected gas may be a mixture from a number of sources. The mixture may be specified by the fractional contribution from each source (keyword WINJMIX). Alternatively the gas from a set of sources may be taken in a specified order (keyword WINJORD).

Groups with gas injection targets should have the nature of their injected gas defined in one of the above ways with the keyword GINJGAS. If there is not enough available gas from the source to fulfil the group's injection requirement, an additional type of gas import named "make-up gas" may be defined for the injecting group. This comes from a named wellstream, which has an infinite supply and will make up the extra requirement to fulfil the group's injection target.

In general, therefore, the composition of the injected gas can vary with the injection rate and the supply of available gas from the sources. When the gas comes from a limited source (namely a group or well production stream), the injection gas is taken from the following supplies in the stated order:

1 The source's advance gas stream (if defined), up to its specified maximum rate.

2 The source's production stream, minus the specified fuel usage and the gas reserved for sale if the source is a group (unless the priorities are re-ordered by WTAKEGAS).

3 The injecting group's make-up gas stream.

If the injecting group does not have a make-up gas stream defined, there may possibly be a shortfall between its injection requirement and the injection gas supply. By default, the group will continue to fulfil its injection requirement and the shortfall between supply and injection rate will show up as a negative excess in the gas accounting tables (see below). However, if the WAVAILIM keyword is entered the injection rate will be limited by the available gas supply. A shortfall between supply and injection demand will cause the group to reduce its injection rate to match the available gas supply; the group will then be under **availability** control for gas injection.

When the injected gas composition may vary with the injection rate (when there is make-up or advance gas, or when the injection stream is defined by WINJORD), the injection calculation may be iterated a number of times until the estimated injection rate (from which the composition is determined) and the actual injection rate converges. Similarly, when the WAVAILIM keyword is entered and a limited source of gas is shared between two or more injection groups, the injection calculation may have to be iterated a number of times to ensure that, if any groups inject less than their allocated share of available gas, their unused share of available gas is re-allocated to other groups. The maximum number of these iterations, and their convergence tolerance, is controlled by items 3 and 4 of keyword GCONTOL.

Gas accounting tables

ECLIPSE 300

The gas rate accounting table is generated in the print file at each report step, and provides information on how produced gas is used. Each column is described below:

1 Fuel volume rate

The rate at which produced gas is used for fuel. (Controlled by the GRUPFUEL keyword.)

2 Sales volume rate

The rate at which produced gas is used for sales. (Controlled by the GRUPSALE keyword.)

3 Re-injection volume rate

The rate at which produced gas is re-injected into the reservoir. (Controlled by the GCONINJE and WCONINJE keywords.) This also includes gas which is injected into the reservoir during production balancing of surface gas (controlled by the GCONPROD keyword).

If OPTIONS3 item 108 is non-zero, then this will **not** include any gas injected during the surface gas production balancing group control mode.

4 Excess volume rate

The rate difference between the gas produced and the gas used to meet the fuel, sales and re-injection requirements.

If OPTIONS3 item 108 is non-zero, then gas injected during the surface gas production balancing group control mode will **not** be used in the reporting of the excess (only the re-injected gas during the re-injection group control mode).

5 Sum volume rate

This is the sum of the first four columns and should equal the gas produced by the field for separators that simply accumulate the gas from each stage (This is the default separator configuration.). If gas from an intermediate stage is used as a feed for a subsequent stage and is also a source of re-injection gas, then the sum of the first four columns will not necessarily equal the produced gas rate. This is discussed further in the "Export tables" on page 1224.

6 Export volume rate

The sum of the excess gas rate and the sales gas rate, i.e. the rate produced gas is available for export after the fuel and re-injection requirements have been met.

If OPTIONS3 item 108 is non-zero, then gas injected during the surface gas production balancing group control mode will **not** be used in the reporting of the export (only the re-injected gas during the re-injection group control mode).

7 Advanced gas volume rate

The rate at which advanced gas is used to augment produced gas in a combined re-injection stream. (Controlled by the GADVANCE or WADVANCE keywords.) The advanced gas is only used for re-injection and not as a source of fuel or sales gas.

This column is not output if the field is operating with a surface gas production balancing limit/target.

8 Make-up gas volume rate

The rate at which a make-up gas stream is used to top-up advanced and re-injected produced gas in order to meet an injection target. (Controlled by the GINJGAS and WINJGAS keywords.)

This column is not output if the field is operating with a surface gas production balancing limit/target.

9 Reinj + Adv + Make

The sum of volume rates of re-injected produced gas, advanced gas and make-up gas. Note that this may differ from the total field injection gas rate, which could include wellstreams or mixtures not associated with produced gas.

This column is not output if the field is operating with a surface gas production balancing limit/target.

Export tables

ECLIPSE 300

Export tables are written to the print file at each report step only when both of the following conditions are met:

1 A separator has been defined in which the gas from an intermediate stage is fed into a subsequent stage or gas plant, instead of the stock tank (see item 7of the SEPCOND keyword).

2 Gas from an intermediate stage of a separator is used as the source of re-injection gas (see item 5 of the GINJGAS keyword).

In these circumstances, re-injection from an intermediate separator stage reduces the feed to a subsequent stage and the separator oil rate is thus reduced. This reduced oil rate is reported in the table under the Oil Volume Rate column. This export rate should be distinguished from the produced oil rate, reported elsewhere in the print file, which is the 'formation' oil rate prior to any re-injection work. The difference then between the formation oil and export oil is the oil vapor that is contained in the re-injected gas that would otherwise have been separated in the subsequent separator stage. Note that it is the produced oil rate and not the export oil rate that may be controlled by the GCONPROD keyword. (This is also the case for gas production rates, that is, it is the produced gas rate and not the export gas rate that may be controlled.)

The export gas rate is also included in this table. This is the remaining gas rate after re-injection and fuel usage have been accounted for. For this type of separator configuration the sum of the re-injected, fuel, sales and excess gas rates will not necessarily equal the produced gas rate.

If a Gas Plant table has been specified in the separator train that produces an NGL stream (see GPTABLEN or GPTABLE3), then the Export table also reports the reduced NGL rate.

The export oil, gas and NGL rates and totals may also be written to the summary file using the summary section mnemonics (F,G)E(O,G,N)(R,T).

Note The export rate calculation should be treated with caution when the intermediate separator stage gas is used as part of a mixture to be injected (see the WINJMIX and WINJORD keywords) as it may be unable to take the re-injection into account.

Apportioning group injection targets by guide rate

A group or field target injection rate is apportioned between the subordinate wells or groups in a precisely analogous manner to a production target, as described in "Group production rate targets - guide rate control" on page 1216. A well-group's target is shared between its group-controlled injectors in proportion to their guide rates, after subtracting the injection rate of any injectors under independent control. An injector's guide rate can be set with keyword WGRUPCON, otherwise it will be set equal to the well's injection potential at the beginning of each time step. Wells are placed under independent control if they cannot inject their share of the group's target, or if they have been declared unavailable for group control in keyword WGRUPCON. If a group cannot inject its target rate with its existing wells, the drilling queue is scanned for a suitable injector to open.

Similarly a node-group's (or the field's) target rate is apportioned among its subordinate wells unless a subordinate group either

- has a target injection rate of its own for that phase, or
- has been declared unavailable for injection control at a higher level for that phase, or
- has been given an injection guide rate for that phase.

ECLIPSE 100

Group injection guide rates can be used to control the distribution of the injected phase between various groups in the same way as for production control. But as an alternative to setting the guide rates specifically, they can be calculated at the beginning of each time step according to a choice of two options. One option sets them equal to the groups' voidage production rate. The other option (available only to the 'top-up' phase) sets them equal to the groups' net voidage,

that is the production voidage minus the reservoir volume injection rate in the other phases. An example application is for a platform to re-inject its available gas, distributing it to selected groups in proportion to their voidage production rate. The platform could also inject water to replace the overall production voidage, distributing it to all groups in proportion to their net voidage rate. This would give less water to the groups that have gas injection.

Sales gas production control

ECLIPSE 100

The associated gas from an oil field, which may at the same time have controls on the oil production rate. The option is selected by using the keyword GCONSALE. The sales gas production rate of a group or the field is defined as its total gas production rate, minus its total gas injection rate, minus the gas consumption rate of the group and any subordinate groups, plus the gas import rate of the group and any subordinate groups. (The group gas consumption and import rates are set with the keyword GCONSUMP.)

The sales gas production rate is controlled by re-injecting the surplus gas not required for sale. Sales gas control can be applied independently of any other production controls on the group or field, provided that there is enough injection capacity to inject the surplus gas. There must be one or more gas injectors subordinate to each group with a sales gas target. The group or field is automatically placed under gas re-injection control, and its target re-injection fraction is determined dynamically at each time step to inject the surplus gas. Other limits on gas injection for the group or field (e.g. maximum surface rate) can be simultaneously applied using the keyword GCONINJE. The distribution of the injected gas between any subordinate groups can also be controlled by giving these groups upper rate limits and guide rates for gas injection, with the keyword GCONINJE.

If the group or field is producing more surplus gas than the injectors can handle, the drilling queue will be scanned for a new gas injector to open. Failing this, a choice of actions can be performed at the end of the time step to reduce the gas production in the next time step. These include closing or working over the worst-offending well, or controlling the group or field gas production rate. There are two options for controlling the group/field production rate.

1. The RATE option calculates the gas production rate necessary to meet the sales gas target after allowing for any consumption and the current rate of re-injection, then places the group/field on gas production rate control. The production rate target will remain at this value unless another limit is violated. Note that if the injection capacity were subsequently to increase, the production rate would not increase to take advantage of the increased injectivity.
2. The alternative MAXR option also places the group/field under gas production rate control with a similarly calculated production target. But it also removes the re-injection fraction limit (by setting the re-injection fraction limit to 1) and continues to recalculate the gas production rate target according to what the injection rate was in the previous iteration. This option therefore maximizes the production rate when the system is constrained by its injection capacity, because if the injection capacity were subsequently to increase then it would allow the production rate to increase accordingly while still meeting the sales target.

   There is a flaw in this algorithm if the net sales, import and consumption is zero: stagnation of the injection and production amounts results even if increased injection capacity is available. In this special case, the re-injection fraction limit is set to a larger value (100 by default) so that any increased injection capacity can be used. The value of this increased re-injection fraction limit can be controlled using OPTIONS item 103.

The sales target may not be met exactly, however, because the production rate calculation uses the injection rate from the previous iteration, and also the well rates will not be updated after the first NUPCOL iterations of each time step. The MAXR option will over-write the group's gas production rate limit with a value equal to the sales gas target plus the injection rate plus any gas consumption minus any import rate, while the group is operating on gas production rate control. Any user-defined limit on this quantity will be erased.

The RATE and MAXR options require guide rate group control, and are not allowed if prioritization (keyword GCONPRI) is used to control the group/field's production.

If on the other hand the group or field is not producing enough gas to fulfil its sales requirement, corrective action will be taken at the end of the time step to increase its gas production rate.

- If it is already under gas production rate control, the target rate will be increased by the amount required to meet the sales target.
- If there is a special gas producer (declared with keyword WGASPROD) open and able to increase its production rate, its gas rate target will be increased by a specified increment.
- Otherwise the drilling queue will be scanned for a suitable production well to open, preferentially opening any special gas producers in the queue.

ECLIPSE 300

A more limited means of controlling sales gas production is available in ECLIPSE 300. The rate of produced gas required for sale is specified with keyword GRUPSALE. A fuel usage rate may also be set with keyword GRUPFUEL. The group can then be given a re-injection fraction target of 1.0, and it will re-inject its produced gas minus its fuel and sales requirements. However, there is no feedback on the production controls if there is an excess or shortfall of gas. If there is insufficient injection capacity to re-inject all the surplus gas, the surplus gas that cannot be injected will show up as an "excess" in the gas accounting tables. Similarly, no remedial action will be taken if there is insufficient produced gas to meet the sales target.

Pressure maintenance

The voidage replacement control for group injection (keyword GCONINJE) provides one method of maintaining the reservoir pressure at its current value. This method, however, achieves only approximate pressure maintenance, and for a number of reasons the average reservoir pressure may drift gradually from its original value. A more flexible alternative method of maintaining the reservoir pressure is provided by the keyword GPMAINT.

The GPMAINT keyword enables the user to designate one or more groups to maintain the average pressure of the field or a particular fluid-in-place region at a specified target value. These groups will automatically adjust either their production rate or their injection rate of a nominated phase, in response to the difference between the current pressure and its target value. Thus, for example, a group may adjust its injection rate to bring a region's pressure up to a desired target value and maintain it there. Alternatively, if injection capacity is limited, the group may adjust its production rate to maintain the pressure while the injectors operate at their user-specified targets or limits.

The target production or injection rate of each pressure maintenance group is calculated at every time step from an expression derived from elementary control theory. The control strategy employed is that of a 'proportional + integral controller'. There are two associated control parameters which the user may adjust to achieve an acceptable compromise between the speed of response and the stability of the pressure. The group's flow rate will, of course, honor any additional constraints that the user wishes to apply.

Calculating flow targets for wells under group control

When any form of group control option is being used (keywords GCONPROD, GCONPRI, GCONINJE and GCONSALE), the flow targets of certain wells are directly influenced by the behavior of other wells in the field. For example, the target rates of producers under group control must allow for the production from wells that are under other modes of control within the group. Also injection wells performing re-injection or voidage replacement have flow targets that depend on the behavior of the production wells.

For the group targets to be met exactly, the flow targets of the wells under group control must be re-calculated at every Newton iteration. But when the well flow targets change at each iteration, the convergence rate of the iterations is reduced. This is because the Jacobian matrix calculation does not include the terms representing the mutual dependency of the well rates.

As a compromise, the well flow targets are updated in the first NUPCOL Newton iterations of each time step, and are kept constant for any subsequent iterations. (NUPCOL's value is initialized by keyword NUPCOL in the RUNSPEC section, and can be re-set at any time in the simulation with keyword NUPCOL or GCONTOL in the SCHEDULE section.) Thus the group/field flow targets will be met exactly if the time step converges within NUPCOL iterations. Any changes in the solution after the first NUPCOL iterations may cause the group/field flow rates to drift slightly away from their targets. In general, larger values of NUPCOL will result in the group/field flow targets being met more accurately, but perhaps at the expense of requiring more Newton iterations to achieve convergence.

ECLIPSE 100

The discrepancy between a group/field flow target and its actual flow rate can be limited to within a specified tolerance fraction by using the keyword GCONTOL. If a flow target is not met to within this tolerance, when there is sufficient capacity to meet the target, the well flow rates are re-calculated to meet that target regardless of the value of NUPCOL.

Workover and drilling rig constraints

An option is available to limit the rate at which workovers are performed and new wells are drilled, according to the availability of workover and drilling rigs. Each group may have up to 5 workover rigs and up to 5 drilling rigs assigned to it, using the keyword GRUPRIG. Several groups may share the same rig, if required. The default is that one rig of each type is shared by the whole field.

The availability of a workover rig determines the rate at which automatic workovers can be performed. Automatic workovers are performed after violation of a well or group economic limit, when in ECLIPSE 100 connections on AUTO opening are being opened, and optionally after a group flow limit is exceeded. The time taken for each well to be worked over is set with keyword WORKLIM, and the appropriate rig is made unavailable for other workovers for this duration. A workover on a particular well is postponed if all its available workover rigs are already occupied until the end of the time step. When a workover is postponed, the well continues to operate 'uneconomically' until its rig is available. A flag can be set with keyword WDRILTIM which causes a well to be temporarily closed while a workover takes place.

Similarly, the availability of a drilling rig determines the rate at which new wells can be drilled automatically (for example, from the drilling queue set up with keyword QDRILL or WDRILPRI, or in ECLIPSE 100 using the 'AUTO' option in keyword WCONPROD or WCONINJE). The time taken for each well to be drilled can be set individually with keyword WDRILTIM, and the appropriate rig is not able to drill any other wells for this duration. The drilling of a well is postponed if all its available drilling rigs are already occupied until the end of the time step. It is drilled during the time step in which a rig becomes available.

Vertical flow performance

Calculations involving tubing head pressure are handled using Vertical Flow Performance (VFP) tables, which are supplied as input data. These tables relate the bottom hole pressure of a well to the tubing head pressure at various sets of flowing conditions.

There are two types of VFP table:

1 Injector tables, which describe the vertical flow performance of injection wells, entered with keyword VFPINJ.

These are tables of

bottom hole pressure versus injection rate

versus tubing head pressure

2 Producer tables, which describe the vertical flow performance of production wells,

entered with keyword VFPPROD. Producer tables may also be used describe how the temperature of the fluid at the tubing head varies with flowing conditions in the well (see also WHTEMP keyword).

These are tables of

bottom hole pressure (or) tubing head temperature

versus production rate of oil, liquid or gas

versus tubing head pressure

versus water-oil ratio, water cut, or water-gas ratio

versus gas-oil ratio, gas-liquid ratio, or oil-gas ratio

versus artificial lift quantity

Note that there are three different ways of defining the flow rate, water fraction and gas fraction in producer VFP tables. The engineer should select the most appropriate definitions for the type of well being modeled. For example, if it is an oil well which eventually waters out, a VFP table defined in terms of the liquid rate, water cut and gas-liquid ratio would be suitable. For gas wells, the VFP table should be defined in terms of the gas rate, water-gas ratio and oil-gas ratio.

The artificial lift quantity (ALQ) is a fifth variable that can be used to incorporate an additional look-up parameter, such as the level of artificial lift. For example, if gas lift is being applied, the artificial lift quantity could refer to the injection rate of lift gas into the well bore. Or if the lift mechanism is a down-hole pump, the artificial lift quantity could refer to the pump rating. In ECLIPSE 100 artificial lift can be switched on automatically when a well's production rate falls below a specified limit (see keyword WLIFT), subject to optional constraints on the total group or field lift capacity (see keyword GLIFTLIM).

ECLIPSE 100

By default, VFP tables are interpolated linearly in all variables. However, there is an option to apply cubic spline interpolation to the ALQ (Artificial Lift Quantity) variable. This option is primarily intended to be useful for the Gas Lift Optimization facility, in which the gradient of the wells' performance with respect to the lift gas injection rate is important for deciding which wells could gain the most benefit from an increased lift gas supply. The option is enabled with the VFPTABL keyword.

Note If the actual simulation conditions fall outside the range given in a VFP table, then linear extrapolation is used. This can produce unrealistic results and should be avoided (by ensuring that each VFP table covers all conditions that will arise during the simulation).

Two or more different wells may use the same VFP table, provided they have similar depths, deviations and internal diameters. Any difference between the bottom hole datum depths of the wells and that of the VFP table will be taken into account using a simple hydrostatic correction to the BHP obtained from the VFP table. In ECLIPSE 100 individual wells may also have a further user-defined adjustment made to their BHPs obtained from the table, specified with keyword `WVFPDP`. This could be applied, for example, to help match a well's flow rate at a given THP.

When a well is under THP control, its flow rate is determined by the intersection of its inflow performance relationship (IPR) and the VFP curve interpolated at the current value of its water and gas flowing fractions, on a plane of BHP versus flow rate. Often the VFP curve for a producer is U-shaped, and when there are two intersections the one with the higher flow rate is chosen (the other being unstable). If the IPR does not intersect the VFP curve, the well is deemed to have died. Occasionally, wells dying suddenly in this fashion may cause problems in the simulation. The ECLIPSE 100 keyword `WVFPEXP` controls some possible remedies, either by using explicit values of the water and gas fractions for the interpolation of the tables or by the use of 'stabilized' VFP curves.

Producer tables are input with the keyword `VFPPROD`, and injector tables are input with the keyword `VFPINJ`. A separate interactive program VFP*i* is available to calculate VFP tables, using a choice of multi-phase flow correlations to calculate the pressure losses within the tubing string. The program has been specifically tailored for ECLIPSE, in that it can produce any type of VFP table required by the simulator in a form which can be included directly in the input data. It contains extensive graphical facilities for examining and visually editing VFP tables in cross-section and displaying them in 3D.

Well name and well list template matching

The template matching feature of ECLIPSE enables multiple wells to be referenced via a single template containing wildcard characters. The wildcard characters supported by ECLIPSE are the asterisk (*) to match zero or more characters, and the question mark (?) to match a single non-empty character. When these characters are required at the start of a template they should be preceded by a backslash (\) to differentiate between alternative uses of the symbols, for example to indicate a well list (*) and as an alias for the triggering wells with an action block (?).

Consider the specification of the following wells in the SCHEDULE section.

```
WELSPECS
 'PR10-OIL'   GR-A1   3   2   1*   OIL /
 'PR11-OIL'   GR-A1   3   5   1*   OIL /
 'GAS-PR1'    GR-A2   3   8   1*   GAS /
 'GAS-PR2'    GR-A2   3   3   1*   GAS /
 'IN1-GAS'    GR-A3   4   4   1*   GAS /
 'IN2-GAS'    GR-A3   4   6   1*   GAS /
 'IN10-WAT'   GR-A4   5   2   1*   WAT /
 'IN11-WAT'   GR-A4   6   5   1*   WAT /
 'X'          GR-A5   7   2   1*   LIQ /
 'Y'          GR-A5   7   5   1*   LIQ /
/
```

New well lists consisting of all the wells containing the sub-string ‘PR’ and another consisting of all the wells containing the sub-string ‘IN’ could then be created in the following manner.s

```
WLIST
 '*PR-WELS'   NEW    '\*PR*' /
 '*IN-WELS'   NEW    '\*IN*' /
/
```

Similarly to shut all wells with names composed of seven characters the question mark symbol can be used.

```
WELOPEN
 '\???????'    SHUT /
/
```

In this context the preceding backslash (\) is not strictly required; however, if the keyword appears within an action block it is necessary to differentiate between the wildcard character and the alias for the triggering wells. For example, when the following action is triggered all wells with a water cut greater than 0.25 will be shut, whilst all wells with single character names will have their liquid rate target / limit halved.

```
ACTIONX
 ACT1 100000 /
 WWCT '*' > 0.25 /
/
-- either use the alias ? to select triggering wells
WELOPEN
 '?'   SHUT /
/
-- or match template against all the wells in the field with a single
-- character name
WTMULT
 '\?'   LRAT   0.5 /
/
ENDACTIO
```

To maintain back-compatibility a single asterisk still represents all the wells in the field. Alternatively, the above action could test only a subset of wells, for example those belonging to a well list

```
ACTIONX
 ACT1 100000 /
 WWCT '*PR-WELS' > 0.25 /
/
```

or matching a given template

```
ACTIONX
 ACT1 100000 /
 WWCT '\*GAS*' > 0.25 /
/
```

However, note that it is not possible to use the template matching mechanism to select a subset of the wells triggering the action.

Wildcard characters can be included in well names and well lists in all appropriate `SCHEDULE` keywords. In addition wildcard characters can be included in well names and well lists for `W*` and `LW*` keywords in the `SUMMARY` section. For example

```
-- well water production rate for wells containing the
-- substring 'PR' and the wells X & Y
WWPR
 '\*PR*' 'X' 'Y' /
-- well mode of control for wells belong to well lists
-- containing the substring 'WELS'
WMCTL
 '**WELS*' /
```

Note For all keywords, well name and well list templates containing wildcard characters must be enclosed within quotes.

The `WTEMPQ` keyword can be used to further explore the well template matching mechanism. The `WTEMPQ` keyword writes a list of all the currently defined wells matching the supplied well name or well list template to the PRT file.

ECLIPSE 100

The pre-2006.1 well name and well list behavior can be restored through switch 93 of the `OPTIONS` keyword.

ECLIPSE 300

The pre-2006.1 well name and well list behavior can be restored through switch 94 of the `OPTIONS3` keyword.

RUNSPEC section: pre-95A fixed record structure

Appendix A

Compatibility with earlier releases

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL

To maintain compatibility with older data sets, the fixed record structure used by ECLIPSE 100 before 95A is still recognized. However, this allows a limited set of facilities, and newer RUNSPEC keywords should not be used in conjunction with the fixed record format. Use of this structure in new work is strongly discouraged, and it is recommended that old data sets be converted to the current form of RUNSPEC. To help interpret older data sets, the original form of the RUNSPEC section is described below.

In this format, the RUNSPEC section should occupy 30 records or lines. Record 1 contains the keyword name RUNSPEC starting in column 1. All subsequent odd numbered records are comments for user guidance and are ignored by ECLIPSE. The data in the even numbered records are a mixture of numbers and logical switches. Each data record must be terminated by a slash (/). All data items with the exception of the title, number of blocks, and units have set default values (shown in angled brackets in the detailed description below). If a data record is terminated early with a slash the remaining data items are set to their default values. For most runs, the majority of the data items can be left at their default values.

The minimum data required for the RUNSPEC records are:

Table A.1 Minimum data requirements

Record	Description
Record 2	Title
Record 4	Number of blocks in X,Y,Z directions
Record 6	Phases solved for, that is which saturations, R_s or R_v vary
Record 8	Units
Record 18	Maximum number of wells, groups, wells per group, connections per well.
Record 28	Start date of the simulation
Record 30	Run Simulation (default) or Data Check

An explanatory listing of the contents of the RUNSPEC section will be written to the PRINT file if the RUNSPEC section is followed immediately by the keyword RPTRUNSP, before the GRID keyword.

Detailed description of RUNSPEC records

The contents of each record are listed below. For a more detailed explanation of each item, please refer to the description of the equivalent keyword in the new RUNSPEC format above.

Record 1

Record 1 should contain the RUNSPEC keyword in columns 1 to 8.

Record 2

Record 2 should contain the run title (up to 72 characters). It need not be enclosed in quotes. This record need not be terminated by a slash.

Record 3

Record 3 and all subsequent odd-numbered records (5,7,9,...,29) are ignored by ECLIPSE and may be used for explanatory comments. Note that the first character in these records should **not** be a letter, as lines which begin with a letter may be interpreted as a keyword by some applications.

Record 4

Record 4 contains the grid dimensions, the radial/Cartesian switch, the non-neighbor connection switch, the numerical aquifer dimensioning information, and switches for dual porosity runs.

NDIVIX the number of cells in the X (or R) direction

- DEFAULT: Error

NDIVIY the number of cells in the Y (or Theta) direction

- DEFAULT: Error

NDIVIZ the number of cells in the Z direction (keyword DIMENS)

- DEFAULT: Error

Options

Radial, NNC, Numerical Aquifers, Dual Porosity, Faults

QRDIAL T for radial geometry or F for Cartesian (keyword RADIAL)

- DEFAULT: False

NUMRES The number of reservoirs in the study (keyword NUMRES

- DEFAULT: 1

QNNCON T if there are non-neighbor connections (keyword NONNC)

- DEFAULT: True

MXNAQN The maximum number of lines of numerical aquifer data entered in GRID section keyword AQUNUM (keyword AQUDIMS)

- DEFAULT: 0

MXNAQC The maximum number of lines of connection data for numerical aquifers entered in GRID section keyword AQUCON (keyword AQUDIMS)

- DEFAULT: 0

QDPORO T for a dual porosity run (keyword DUALPORO)

- DEFAULT: False

QDPERM T to allow flow between matrix cells in a dual porosity run (dual porosity/dual permeability) (keyword DUALPERM)

- DEFAULT: False

MFSEGS Maximum number of fault segments (keyword FAULTDIM

- DEFAULT: 0

QGMLTL T if negative direction transmissibility multipliers will be used (keyword GRIDOPTS)

- DEFAULT: False

Record 6

Record 6 contains flags to specify which phases are present (T if the phase is present or F if not), and switches for the Fluid Tracking options.

Phases present:

OIL T or F (keyword OIL)

- DEFAULT: False

WATER T or F (keyword WATER)

- DEFAULT: False

GAS T or F (keyword GAS)

- DEFAULT: False

Dissolved GAS in LIVE OIL T or F (keyword DISGAS)

Vaporized OIL in WET GAS T or F (keyword VAPOIL)

Options

API, Brine, Tracers, Miscible, Polymer, GI, Solvent, Surfactant.

QAPITR Set to T if the API Tracking option is required (keyword API)

- DEFAULT: False

QPOLYM Set to T if the polymer flood option is required (keyword POLYMER)

- DEFAULT: False

QBRINE Set to T if the Brine option is required (keyword BRINE

- DEFAULT: False

NOTRAC The number of passive oil tracers (keyword TRACERS)

- DEFAULT: 0

NWTRAC The number of passive water tracers (keyword TRACERS)

- DEFAULT: 0

NGTRAC The number of passive gas tracers (keyword TRACERS)

- DEFAULT: 0

QMISCL Set to T if the Miscible Gas Flood option is required (keyword MISCIBLE)

- DEFAULT: False

QGGICO Set to T if the GI-model is required (keyword GIMODEL)

- DEFAULT: False

QSOLVT Set to T if the Solvent model (keyword SOLVENT)

- DEFAULT: False

QSFACT Set to T if the Surfactant model is required (keyword SURFACT)

- DEFAULT: False

QQFOAM Set to T if the Foam model is required (keyword FOAM)

- DEFAULT: False

Record 8

Record 8 contains the unit convention for all data in this run

('METRIC', 'FIELD' or 'LAB') (keywords METRIC, FIELD, LAB)

- DEFAULT: Error

Record 10

Record 10 contains the dimensions of the pressure tables.

Note that the PVT properties of live oil (oil with dissolved gas) are entered as 2-dimensional tables - B_o and μ_o against pressure for various values of R_s. Similarly wet gas (gas with vaporized oil) PVT properties are entered as B_g and μ_g against R_v for various values of pressure.

NRPVT The maximum number of R_s nodes in a live oil PVT table (PVTO, PVCO) or R_v nodes in a wet gas PVT table (PVTG).

Set to 1 if there is no live oil or wet gas. (keyword TABDIMS)

- DEFAULT: 20

NPPVT The maximum number of pressure nodes in any PVT table (PVDG, PVDO, PVTO or PVTG) and Rock Compaction table (ROCKTAB). (keyword TABDIMS)

- DEFAULT: 20

NTPVT The number of PVT tables (keyword TABDIMS)

- DEFAULT: 1

Options

Rock compaction, Compressed VE, P_{cap}/surface tension functions, Molecular diffusion, Coal Bed Methane.

NTROCC The number of Rock Compaction tables (keyword ROCKCOMP)

- DEFAULT: 1

QROCKC Set to T if the Rock Compaction option is to be used (keyword ROCKCOMP)

- DEFAULT: False

QRCREV Set to T if rock compaction is reversible (keyword ROCKCOMP)

- DEFAULT: True

QROCKH Set to T if rock compaction is hysteretic (keyword ROCKCOMP) The reversibility switch is ignored if QROCKH is set to T

- DEFAULT: False

QVEO2D Set to T if the Compressed Vertical Equilibrium option is to be used (keyword VE)

- DEFAULT: False

QSURFT Set to T if the oil-gas or oil-water capillary pressures vary with surface tension (keyword SATOPTS)

- DEFAULT: False

QMDIFF Set to T if the molecular diffusion option is required (keyword DIFFUSE)

- DEFAULT: False

QQCOAL Set to T if the ECLIPSE 200 coal bed methane model is required (keyword COAL)

- DEFAULT: False

QQHEAT Set to T if the Temperature option is required (keyword TEMP)

- DEFAULT: False

Record 12

Record 12 Contains the dimensions of the saturation tables, and flags for directional relative permeability, Vertical Equilibrium, Hysteresis, saturation table End-point Scaling and Miscible options.

NSSFUN The maximum number of saturation nodes in any saturation table (keyword TABDIMS)

- DEFAULT: 20

NTSFUN The number of saturation tables (keyword TABDIMS)

- DEFAULT: 1

Options

Directional K_r, VE, Hysteresis, End-point Scaling, Miscible

QDIRKR Set to T if directional relative permeability tables are to be used (keyword SATOPTS)

- DEFAULT: False

QREVKR Set to F if directional relative permeabilities are irreversible (keyword SATOPTS)

- DEFAULT: True

QVEOPT Set to T if the Vertical Equilibrium option is required (keyword VE)

- DEFAULT: False

QHYSTR Set to T if the Hysteresis option is required (keyword SATOPTS)

- DEFAULT: False

QSCALE Set to T if the saturation table End-point Scaling option is to be used (keyword ENDSCALE)

- DEFAULT: False

QSCDIR Set to T if the saturation table End-point Scaling option is directional (keyword ENDSCALE)

- DEFAULT: False

QSCREV Set to F if the saturation table End-point Scaling option is irreversible (keyword ENDSCALE)

- DEFAULT: True

NSENDP The maximum number of nodes in any saturation table end point versus depth table (keyword ENDSCALE)

- DEFAULT: 20

NTENDP The maximum number of saturation table end point versus depth tables (keyword ENDSCALE)

- DEFAULT: 1

QTWOPT Set to T if the two point upstreaming algorithm is to be used to calculate the hydrocarbon component relative permeabilities when the miscible gas flood option is activated (keyword MISCIBLE)

- DEFAULT: False

NSMISC The maximum number of saturation nodes in any table of miscible residual oil saturations versus water saturation (keyword MISCIBLE)

- DEFAULT: 20

NTMISC The maximum number of tables of miscible residual oil saturation versus water saturation and the maximum number of mixing parameter regions (keyword MISCIBLE)

- DEFAULT: 1

Record 14

Record 14 contains dimensions and parameters that control the equilibration calculation (see SOLUTION section).

NDRXVD The maximum number of depth nodes in any RSVD, RVVD or PBVD table (keyword EQLDIMS).

- DEFAULT: 20

NTEQUL The number of equilibration regions (keyword EQLDIMS).

- DEFAULT: 1

NDPRVD The number of depth nodes in the table of phase pressure against depth constructed internally by the equilibration code (keyword EQLDIMS).

- DEFAULT: 100

Options

Quiescence, Threshold Pressure, Mobile Oil, Tracers.

QUIESC Set to T if pressure modifications are required to achieve initial quiescence (keyword EQLOPTS).

- DEFAULT: False

QTHPRS Set to T to enable the Threshold Pressure option (keyword EQLOPTS).

- DEFAULT: False

QREVTH If set to T, the threshold pressures are assumed to apply equally to flow in either direction (keyword EQLOPTS).

- DEFAULT: True

QMOBIL If set to T, the initial mobile fluid critical saturation end-point correction is activated (keyword EQLOPTS)

- DEFAULT: False

NTTRVD The number of initial tracer concentration versus depth tables (keyword EQLDIMS).

- DEFAULT: 1

NSTRVD The maximum number of depth nodes in any initial tracer concentration versus depth table (keyword EQLDIMS).

- DEFAULT: 20

Record 16

Record 16 contains the number of fluid-in-place regions and other flags.

NTFIP The maximum number of fluid-in-place regions defined with keyword FIPNUM in the REGIONS section (keyword REGDIMS).

- DEFAULT: 1

Options

Gravity Imbibition (for dual porosity), Tracers, LGR, Flux, Independent Reservoirs, Additional Fluid-in-place regions, Environmental Tracers.

QGRAID If set to T, gravity imbibition between a fracture cell and its corresponding matrix cell is activated in dual porosity runs (keyword GRAVDR).

- DEFAULT: False

QINTRP If set to T, pressures are interpolated in global cells neighboring a Cartesian local grid refinement to enhance the accuracy of flow calculations between local and global cells (keyword LGR).

- DEFAULT: False

QTDISP If set to T, a numerical diffusion control algorithm will be invoked during the calculation of single phase tracer concentrations in tracer tracking runs (keyword TRACERS).

- DEFAULT: False

NTFREG The maximum number of flux regions (keyword REGDIMS).

- DEFAULT: 0

QTDISK If set to T, the tracer disking option is invoked (keyword DISKING).

- DEFAULT: False

NRFREG The maximum number of independent reservoir regions (keyword REGDIMS).

- DEFAULT: 0

NMFIPR The number of sets of fluid-in-place regions (keyword REGDIMS).

- DEFAULT: 1

NETRAC The number of environmental tracers in the Environmental Tracers option (keyword TRACERS).

- DEFAULT: 0

QVISCD If set to true, the viscous displacement mechanism will be activated in dual porosity runs (keyword VISCD).

Record 18

Record 18 contains well and local grid refinement (LGR) dimensioning data.

NWMAXZ The maximum number of wells in the model (keyword WELLDIMS).

- DEFAULT: 0

NCWMAX The maximum number of grid blocks connected to any one well (keyword WELLDIMS).

- DEFAULT: 0

NGMAXZ The maximum number of groups in the model (keyword WELLDIMS).

- DEFAULT: 0

NWGMAX The maximum number of wells in any one group (keyword WELLDIMS).

- DEFAULT: 0

MAXLGR The maximum number of local grid refinements (keyword LGR)

- DEFAULT: 0

MAXCLS The maximum number of cells in any local grid refinement (keyword LGR)

- DEFAULT: 0

MCOARS The maximum number of amalgamated coarsened cells (keyword LGR)

- DEFAULT: 0

LSTACK The length of the stack of previous search directions used by the linear solver for the LGR simulations (keyword LGR).

- DEFAULT: 10

MAMALG The maximum number of LGR amalgamations (keyword LGR).

- DEFAULT: 0

MXLALG The maximum number of LGRs in any amalgamation (keyword LGR)

- DEFAULT: 0

Record 20

Record 20 contains miscellaneous data for well control.

QEXGOP Set to T if you want to use the extended compressor option or define multiple contract groups for the Gas Field Operations model (keyword GASFIELD).

- DEFAULT: False

NWFRIC Maximum number of wells requiring the Wellbore Friction calculation (keyword FRICTION).

- DEFAULT: 0

NUPCOL Number of non-linear iterations in each time step in which the flow targets of wells under group control or drawdown control are updated (keyword NUPCOL).

- DEFAULT: 3

Record 22

Record 22 contains Production well Vertical Flow Performance table dimensions. (keyword VFPPDIMS)

MXMFLO The maximum number of flow values per table.

- DEFAULT: 0

MXMTHP The maximum number of tubing head pressure values per table.

- DEFAULT: 0

MXMWFR The maximum number of water fraction values per table.

- DEFAULT: 0

MXMGFR The maximum number of gas fraction values per table.

- DEFAULT: 0

MXMALQ The maximum number of artificial lift quantities per table.

- DEFAULT: 0

NMMVFT The maximum number of production well VFP tables.

- DEFAULT: 0

Record 24

Record 24 contains injection well Vertical Flow Performance table dimensions. (keyword VFPIDIMS)

MXSFLO The maximum number of flow values per table.

- DEFAULT: 0

MXSTHP The maximum number of tubing head pressure values per table.

- DEFAULT: 0

NMSVFT The maximum number of injection well VFP tables.

- DEFAULT: 0

Record 26

Record 26 contains dimensioning data for analytic (Fetkovich, Carter-Tracy or constant flux) aquifers and quantities written to the SUMMARY file.

NANAQU The number of analytic aquifers in the model (keyword AQUDIMS).

- DEFAULT: 0

NCAMAX The maximum number of grid blocks connected to any single analytic aquifer (keyword AQUDIMS).

- DEFAULT: 0

NIFTBL Number of influence function tables for Carter-Tracy aquifers (keyword AQUDIMS).

- DEFAULT: 0

NRIFTB Maximum number of rows in any influence function table. (keyword AQUDIMS).

- DEFAULT: 0

NSUMMX Maximum number of quantities written to the SUMMARY file (keyword SMRYDIMS).

- DEFAULT: 3000

QWRSSP Requests a restart index file (keyword RSSPEC)

- DEFAULT: F

Record 28

Record 28 contains the date at the start of the simulation (keyword START)

DAY Day of month

- DEFAULT: 1

MONTH The name of the month

- DEFAULT: 'JAN'

YEAR 4 digits

- DEFAULT: 1983

Record 30

Record 30 contains Data Check switch, Solver Stack size, File types, Disking, Non-Orthogonal correction, and Parallel option domains.

QSOLVE Set to F for data checking mode only (keyword NOSIM).

- DEFAULT: True

NSTACK The length of the stack of previous search directions used by the linear solver (keyword NSTACK).

- DEFAULT: 10

QFMTOU Set to T if output SUMMARY, RESTART, GRID and PSEUDO files are to be formatted (keyword FMTOUT).

- DEFAULT: False

QFMTIN Set to T if the run is to be restarted from a formatted RESTART file (keyword FMTIN).

- DEFAULT: False

QUNOUT Set to T if a single unified SUMMARY file and/or a single unified RESTART file is to be output (keyword UNIFOUT).

- DEFAULT: False

QUNINP Set to T if the run is to be restarted from a unified RESTART file (keyword UNIFIN).

- DEFAULT: False

NGDISK The number of matrix disking groups (keyword DISKING).

- DEFAULT: 0

IDYNAM GRID data disking option (keyword DISKING).

- DEFAULT: 0

`QOPT9P` Set to T if the finite-volume 9-point option is to be used (obsolete keyword `NOPC9`, now removed).

- DEFAULT: False

`NDMAIN` The number of domains for the Shared Memory Parallel option (keyword `PARALLEL`).

- DEFAULT: 1

RPTPROPS: pre-2002A integer controls

Appendix B

RPTPROPS Controls on output from PROPS section

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

This appendix is the pre-2002A listing of the RPTSCHED keywords and lists the integer controls for ECLIPSE 100.

The keyword should be followed by a list of mnemonics which control the output of PROPS section data to the Print file. The list should be terminated by a slash (/).

ECLIPSE 100 Entering the mnemonic NOTHING clears all the reporting flags.

ECLIPSE 100 For additional control over the output, integer values greater than one may be assigned to selected mnemonics using the syntax:

```
mnemonic=integer
```

where there must be no spaces either side of the equals sign.

ECLIPSE 100 Optionally, for back compatibility, a list of integers (terminated with a slash) may be used to indicate the required output. A value less than or equal to zero switches off the corresponding output. Values greater than zero switch the output on. Repeat counts (for example 3*0) can be used if required (with no spaces before or after the asterisk). If fewer than 25 integers are read, the remainder are left unchanged. The integer list cannot be mixed with mnemonics.

Note In ECLIPSE 100 any of the entries marked with an integer control of 9 gives the output of endpoint scaling values.

The following table lists the mnemonics available, and also their equivalent integers for the ECLIPSE 100 integer lists. Mnemonics without an integer in the E100 column are not available in ECLIPSE 100. Mnemonics available in ECLIPSE 300 are identified by an x in the E300 column.

Table B.1 RPTPROPS output controls

Input Keyword or Mnemonic	Output	E100 Integer Control	E300
AQUTAB	Outputs the Carter-Tracy influence tables	25	
DENSITY, GRAVITY, SDENSITY, BDENSITY	Output of surface densities/gravities	7 7 7	x x x
DIFFC or DIFFCOAL	Output of the diffusion data	18	
DETAILVD	Output of the DETAILVD tables		x
ENDPT	Output of endpoint scaling values	9	
ENDPTVD	Output of endpoint versus depth tables		x
FOAM	Output of the foam properties	23	
FRACMOVE	Output of fracture propagation (fracture modeling only)	27	
GINODE etc.	Output of Gi-model saturated fluid properties	12	
HERCHBL	Output of Herschel-Bullkey data (fracture modeling only)	29	
KRG	Output of maximum gas relative permeabilities	9	x
KRGR	Output of gas relative permeabilities at residual	9	x
KRO	Output of maximum oil relative permeability		x
KROR	Output of oil relative permeabilities at residual		x
KRW	Output of maximum water relative permeabilities	9	x
KRWR	Output of water relative permeabilities at residual	9	x
LANGMUIR, LANGSOLV, COALADS	Output of the Langmuir Isotherms	22	
MISC, PMISC	Output of the miscibility function and the pressure miscibility function	17	
OVERBURD	Overburden pressure data		x
PCG	Output of scaled maximum gas capillary pressures	9	x
PCW	Output of scaled maximum water capillary pressures	9	x
PLYVISC etc.	Output of polymer flood model properties	11	
PVDG, PVTG	Output of gas PVT tables	6	
PVDO, PVTO	Output of oil PVT tables	4	x
PVDS	Output of Solvent PVT properties	15	
PVTW	Output of water PVT tables	5	x
PVTWSALT	Output of brine properties	20	x
ROCK,	Output of rock properties	8	

Table B.1 RPTPROPS output controls (Continued)

Input Keyword or Mnemonic	Output	E100 Integer Control	E300
ROCKTAB	Output of rock properties	8	x
ROCKTABH	Output of rock properties (Hysteretic option only)		x
SGCR	Output of scaled critical gas saturations	9	x
SGFN	Output of gas saturation tables	3	x
SGL	Output of scaled connate gas saturations	9	x
SGU	Output of scaled maximum gas saturations	9	x
SOF2, SOF3, TLMIPAR, SORWMIS	Output of oil saturation tables, 3-phase rel-perm tables, Todd-Longstaff mixing parameter, Miscible residual oil saturation tables. If control is set = 2, the output includes a print of the residual oil saturation as a function of water and gas saturations.	1	x
SOGCR	Output of scaled critical oil-in-gas saturations	9	x
SOWCR	Output of scaled critical oil-in-water saturations	9	x
SOLU	Output of aqueous phase property data		x
SPECHEAT, VISCVT	Output of the temperature dependent props	24	
SSFN	Output of Solvent/Gas rel perms	16	
STOG, STOW	Output of Surface Tension properties	14	
SURFVISC etc.	Output of surfactant properties	19	
SWCR	Output of scaled critical water saturations	9	x
SWFN	Output of water saturation tables	2	x
SWL	Output of scaled connate water saturations	9	x
SWU	Output of scaled maximum water saturation	9	x
TRACER	Output of passive fluid tracer names	10	
TRADS, TRDCY, etc.	Output of environmental tracer properties	21	
VDKRG	Velocity Dependent Rel. Perm data for gas.		x
VDKRO	Velocity Dependent Rel. Perm data for oil.		x
VEFRACV, VEFRAC	Output of Vefrac values	13	

ECLIPSE 100

If a value equal to 5 is used for any (or all) of SOF3, SWFN or SGFN, the saturation tables are printed at saturation interval of 5 percent. These values are printed in addition to the user-specified values and are calculated using the user-specified values. Similarly, if a value equal to 5 is used for the fourth, the fifth and the sixth control, the PVT tables are printed at a pressure interval of 200 psi (FIELD), 10 bars (METRIC) or 10 atm (LAB). As before, these values are printed in addition to the user-specified values.

ECLIPSE 300

If directional end point scaling is active via the `DIRECT` and `IRREVERS` arguments of the `ENDSCALE` keyword, then each of the preceding set of mnemonics (except for the capillary pressure mnemonics `PCG` and `PCW`) can be followed by either `X`, `Y` or `Z` (for the `DIRECT` option) and `X-`, `Y-` or `Z-` (if the `IRREVERS` option is active), e.g. `KRGRZ-` etc.

Examples

Example 1

```
RPTPROPS
 SWFN SGFN PVTO PVTW PVDG /
```

Example 2

ECLIPSE 100

Changing the level of output

```
RPTPROPS
 SOF3=5 SWFN=5 SGFN=5 /
```

Example 3

ECLIPSE 100

The old integer format:

```
RPTPROPS
 10*1 10*0 1 1 0 1 /
```

RPTGRID - definition with integer controls

Appendix C

RPTGRID Controls on output from GRID section

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

The keyword should be followed by a string of mnemonics to request output of grid data to the print file. The appropriate keywords used to input grid data, such as `DX`, `DY` etc. may be used as mnemonics. Aliases may be used; for example `DR` may be used in place of `DX`, and `DTHETA` may be used in place of `DY` etc. The data should be terminated by a slash (/).

ECLIPSE 100 Entering the mnemonic `NOTHING` clears all the reporting flags.

ECLIPSE 100 For additional control over the output, integer values greater than one may be assigned to selected mnemonics using the syntax:

```
mnemonic=integer
```

where there must be no spaces either side of the equals sign.

ECLIPSE 100 Optionally, for back compatibility, a list of integers (terminated with a slash) may be used to indicate the required output. A value less than or equal to zero switches off the corresponding output. Values greater than zero switch the output on. Repeat counts (for example 3*0) can be used if required (with no spaces either side of the asterisk). If fewer than 57 integers are read, the remainder are left unchanged. The integer list cannot be mixed with mnemonics.

The following table lists the mnemonics available, and also their equivalent integers for the ECLIPSE 100 integer lists. Mnemonics without an integer in the E100 column are not available in ECLIPSE 100. Mnemonics available in ECLIPSE 300 are identified by an x in the E300 column.

Table C.1 RPTGRID output controls

Input Keyword Or Mnemonic	Output	E100 Integer Control	E300 Availability
ALLNNC	Output of all non-neighbor connections from all sources	24	x
AQUCON	Output of numerical aquifer connections	23	x
AQUNUM	Output of numerical aquifer definitions	22	x
COLLAPSE	Output of VE compression flag	40	
COORD	Output of coordinate lines	19	
COORDSYS	Output of coordinate systems	18	
DEPTH	Output of grid block center depths	14	x
DIFFMMF	Output of Matrix/Fracture diffusivity multipliers	30	
DIFFMX	Output of X diffusivity multipliers	27	x
DIFFMY	Output of Y diffusivity multipliers	28	x
DIFFMZ	Output of Z diffusivity multipliers	29	x
DIFFX or DIFFTX	Output of X diffusivities	31	x
DIFFY or DIFFTY	Output of Y diffusivities	32	x
DIFFZ or DIFFTZ	Output of Z diffusivities	33	x
DOMAINS	Output of domains (Parallel Option). See also the DOMAINS keyword. The output consists of: a. The solver direction, which ECLIPSE calculates automatically but which may be modified by using the SOLVDIRS keyword. b. The split of the outer solver dimension into domains. The split can be entered using the DOMAINS keyword, otherwise ECLIPSE will calculate it automatically.	43	
DPCON or SIGMA	Output of dual porosity connections	39	
DPNUM	Output of dual porosity/single porosity regions	44	
DX	Output of X direction grid block sizes	1	x
DY	Output of Y direction grid block sizes	2	x
DZ	Output of Z direction grid block sizes	3	x
DZMTRXV	Output of D_{zmtrx} values	26	
DZNET	Output of net Z-direction grid block sizes		x

Table C.1 RPTGRID output controls (Continued)

Input Keyword Or Mnemonic	Output (Continued)	E100 Integer Control	E300 Availability
EXTHOST or EXTFIN	Output the host cell map for Pebi local grid refinements supplied using the EXTFIN keyword	57	
FAULTS	Output of fault data	42	x
FLUXNUM	Output of flux region numbers	36	x
FRACTURE	Output of fracture data (fracture modeling only)	61	
HEATTX	X thermal transmissibility		x
HEATTY	Y thermal transmissibility		x
HEATTZ	Z thermal transmissibility		x
HM****	Outputs multipliers for gradient parameters		x
ISOLNUM	Output of independent reservoir region numbers	37	
KOVERD	Output of K/D values (Surfactant Model)	34	
LIMITS	Outputs limits (max/min) of grid arrays		x
LX	Output of matrix block size, X-direction (Viscous Displacement Option)	48	
LY	Output of matrix block size Y-direction	49	
LZ	Output of matrix block size Z-direction	50	
MIDS	Output of cell mid-point depths		x
MINPV or MINPVV	Output of cells with pore volume less than the input minimum values	35	x
MPFANUM	Output the MPFANUM data when using the multi-point flux approximation		x
MPFORTH	Output the multi-point flux k-orthogonality error		x
MPFTRAN	Output the multi-point flux forward transmissibilities		x
MPFTREV	Output the multi-point flux reverse transmissibilities		x
MULTNUM	Output of regions for inter-region transmissibility multipliers	53	x
MULTPV	Output of the pore volume multipliers	41	x
MULTREGP	Output region pore volume multipliers	59	x
MULTREGT	Output of multipliers between regions	47	x
MULTX	Output of X transmissibility multipliers	7	x
MULTY	Output of Y transmissibility multipliers	8	x
MULTZ	Output of Z transmissibility multipliers	9	x
NINENUM	Output the NINENUM data when using the nine-point scheme	56	x
NNC	Output of non-neighbor connections	21	x
NTG	Output of grid block net-to-gross ratios	11	x
PERMX	Output of X direction permeabilities	4	x
PERMY	Output of Y direction permeabilities	5	x

Table C.1 RPTGRID output controls (Continued)

Input Keyword Or Mnemonic	Output (Continued)	E100 Integer Control	E300 Availability
PERMZ	Output of Z direction permeabilities	6	x
PERMXY, PERMYZ or PERMZX	Output of all tensor permeability coefficients		x
PINCH or PINCHREG	Output of pinchout data	51	
PINCHNUM	Output of pinchout region numbers	52	
POLYMER or FOAM	Output of 1/(porosity*area) (Polymer and Foam Models)	38	
PORO	Output of grid block porosities	10	x
PORV	Output of grid block pore volumes	13	x
ROCKVOL	Output of bulk (net and gross) rock volumes to the print file. In addition, if FIPNUM regions are defined, the bulk rock volumes for each region will be output.		x
SIGMAGDV	Output of SIGMAGD values (gravity drainage option)		x
SIGMAK	Output of the processed $\frac{f_x \cdot K_x}{l_x^2} + \frac{f_y \cdot K_y}{l_y^2} + \frac{f_z \cdot K_z}{l_z^2}$ term for the matrix-fracture transmissibility in dual porosity runs, if item 5 of the LTOSIGMA keyword is set to ALL.	55	
SIGMAV	Output of Sigma values	25	x
SOLVMUM	Output the mapping between cells in the input order and the order used for the nested factorization solver when using Pebi grid data and supplied by the SOLVNUM keyword.	54	x
THCONR	Output of thermal conductivities	45	x
TOPS	Output of grid block top depths	12	x
TRANX	Output of X transmissibilities	15	x
TRANY	Output of Y transmissibilities	16	x
TRANZ	Output of Z transmissibilities	17	x
TRTHERM	Output of thermal transmissibilities	46	
ZCORN	Output of cell corner depths	20	

ECLIPSE 100

The coordinate line data (COORD) and the corner depth data (ZCORN) can be written out to the DEBUG file in keyword form, by setting the 19th and 20th controls greater than 1. This keyword data can be read directly into the program. When used in conjunction with the BOUNDARY keyword, a grid corresponding to a small portion of the field can easily be generated. Note that block centred grids input using the DX, DY, DZ, TOPS keywords can also be output in corner point form.

Control 19

0 No output

1 Output to the `PRINT` file.

2 `COORD` keyword written to the `DEBUG` file in floating point format (F12.2)

3 `COORD` keyword written in exponent format (1PE12.6)

Control 20

0 No output

1 Output to the `PRINT` file.

2 `ZCORN` keyword written to the `DEBUG` file in floating point format (F12.2)

3 `ZCORN` keyword written in exponent format (1PE12.6)

Examples

Example 1

```
RPTGRID
 COORD TRANX TRANY TRANZ ALLNNC /
```

Example 2

ECLIPSE 100

Changing the level of output

```
RPTGRID
 COORD=2 ZCORN=2 /
```

Example 3

ECLIPSE 100

The old integer format:

```
RPTGRID
 14*1 3*0 /
```

RPTRST keyword: pre-2002A integer controls

Appendix D

RPTRST

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
x	SCHEDULE

Integer controls of the RPTRST keyword do still work from 2002A onwards, but are no longer documented in the main part of the manual; the use of mnemonics is considered preferable. Users are recommended to use mnemonics as it makes data sets more readable and they are less prone to errors on input. ECLIPSE OFFICE performs this job automatically for users.

For users with old data sets with the RPTRST keyword in the integer format the integer controls are listed below.

Controls on output to the RESTART file

A list of integers (terminated with a slash) may be used to indicate the required output. A value less than or equal to zero switches off the corresponding output. Values greater than zero switch the output on. Repeat counts (for example 3*0) can be used if required (with no spaces before or after the asterisk). If fewer than 29 integers are read, the remainder are left unchanged. The integer list cannot be mixed with mnemonics.

Output to the RESTART file may also be controlled by using the mnemonic RESTART (switch 7) of the RPTSCHED and RPTSOL keywords. However, RPTRST allows greater flexibility when specifying extra data to be written to the RESTART file. If the first control of RPTRST (or mnemonic BASIC) is set to 1 or 2, subsequent use of RPTSCHED will override the output frequency. However, if the first control (or mnemonic BASIC) is set to 3 or more, then any value of 7th switch of RPTSCHED will be ignored.

The following table lists the mnemonics available, and also their equivalent integers in the integer lists.

Table D.1 RPTRST output controls

Input Keyword Or Mnemonic	Output	Integer Control
ALLPROPS	Requests output of the data required by some programs that can interface with ECLIPSE. The output includes fluid densities, viscosities formation volume factors and phase relative permeabilities.	19
BASIC	Output of basic RESTART files If set to 1 then RESTART files are created at every report time, but if the files are not unified, only the last one in the run is kept designed for fail-safe restarts). If set to 2 then RESTART files are created at every report time until this switch is reset, and all are kept. If set to 3 then RESTART files are created every n^{th} report time. The frequency of the restarts is governed by the mnemonic FREQ. If set to 4 then a RESTART file is written at the first report step of each year. Optionally, if the mnemonic FREQ is set >1 then the restart is written only every n^{th} year. If set to 5 then a RESTART file is written at the first report step of each month. Optionally, if the mnemonic FREQ is set >1 then the restart is written only every n^{th} month. If set to 6 then a RESTART file is written at every time-step.	1
COMPRESS	Requests output of the well connection data on a well by well basis to make the RESTART file smaller. This can be useful when a model has large numbers of wells, with just a few wells having a large number of connections.	31
DEN	Output of phase reservoir densities	9
DRAIN	Output of the drainage regions. If set >1 than an addition array is output marking the sinks (typically wells) in the drainage region calculation. (These may also be output to the debug file - See the 25th control in the OPTIONS keyword).	10
FIP	Output of fluids-in-place	3
FLOWS	Output of interblock flows (including non-neighbor connection flows and flows between global and local grids).	2
FREQ=n	Controls the frequency of the RESTART files if BASIC is set >2	6
GIMULT	Outputs RSGI, RVGI, BOGI and BGGI values	25
KRG	Output of gas relative permeabilities	13

Table D.1 RPTRST output controls (Continued)

Input Keyword Or Mnemonic	Output	Integer Control
KRO	Output of oil relative permeabilities	11
KRW	Output of water relative permeabilities	12
NOGRAD	Output of total derivatives for the Gradient option If set > 0 then extra information will not be written to the RESTART file for the Gradient option.	15
NORST	Requests a graphics only RESTART file. If set to 1 then the RESTART file will not contain arrays required for restarting but not usually required for graphical output (for example hysteresis arrays). If set to 2 then the well arrays will not be included on the RESTART file. In a standard black oil case the RESTART file will contain header data and 4 arrays, PRESSURE, SWAT, SGAS and RS (+ any other arrays requested by the other controls in this keyword). The RESTART files dumped while this control is set > 0 cannot be used to restart the run. You could, however, opt to have the restart data written to the Save file (see the mnemonic SAVE, and keyword RESTART).	16
PBPD	Output of bubble point and dew point pressures	5
PCOW, PCOG	Output of capillary pressures	27
PORO	Output of effective porosities (Gradient Option only)	14
POT	Output of phase potentials (including the initial contact corrected potential)	4
PRES	Output of water and gas phase pressures	7
RK or PERMREDN	Permeability reduction for the aqueous phase due to polymer retention in the rock (Polymer Flood model)	29
ROCKC	Output of the modified pore volume and transmissibility multiplier (for the rock compaction model). Setting this to 2 also gives the modified DZ assuming compaction in the Z-direction (for hysteretic compaction).	20
RSSAT	Output of saturated Rs values (dissolved gas-oil ratio).	23
RVSAT	Output of saturated Rv values (vaporized oil-gas ratio).	24

Table D.1 RPTRST output controls (Continued)

Input Keyword Or Mnemonic	Output	Integer Control
SAVE	Requests a RESTART file be written to the Save file. This item has a similar effect to item 1, the BASIC restart control, but governs the writing of time-dependent restart data to the Save file. If set to 1 or 2 then restart data is written to the Save file every report time until this switch is reset. If set to 3 then restart data is written every n^{th} report time. The frequency of the restarts is governed by the mnemonic SFREQ. If set to 4 then restart data is written at the first report step of each year. Optionally, if SFREQ is set >1 then the restart is written only every n^{th} year. If set to 5 then restart data is written at the first report step of each month. Optionally, if SFREQ is set >1 then the restart is written only every n^{th} month. If set to 6 this setting is reserved for use by FrontSim. Note that the Save file **must** be requested either by using the SAVE keyword in the RUNSPEC section or by requesting a Save file in the LOAD keyword, otherwise no restart data will be output to the Save file.	17
SFREQ=n	Controls the frequency of the RESTART files written to the Save file if the mnemonic SAVE is set >2.	18
SGTRAP	Output of the trapped gas saturation Sg_{trap} for the WAG hysteresis option.	21
STREAM=n	Reserved for use by FrontSim	28
SURFBLK	Output of surfactant concentrations (SURFACT), maximum surfactant concentrations (SURFMAX), adsorption (SURFADS) and log (capillary number) (SURFCNM). Surfactant option.	26
VELOCITY	Output water velocity for polymer shear-thinning in the Polymer Flood Model and /or output of gas velocity for shear thinning in the Foam Model	30
VISC	Output of fluid viscosities	8
	Reserved for internal use	22

Examples

Example 1

```
RPTRST
 BASIC=2 FLOWS POT PBPD /
```

Example 2

The old integer format:

```
RPTRST
 2 1 0 1 1 /
```

RPTSCHED: pre-2002A integer controls

Appendix E

RPTSCHED Controls on output from SCHEDULE section

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

This appendix is the pre-2002A listing of the RPTSCHED keywords and lists the integer controls for ECLIPSE 100.

The keyword should be followed by a list of mnemonics which control the output of SCHEDULE section data to the PRINT file. Wherever possible, the mnemonics correspond to the related keywords. The list should be terminated by a slash (/).

The RPTSCHED keyword may be used to reset output controls for the SCHEDULE section as often as required.

ECLIPSE 100 Entering the mnemonic NOTHING clears all the reporting flags.

ECLIPSE 100 For additional control over the output, integer values greater than one may be assigned to selected mnemonics using the syntax:

```
mnemonic=integer
```

where there must be no spaces either side of the equals sign.

ECLIPSE 100 Optionally, for back compatibility, a list of integers (terminated with a slash) may be used to indicate the required output. A value less than or equal to zero switches off the corresponding output. Values greater than zero switch the output on. Repeat counts (for example 3*0) can be used if required (with no spaces before or after the asterisk). If fewer than 79 integers are read, the remainder are left unchanged. The integer list cannot be mixed with mnemonics.

The following table lists the mnemonics available, and also their equivalent integers for the ECLIPSE 100 integer lists. Mnemonics without an integer in the E100 column are not available in ECLIPSE 100. Mnemonics available in ECLIPSE 300 are identified by an x in the E300 column.

Table E.1 RPTSCHED output controls

Input Keyword or Mnemonic	Output	E100 integer control	E300
AIM	Output of AIM status array (1 for implicit cells; 2 for explicit cells)		x
AMF	Output of aqueous mole fractions (CO2SOL option)		x
AQUCT or AQUFET or AQUFETP	Output of Fetkovich or Carter-Tracy aquifer status	13	
BFORG	Output of gas Forchheimer B-factor (VELDEP option)		x
BFORO	Output of oil Forchheimer B-factor (VELDEP option)		x
BGAS	Output of gas reservoir molar densities		x
BOIL	Output of oil reservoir molar densities		x
BWAT	Output of water reservoir molar densities		x
CPU	Output of breakdown of CPU and elapsed time for current run. If set to 1 then report is sent to Print file. If set to 2 then in addition, a copy of the report is sent to the short form output (terminal) If set to 3 then report is sent to short form output only. See "Timing Reports" on page 1093 for an explanation of the outputs. Internal clocks within TS are not recorded during intermediate time step when set to 0.	12	
DENG	Output of grid block gas reservoir density	53	x
DENO	Output of grid block oil reservoir density	51	x
DENW	Output of grid block water reservoir density	52	x
ENERGY	Output total energies/bulk volume (energy/(rock+pore volume)) (ECLIPSE 300 Thermal)		x
FFORG	Output of Forchheimer Non-Darcy flow factor for Gas (VELDEP option)		x
FFORO	Output of Forchheimer Non-Darcy flow factor for Oil (VELDEP option)		x

Table E.1 RPTSCHED output controls (Continued)

Input Keyword or Mnemonic	Output	E100 integer control	E300
FIP	Output of fluid in place and inter-region flow reports. If set to 1: then fluids in place are reported for the whole field. If set to 2: then in addition, a balance sheet is produced for each fluid in place region defined with the FIPNUM keyword. This shows current and initial fluids in place, cumulative flows to wells and other regions and material balance errors. If set to 3: then in addition, a balance sheet is produced for all sets of fluid-in-place regions defined with the FIP keyword.	8	
FIPFOAM	Output of foam fluid in place reports (for the Foam Model) 1 gives a report for the whole field 2 in addition gives a report for each fluid in place region	64	
FIPRESV	Output of region reservoir volume reports	76	
FIPSALT	Output of salt fluid in place reports (for Brine option and salt-sensitive Polymer Flood option) 1 gives a report for the whole field 2 in addition gives a report for each fluid-in-place region	38	
FIPSOL	Output of the Solvent fluid in place reports. 1 gives a report for the whole field 2 in addition gives a report for each fluid-in-place region	43	
FIPSURF	Output of surfactant fluid in place reports (for the Surfactant model) 1 gives a report for the whole field 2 in addition gives a report for each fluid-in-place region	46	
FIPTEMP or FIPHEAT	Output of energy in place reports (for the Temperature option) 1 gives a report for the whole field 2 in addition gives a report for each fluid-in-place region	66	
FIPTR	Output of tracer fluid-in-place reports (for Tracer Tracking) 1 gives a report for the whole field 2 in addition gives a report for each fluid-in-place region	32	

Table E.1 RPTSCHED output controls (Continued)

Input Keyword or Mnemonic	Output	E100 integer control	E300
FIPVE	Output of fluid in place in original oil, water and gas zones (for the VE option)	21	
FLOCn	Output of inter-block component flows (for component n, where n is an integer)		x
FLOE	Output of inter-block energy flow (ECLIPSE 300 Thermal option, using THERMAL keyword)		x
FLOGAS	Output of inter-block gas flows (black oil cases only)	74	x
FLOOIL	Output of inter-block oil flows (black oil cases only)	72	x
FLOSOL	Output of inter-block solvent flows	77	
FLOWAT	Output of inter-block water flows (except for GASWAT and CO2STORE options)	73	x
FMISC	Output of relative permeability miscibility factor (Miscible option)		x
FOAM	Output of grid block foam concentration (for the Foam option)	63	
FOAMADS	Output of grid block foam adsorption (for the Foam option)	68	
FOAMDCY	Output of grid block foam decay (for the Foam option)	69	
FOAMMOB	Output of gas mobility factor due to foam (for the Foam option)	70	
FPC	Output of capillary pressure miscibility factor (Miscible option)		x
FUGG	Output of component fugacities in gas (Activity Driven Diffusion)		x
FUGO	Output of component fugacities in oil (Activity Driven Diffusion)		x
GASCONC	Output of matrix grid block gas concentration (for the Coal Bed Methane option)	54	
GASPOT	Output of gas potentials		x
GDOWAT	Output of gas displaced into oil and water zones (for the VE option) 1 = by regions and field, >1 = by grid blocks	29	
GI	Output of current GI value for GI Pseudo-compositional model. Setting the value greater than 1 will also output the cell RSGI, RVGI, BOGI and BGGI multipliers.	40	
GMEG	Output gas phase energy mobility (ECLIPSE 300 Thermal)		x
GMEO	Output oil phase energy mobility (ECLIPSE 300 Thermal)		x
GMEW	Output water phase energy mobility (ECLIPSE 300 Thermal)		x

Table E.1 RPTSCHED output controls (Continued)

Input Keyword or Mnemonic	Output	E100 integer control	E300
GOC	Output of current gas-oil contact (GOC) depths (for VE option) (Output is AAAAA if contact depth is above top of cell; output is BBBBB if contact depth is below bottom of cell)	23	
GOCDIFF	Output of current GOC depth - initial GOC depth (for VE option)	25	
GRAD	Summary of gradient calculations for the Gradient Option	79	
HGAS	Output gas enthalpies (ECLIPSE 300 Thermal)		x
HOIL	Output oil enthalpies (ECLIPSE 300 Thermal)		x
HWAT	Output water enthalpies (ECLIPSE 300 Thermal)		x
JV	Output of capillary pressure miscibility factor (Miscible option)		x
KRG	Output of grid block gas relative permeabilities	59	x
KRN	Output of grid block solvent relative permeabilities (for the Solvent Model)	78	
KRO	Output of grid block oil relative permeabilities	58	x
KRW	Output of grid block water relative permeabilities	57	x
LAMB	Output saturation dependent thermal conductivity transmissibility multiplier (ECLIPSE 300 Thermal)		x
MLSC	Output of total molar density		x
MULT	Output of grid block pore volumes and transmissibilities if they have been modified	60	x
NCNG	Output of Normalized Capillary Numbers for Gas (VELDEP option)		x
NCNO	Output of Normalized Capillary Numbers for Oil (VELDEP option)		x
NEWTON	Output of a summary of the convergence of the Newton iteration. If set to 1 then report is sent to print file. If set to 2 then in addition, a copy of the report is sent to the short form output (terminal) If set to 3 then report is sent to short form output only.	15	
ODGAS	Output of oil displaced into gas zone (for VE option) 1 = by regions and field, >1 = by grid blocks	27	
ODWAT	Output of oil displaced into water zone (for VE option) 1 = by regions and field, >1 = by grid blocks	28	
OILAPI	Output of grid block oil API values (for API Tracking)	31	

Table E.1 RPTSCHED output controls (Continued)

Input Keyword or Mnemonic	Output	E100 integer control	E300
OILPOT	Output of oil potentials		x
PART	Output of domain partitions (Parallel option)		x
PB or PBUB RSSAT	Output of grid block bubble point pressures 1 outputs bubble point pressures only 2 in addition, outputs the saturated dissolved gas-oil ratios	55	x
PBLK or POLYMER	Output of grid block polymer concentration values (for Polymer Flood option)	34	
PCOG	Output of gas-oil capillary pressures		x
PCOW	Output of oil-water capillary pressures		x
PCGW	Output of gas-water capillary pressures		x
PD or PDEW RVSAT	Output of grid block dew point pressures 1 outputs dew point pressures only 2 in addition, outputs the saturated vaporized oil-gas ratios	56	x
PGAS	Output of grid block gas phase pressures	19	x
PGASD or POTG	Output of grid block gas phase pressures (corrected to datum depth) (See also the DATUM keyword)	20	
PLYADS	Output of current adsorbed polymer concentration on the formation rock (for Polymer Flood option)	36	
POILD or POTO	Output of grid block oil phase pressures (corrected to datum depth)	16	
POTC	Output of initial contact corrected potential	67	
PRES or POIL or PRESSURE	Output of grid block pressures	1	x
PRESMIN	Output of minimum pressures (Irreversible rock compaction option)		x
PSAT	Output of saturation pressures (Compositional mode)		x
PWAT	Output of grid block water phase pressures	17	
PWATD or POTW	Output of grid block water phase pressures (corrected to datum depth)	18	
RECOV	Output of recovery mechanisms (See "Recovery Mechanisms" on page 805)	71	

Table E.1 RPTSCHED output controls (Continued)

Input Keyword or Mnemonic	Output	E100 integer control	E300
RESTART	Output of Restart files If set = 1 then Restart files are created at every report time, but if the files are not unified only the last one in the run is kept (designed for fail-safe restarts). If set > 1 then Restart files are created at every report time until this switch is reset, and all are kept. If set > 2 then the inter block flows of each phase are written to the Restart files. If set > 3 then the current fluids-in-place and phase potentials in each grid cell are written to the Restart files. If set = 6 then Restart files are created at every time-step. See also the keywords RESTART, RPTRST PSEUDOS, FMTOUT, FMTIN, UNIFIN and UNIFOUT. The value of this control will be ignored if the mnemonic BASIC (switch 1) in keyword RPTRST has previously been set > 2. Output of the initial Restart file (at 0.0 Days) is governed by the mnemonic RESTART (switch 7) in keyword RPTSOL, but there must be at least one time step. In ECLIPSE 300 the keyword OUTSOL controls output to the Restart file.	7	
RK	Output of the permeability reduction factor currently being applied to water phase relative permeability (for Polymer Flood option)	37	
ROCKC	Output of the modified pore volume and transmissibility multiplier (for the Rock Compaction model). 2: gives the modified DZ assuming compaction in the Z-direction (for hysteretic compaction) 3: in addition sums DZ for each column of grid blocks (for hysteretic compaction).	41	
RPORV	Output of reservoir pore volumes		x
RS	Output of grid block solution gas-oil ratios	5	x
RV	Output of grid block vapor oil-gas ratios If set >1: With the discretized matrix option active, the solution values into the matrix are also reported.	6	x
SALT	Output of grid block salt concentration values (for Brine option and salt-sensitive Polymer Flood option)	35	x
SGAS	Output of grid block gas saturations	4	x

Table E.1 RPTSCHED output controls (Continued)

Input Keyword or Mnemonic	Output	E100 integer control	E300
SGTRAP	Trapped gas saturation and dynamic trapped gas saturation (for WAG Hysteresis model only)	75	
SL	Output liquid mole fractions (ECLIPSE 300 Thermal)		x
SOIL	Output of grid block oil saturations	2	x
SPENGAS	Output gas internal energy (ECLIPSE 300 Thermal)		x
SPENOIL	Output oil internal energy (ECLIPSE 300 Thermal)		x
SPENWAT	Output water internal energy (ECLIPSE 300 Thermal)		x
SSOL	Output of the current Solvent saturation.	42	
STEN	Output of oil/gas surface tensions (Miscible option)		x
SUMMARY	Output of solution summary at end of each time step If set to 1 then report is sent to Print file. If set to 2 then in addition, a copy of the report is sent to the short form output (terminal) If set to 3 then report is sent to short form output only. For runs using Local Grid Refinement, adding 4 to the above reports the fluid-in-place errors in global cells due to adjacent local grids.	11	
SURFADS	Output of current adsorbed surfactant concentration on the formation rock (for the Surfactant model)	45	
SURFBLK	Output of the surfactant concentrations and the SURFBLK capillary number (for the Surfactant model).	44	
SV	Output vapor mole fractions (ECLIPSE 300 Thermal)		x
SW	Output water mole fractions (ECLIPSE 300 Thermal)		x
SWAT	Output of grid block water saturations	3	x
T2	Output temperature at 2-3 phase transition (ECLIPSE 300 Thermal)		x
T3	Output bubble point (ECLIPSE 300 Thermal)		x
TBLK or TRACER	Output of grid block tracer concentrations (for Tracer Tracking)	33	
TEMP	Output of the grid block temperature (for the Temperature option or ECLIPSE 300 Thermal)	65	x
TET	Output temperatures at last time level (ECLIPSE 300 Thermal)		x
TO	Output temperature at which the oil phase leaves the system (ECLIPSE 300 Thermal)		x
TOTCOMP	Output the total compressibility of the fluid system (excluding the rock)		x
TW	Output temperature at which the water phase leaves the system (ECLIPSE 300 Thermal)		x

Table E.1 RPTSCHED output controls (Continued)

Input Keyword or Mnemonic	Output	E100 integer control	E300
TRADS or TRDCY	Output of tracer adsorption and decay (for Environmental Tracers)	47	
TUNING	Output of simulator control parameters set by TUNING keyword	39	
UGAS	Output gas phase specific volumes (ECLIPSE 300 Thermal)		x
UOIL	Output oil phase specific volumes (ECLIPSE 300 Thermal)		x
UWAT	Output water phase specific volumes (ECLIPSE 300 Thermal)		x
V2	Output vapor mole fraction at the system 2-3 phase transition (ECLIPSE 300 Thermal)		x
VFPPROD	Output of vertical flow performance tables. (In order to obtain this, RPTSCHED must be placed before VFPPROD etc.) This also turns on network convergence messages in the Network option. If set > 1 then a table is also written comparing the maximum dimensions as defined in the VFPPDIMS keyword with those observed in the input tables. If set > 2 then only the table comparing the maximum dimensions is written.	10	
VGAS	Output of grid block gas viscosity	50	x
VMF	Output of vapor mole fractions (fraction of total hydrocarbon moles in gas)		x
VOIL	Output of grid block oil viscosity	48	x
VWAT	Output of grid block water viscosity	49	x
WATPOT	Output of water potentials		x
WDOGAS	Output of water displaced into oil and gas zones (for VE option) 1 = by regions and field, 1= by grid blocks	30	
WELLS	Output of well reports with production and injection rates and totals. 1 gives a report of well flows 2 gives a report of well and connection flows 3 gives a report of layer totals 4 gives a report of layer totals and well flows 5 gives a report of layer totals, well and connection flows.	9	

Table E.1 RPTSCHED output controls (Continued)

Input Keyword or Mnemonic	Output	E100 integer control	E300
WELSPECS or WELSPECL	Output of connection, well and group keyword data from the SCHEDULE section.	14	
WOC	Output of current water-oil contact (WOC) depths (for VE option) (Output is AAAAA if contact depth is above top of cell; output is BBBBB if contact depth is below bottom of cell)	22	
WOCDIFF	Output of current WOC depth - initial WOC depth (for VE option)	24	
WOCGOC	Output of current WOC depth - current GOC depth (for VE option)	26	
XGAS	Output of gas fractional saturations (Sudation option)		x
XMF	Output of liquid component mole fractions (fraction of each component in liquid)		x
XWAT	Output of water fractional saturations (Sudation option)		x
YMF	Output of gas component mole fractions (fraction of each component in vapor)		x
ZMF	Output of total component mole fractions (fraction of each component in liquid)		x

Note Saturation pressures (PSAT) are not normally used in ECLIPSE 300, and their calculation may involve significant extra CPU time.

Note In ECLIPSE 300, to output the distribution of a **tracer**, add the name of the tracer as one of the mnemonics.

Examples for ECLIPSE 100

Example 1

```
RPTSCHED
 'OIL'  'SGAS'  'RS'  'RESTART'  'FIP=2'  'WELSPECS'  'WELLS=2'
 'SUMMARY=2'   'AQUCT'  'NEWTON'  /
```

Example 2

Equivalent to Example 1

```
RPTSCHED
 0 1 0 1 1 0 1 2 2 0 2 2 1 1 1 /
```

Example for ECLIPSE 300

```
RPTSCHED
PRES SWAT SOIL SGAS PSAT XMF YMF /
```

RPTSOL: pre-2002A integer controls

Appendix F

RPTSOL

Controls on output from SOLUTION section

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

This appendix is the pre-2002A version of the description of the RPTSOL keywords and includes the integer controls for ECLIPSE 100.

The keyword should be followed by a list of mnemonics which control the output of SOLUTION section data to the Print file. Wherever possible, the mnemonics correspond to the related keywords. The list should be terminated by a slash (/).

ECLIPSE 100 Entering the mnemonic NOTHING clears all the reporting flags.

ECLIPSE 100 For additional control over the output, integer values greater than one may be assigned to selected mnemonics using the syntax:

```
mnemonic=integer
```

where there must be no spaces either side of the equals sign.

ECLIPSE 100 Optionally, for back compatibility, a list of integers (terminated with a slash) may be used to indicate the required output. A value less than or equal to zero switches off the corresponding output. Values greater than zero switch the output on. Repeat counts (for example 3*0) can be used if required (with no spaces before or after the asterisk). If fewer than 81 integers are read, the remainder are left unchanged. The integer list cannot be mixed with mnemonics.

The following table lists the mnemonics available, and also their equivalent integers for the ECLIPSE 100 integer lists. Mnemonics without an integer in the E100 column are not available in ECLIPSE 100. Mnemonics available in ECLIPSE 300 are identified by an x in the E300 column.

Table F.1 RPTSOL output controls

Input Keyword or Mnemonic	Output	E100 Integer value	E300
AIM	Output AIM status array (1 for implicit cells; 2 for explicit cells)		x
AMF	Output of aqueous mole fractions (CO2SOL option)		x
APIVD	Output of tables of initial API versus depth	14	
AQUFET or AQUFETP or AQUCT or AQUFLUX or AQUANCON (*)	Output of analytic (Fetkovich, Carter-Tracy or Constant Flux) aquifer data (entered using AQUFET, AQUFETP, AQUCT, AQUFLUX) If set to 2 then in addition, data for each aquifer-grid block connection will be printed (input using AQUANCON)	12	x (only * output)
BFORG	Output of gas Forchheimer B-factor (Velocity-dependent flow)		x
BFORO	Output of oil Forchheimer B-factor (Velocity-dependent flow)		x
BGAS	Output of water reservoir molar densities		x
BOIL	Output of oil reservoir molar densities		x
BWAT	Output of gas reservoir molar densities		x
DENG	Output of grid block gas reservoir density	53	x
DENO	Output of grid block oil reservoir density	51	x
DENW	Output of grid block water reservoir density	52	x
ENERGY	Internal energy / bulk volume (THERMAL option)		x
EQUIL	Output of equilibration data	9	
FFORG	Forchheimer Non-Darcy flow factor for Gas (VELDEP option)		x
FFORO	Forchheimer Non-Darcy flow factor for Oil (VELDEP option)		x
FIP	Output of fluid in place reports: If set to 1 then initial fluids in place are reported for the whole field, If set to 2 then in addition, initial fluids in place are reported for each fluid in place region defined with the FIPNUM keyword, If set to 3 then in addition, initial fluids in place are reported for all sets of fluid in place regions defined with the FIP keyword.	8	
FIPFOAM	Output of foam fluid in place reports (for the Foam model) 1 gives a report for the whole field 2 in addition gives a report for each fluid-in-place region	64	

Table F.1 RPTSOL output controls (Continued)

Input Keyword or Mnemonic	Output	E100 Integer value	E300
FIPPLY	Output of polymer/salt fluid in place reports (for Brine option and salt-sensitive Polymer Flood Model) 1 gives a report for the whole field 2 in addition gives a report for each fluid-in-place region	38	
FIPRESV	Output of region reservoir volume reports	76	
FIPSOL	Output of the solvent fluid in place reports. 1 gives a report for the whole field 2 in addition gives a report for each fluid-in-place region	43	
FIPSURF	Output of surfactant fluid in place reports (for the Surfactant Model) 1 gives a report for the whole field 2 in addition gives a report for each fluid-in-place region	46	
FIPTEMP or FIPHEAT	Output of energy in place reports (for the Temperature option) 1 gives a report for the whole field 2 in addition gives a report for each fluid-in-place region	66	
FIPTR	Output of tracer fluid in place reports (for Tracer Tracking) 1 gives a report for the whole field 2 in addition gives a report for each fluid-in-place region	32	
FIPVE	Output of fluid in place in original oil, water and gas zones in the VE option	21	
FLOCn	Output of inter-block component flows (for component n)		x
FLOE	Output of inter-block energy flow (THERMAL option only)		x
FLOGAS	Output of inter block gas flows (black oil cases only)	74	x
FLOOIL	Output of inter block oil flows (black oil cases only)	72	x
FLOSOL	Output of inter block solvent flows (for the Solvent Model)	77	
FLOWAT	Output of inter block water flows	73	x
FMISC	Output of relative permeability miscibility factor (Miscible option)		x
FOAM	Output of grid block foam concentration (for the Foam option)	63	
FOAMADS	Output of grid block foam adsorption (for the Foam option)	68	
FOAMDCY	Output of grid block foam decay (for the Foam option)	69	
FOAMMOB	Output of gas mobility factor due to foam (for the Foam option)	70	
FPC	Output of capillary pressure miscibility factor (Miscible option)		x
FUGG	Output of component fugacities in gas (Activity Driven Diffusion)		x
FUGO	Output of component fugacities in oil (Activity Driven Diffusion)		x

Table F.1 RPTSOL output controls (Continued)

Input Keyword or Mnemonic	Output	E100 Integer value	E300
GASCONC	Output of matrix grid block gas concentration (for the Coal Bed Methane option)	54	
GASPOT	Output of gas potentials		x
GI	Output of initial GI value for condensate/volatile oil model.	40	
GOC	Output of initial gas-oil contact depths in the VE option (Output is AAAAA for contact depth above top of cell; output is BBBBB for contact depth below bottom of cell)	23	
JV	Output of ln(KV) values		x
KRG	Output of grid block gas relative permeabilities	59	x
KRN	Output of grid block solvent relative permeabilities (for the Solvent model)	78	
KRO	Output of grid block oil relative permeabilities	57	x
KRW	Output of grid block water relative permeabilities	58	x
LAMB	Saturation dependent thermal transmissibility multiplier Λ (THERMAL option)		x
MLSC	Output of total molar density		x
NCNG	Output of Normalized Capillary Numbers for Gas (VELDEP option)		x
NCNO	Output of Normalized Capillary Numbers for Oil (VELDEP option)		x
OILAPI	Output of initial grid block API values (for API tracking)	31	
OILPOT	Output of oil potentials		x
PART	Output of domain partitions (Parallel option)		x
PB or PBUB RSSAT	Output of grid block bubble point pressures 1 outputs bubble point pressures only 2 in addition, outputs the saturated dissolved gas-oil ratios	55	x
PBLK or POLYMER	Output of initial grid block polymer concentration values (for polymer flood model)	34	
PCGW	Output of gas-water capillary pressures (GASWAT cases)		x
PCOG	Output of gas-oil capillary pressures		x
PCOW	Output of oil-water capillary pressures		x
PD or PDEW RVSAT	Output of grid block dew point pressures 1 outputs dew point pressures only 2 in addition, outputs the saturated vaporized oil-gas ratios	56	x
PGAS	Output of initial grid block gas phase pressures	19	
PGASD or POTG	Output of initial grid block gas phase pressures corrected to datum depth) (See also the DATUM keyword)	20	
PKRG	Output of processed max. oil relative permeability and the gas relative permeability at residual oil if the end point scaling is activated.	62	

Table F.1 RPTSOL output controls (Continued)

Input Keyword or Mnemonic	Output	E100 Integer value	E300
PKRO	Output of processed max. oil relative permeability and the oil rel-perm at residual gas and water if the end point scaling is activated.	60	
PKRW	Output of processed max. water relative permeability and the water relative permeability at residual oil if the end-point scaling is activated.	61	
PLYADS	Output of polymer concentration currently adsorbed by the rock formation (for Polymer Flood Model)	36	
POIL	Output of initial grid block pressures	1	x
POILD or POTO	Output of initial grid block oil phase pressures (corrected to datum depth)	16	
POTC	Output of initial contact corrected potential	67	
PPCG	Output of the processed maximum gas-oil capillary pressures (End Point Scaling option).	81	
PPCW	Output of the processed maximum oil-water capillary pressures (End Point Scaling option).	80	
PRES or PRESSURE	Output of initial oil pressures	1	x
PRESMIN	Output of minimum pressures (irreversible Rock Compaction option)		x
PSAT	Output of saturation pressures (Compositional mode)		x
PWAT	Output of initial grid block water phase pressures	17	
PWATD or POTW	Output of initial grid block water phase pressures (corrected to datum depth)	18	
RECOV	Output of recovery mechanisms	71	
RESTART	Output of Restart files: If set > 1 then an initial Restart file is created. If set > 2 then the inter block flows of each phase are written to the Restart files. If set > 3 then the current fluids in place and phase potentials in each grid cell are written to the Restart files. (There must be at least one TIME, TSTEP or DATES keyword if an initial Restart file is required; but the simulation mode can be turned off by the keyword NOSIM in the RUNSPEC section) In ECLIPSE 300 the keyword OUTSOL controls output to the Restart file.	7	
RK	Output of permeability reduction factor currently being applied to the water phase (for Polymer Flood Model)	37	

Table F.1 RPTSOL output controls (Continued)

Input Keyword or Mnemonic	Output	E100 Integer value	E300
ROCKC	Output of the modified pore volume and transmissibility multiplier (for the Rock Compaction Model). 2 gives the modified DZ assuming compaction in the Z-direction (for hysteretic compaction) 3 in addition sums DZ for each column of grid blocks (for hysteretic compaction)	41	
ROCKMASS	Output rock mass (Polymer, Surfactant and Environmental Tracer options)	15	
RPORV	Output of reservoir pore volumes		x
RS	Output of initial grid block solution gas-oil ratios	5	x
RSVD or PBVD	Output of tables of either initial R_s versus depth or initial P_b versus depth	10	
RV	Output of initial grid block vapor oil-gas ratios	6	x
RVVD	Output of tables of initial Rv versus depth	11	
SALT	Output of initial grid block salt concentration values (for Brine option and salt-sensitive Polymer Flood Model)	35	x
SALTVD	Output of tables of initial salt concentration versus depth (entered using SALTVD keyword)	29	x
SGAS	Output of initial grid block gas saturations	4	x
SGCO or SGCR or SGL or SGU	Output of the connate, critical and maximum gas saturations for each grid cell if the end-point scaling or the mobile fluid correction option is activated.	27	
SOCRS	Output of critical oil saturations following a surfactant flood (Surfactant option with end point scaling active)	82	
SOGCR	Output of the critical oil-in-gas saturations for each direction for each grid cell if the end point scaling or the mobile fluid correction is activated.	25	
SOIL	Output of initial grid block oil saturations	2	x
SOWCR	Output of the critical oil-in-water saturations for each direction for each grid cell if the end point scaling or the mobile fluid correction is activated.	24	
SSOL	Output of the initial solvent saturation.	42	
STEN	Output of oil/gas surface tensions (Miscible option)		x
SURFADS	Output of current adsorbed surfactant concentration on the formation rock (for the Surfactant Model)	45	
SURFBLK	Output of the Surfactant concentrations (for the Surfactant Model)	44	
SWAT	Output of initial grid block water saturations	3	x
SWATINIT	Output of scaled maximum water-oil capillary pressures generated from initial water distribution entered with keyword SWATINIT.	30	

Table F.1 RPTSOL output controls (Continued)

Input Keyword or Mnemonic	Output	E100 Integer value	E300
SWCO or SWCR or SWL or SWU	Output of the connate, critical and maximum water saturations for each grid cell if the end point scaling or the mobile fluid correction option is activated.	26	
TBLK or TRACER	Output of initial grid block tracer concentrations (for Tracer Tracking)	33	
TEMP	Output of the grid block temperature (for the Temperature option in E100 or the THERMAL option in E300)	65	x
THPRES	Output of threshold pressures (see keywords EQLOPTS and THPRES)	13	
TRADS or TRDCY	Output of tracer adsorption and decay (for Environmental Tracers)	47	
TVDP	Output of initial tracer concentration versus depth tables for the Tracer Tracking option (input using TVDP)	28	
VGAS	Output of grid block gas viscosity	50	x
VMF	Output of vapor mole fractions (fraction of total hydrocarbon moles in gas)		x
VOIL	Output of grid block oil viscosity	48	x
VWAT	Output of grid block water viscosity	49	x
WATPOT	Output of water potentials		x
WOC	Output of initial water-oil contact depths in the VE option (Output is AAAAA for contact depth above top of cell; output is BBBBB for contact depth below bottom of cell)	22	
XGAS	Output of gas fractional saturations (Sudation option)		x
XMF	Output of liquid component mole fractions (fraction of each component in liquid)		x
XWAT	Output of water fractional saturations (Sudation option)		x
YMF	Output of vapor component mole fractions (fraction of each component in gas)		x
SOLVD	Output of initial solution versus depth tables		x
ZMF	Output of total component mole fractions		x

Examples

Examples 1 and 2 refer to ECLIPSE 100 and example 3 refers to ECLIPSE 300.

Example 1

```
RPTSOL
 RESTART=2 SOIL SWAT SGAS /
```

Example 2

```
RPTSOL
 0 1 0 1 0 1 0 2 1 0 1 0 /
```

Example 3

```
RPTSOL
PRESSURE SWAT SOIL SGAS PSAT XMF YMF /
```

ECLIPSE development history

Appendix G

Introduction

This appendix contains an extensive, detailed record of the developments to ECLIPSE over the years.

A common question asked by clients is “When was such-and-such added to the simulators?” This appendix contains as complete a history of the developments of the simulators as we are able to produce.

From the present back to the 2000A release, when the manuals for ECLIPSE 100, 200 and 300 were assimilated, we have the new features document for each year. Prior to that each manual was separate; indeed each ECLIPSE 200 option had its own new features section. These have been amalgamated as best as is possible.

The ECLIPSE 100 and 200 history goes back to 1984 while the ECLIPSE 300 history goes back to 1997.

New facilities in 2009.2 and 2009.1

New facilities in ECLIPSE 100

RUNSPEC section

- The alkaline (high pH) option can be activated using the `ALKALINE` keyword.
- The low salinity option can be activated using the `LOWSALT` keyword.
- The `PETOPTS` keyword has two new mnemonics: (1) `NOPINCHC` that suppresses calculation of pinchout NNCs; (2) `TRANPORV` that informs ECLIPSE 100 that transmissibilities and pore volumes are contained in the .GSG file. The `INITNNC` mnemonic has been extended and now also applies to `CORSNUM` information. If a grid file is also being output, `INITNNC` adds the requested information to the INIT file but does not suppress its output to the grid file.
- The `START` keyword has a new argument for the time at the start of the run.
- The `VISAGE` keyword has been added to activate the VISAGE external geomechanics workflow.
- The `BRINE` keyword now activates the new Multi-Component Brine model if used in conjunction with keyword `ECLMC`.
- The `PSTEADY` keyword has a new option to speed up the simulator's advance to steady-state.
- The `UDTDIMS` keyword has been added to specify dimensions when using user defined tables in the `UDT` keyword
- The BOBERG rock compaction hysteresis can be activated using the `ROCKCOMP` keyword.
- The `MESSAGE` keyword has been added to output messages to the screen and print file.

GRID section

- The keyword `IONROCK` can be used to input rock ion exchange capacity.
- The `RPTGRID` mnemonic `IONROCK` now outputs the rock ion exchange capacity to the print file.
- The keyword `FILEUNIT` can be used to confirm the units of the data set.
- The keyword `PETGRID` has been extended to import local grids for corner point geometry with curved or truncated pillars. In addition, if this keyword is present in the dataset, no grid file will be output.
- The `MESSAGE` keyword has been added to output messages to the screen and print file.

EDIT section

- The keyword `FILEUNIT` can be used to confirm the units of the data set.
- The `MESSAGE` keyword has been added to output messages to the screen and print file.

PROPS section

- Keywords ALSURFST, ALPOLADS, and ALSURFAD can be used to provide look-up tables for multipliers for the surfactant surface tension, the polymer and surfactant adsorption, as functions of the alkaline concentration.
- Keyword ALKADS and ALKROCK can be used to provide the alkaline adsorption parameters.
- Keyword LSALTFNC can be used to provide the low salinity weighting factors versus the salt concentration. These factors are used in the interpolation between low and high salinity saturation functions.
- A new model for Polymer Dead Pore Volume has been added. The pre-2008.2 model can be restored by setting switch 122 in the OPTIONS keyword to 1.
- The first argument of the PLYROCK keyword (Dead Pore Volume) can now be set to a value between 0 and 1, but strictly smaller than 1.
- Keywords IONXROCK and IONXSURF can be used to provide equilibrium constants for ion exchange reactions.
- Keywords SURFESAL and PLYESAL can be used to activate and provide parameters for the effective salinity calculation.
- Keyword ADSORP can be used to supply coefficients for generic analytic adsorption functions, which can be used with the Polymer, Surfactant, Alkaline, Foam, and Environmental Tracer Options.
- Keyword FILEUNIT can be used to confirm the units of the data set.
- Keyword LCUNIT has been added to allow specification of the units for the linear combination of phases.
- The ELASTIC option can be activated using the ROCKOPTS keyword to determine the initial conditions for the HYSTER and BOBERG options.
- The MESSAGE keyword has been added to output messages to the screen and print file.

REGIONS section

- Keyword LWSLTNUM has been added to specify the regions numbers for the low-salinity saturation functions.
- Keyword LSLTWNUM has been added to specify the regions numbers for the water-wet low-salinity saturation functions.
- The keyword FILEUNIT can be used to confirm the units of the data set.
- The MESSAGE keyword has been added to output messages to the screen and print file.

SOLUTION section

- The VISDATES keyword has been added to specify report times at which stress step calculations should be performed in the VISAGE external geomechanics workflow.
- The VISOPTS keyword has been added to specify options for the VISAGE external geomechanics workflow.
- The RPTRST mnemonic PORV outputs the reservoir pore volumes at reference conditions.
- The SALT keyword now specifies initial salt concentration for multiple salts when the Multi-Component Brine model is active.

- The SALTVD keyword now specifies tables of salt concentration versus depth for multiple salts when the Multi-Component Brine model is active.
- The RPTSOL mnemonics ANIONS and CATIONS have been added to output initial anion and divalent cation concentrations to the print file.
- The RPTSOL mnemonics ESALSUR and ESALPLY have been added to output initial surfactant and polymer effective salinities to the print file.
- The RPTSOL mnemonic SALTVD now outputs tables of initial anion and divalent cation equivalent molar concentrations versus depth to the print file when the Multi-Component Brine model is active.
- The RPTRST mnemonic CONV outputs the worst converging cells. Use CONV=integer to specify the number of cells to output.
- The keyword FILEUNIT can be used to confirm the units of the data set.
- The MESSAGE keyword has been added to output messages to the screen and print file.

SUMMARY section

- New mnemonics FTIRF, FTIRS, FTPRF, FTPRS, FTITF, FTITS, FTPTF, FTPTS, FTICF, FTICS, FTPCF, FTPCS, GTIRF, GTIRS, GTPRF, GTPRS, GTITF, GTITS, GTPTF, GTPTS, GTICF, GTICS, GTPCF, GTPCS, WTIRF, WTIRS, WTPRF, WTPRS, WTITF, WTITS, WTPTF, WTPTS, WTICF, WTICS, WTPCF, WTPCS, CTIRF, CTIRS, CTPRF, CTPRS, CTITF, CTITS, CTPTF, CTPTS, CTICF, CTICS, CTPCF and CTPCS have been added to give data on partitioned tracer free and solution state results.
- New mnemonics FTPRALK, FTPTALK, FTIRALK, FTITALK, GTPRALK, GTPTALK, GTIRALK, GTITALK, WTPRALK, WTPTALK, WTIRALK, WTITALK, CTFRALK, CTPTALK, CTITALK, BTCNFALK, BTSTMALK, BTPADALK, BTSADALK and BTADSALK have been added to output alkaline quantities.
- New mnemonics CSPR and CSIR have been added to give data on salt production and injection flow rates for well connections.
- New mnemonic SRRHEAD has been added to give data on hydraulic head values for river reaches.
- New mnemonics FTPRANI, FTPTANI, FTIRANI, FTITANI, GTPRANI, GTPTANI, GTIRANI, GTITANI, WTPRANI, WTPTANI, WTIRANI, WTITANI, CTPTANI, CTITANI, CTFRANI, BTCNFANI, FTPRCAT, FTPTCAT, FTIRCAT, FTITCAT, GTPRCAT, GTPTCAT, GTIRCAT, GTITCAT, WTPRCAT, WTPTCAT, WTIRCAT, WTITCAT, CTPTCAT, CTITCAT, CTFRCAT, BTCNFCAT, BTRADCAT, BTSADCAT have been added to output anion and divalent cation quantities for the Multi-Component Brine model.
- New mnemonics BESALSUR and BESALPLY have been added to output effective salinities.
- New mnemonics WJPR, GJPR, FJPR, WJPT, GJPT, FJPT, WJPRH, GJPRH, FJPRH, WJPTH, GJPTH, FJPTH and GJPRL have been added to give data on linearly combined production rates and totals.
- New mnemonics BPORVMOD and BPERMMOD have been added to output the pore volume and transmissibility multipliers when rock compaction is activated.
- The MESSAGE keyword has been added to output messages to the screen and print file.

SCHEDULE section

- A new item has been added to the keyword NEXTSTPL so that the maximum length of the next time-step can now be applied to every subsequent report step.
- Keyword WALKALIN has been added for Alkaline injection.
- New mnemonic ALKALINE has been added to the RPTSCHED keyword to output alkaline concentrations and adsorption (if activated).
- User Defined Arguments (UDA) have been extended to the following keywords: WALKALIN, WAPI, WFOAM, WPOLYMER, WSALT, WSOLVENT, WSURFACT, WTRACER, WELDRAW, WTMULT, WTADD, GCONSALE.
- A new method of flow scaling is available on the WSEGSICD keyword. Item 11 allows the user to specify the method by which the flow scaling is calculated. Item 12 allows the user to specify the maximum flow rate for which the pressure drop equation holds.
- A new keyword WSEGAICD has been added which can be used to model an Autonomous Inflow Control Device.
- The DATES keyword has a new argument for the time of the report.
- A new option to remove the capillary pressure term from the river inflow equation has been added to the keyword GRDREACH.
- Items 17,18 and 19 have been added to the keyword GRDREACH to specify the choice of river infiltration equation and also the infiltration equation parameters.
- New options have been added to the keyword WLIFTOPT to permit the inclusion of a weighted gas rate term in the denominator of the lift gas gradient calculation and to permit wells to continue to be allocated lift gas after a group gas target has been achieved but whilst an oil rate limit has not been reached.
- The RPTRST mnemonic PORV outputs the reservoir pore volumes at reference conditions.
- The RPTSCHED mnemonic PORV outputs the reservoir pore volumes at reference conditions.
- The RPTSCHED mnemonics ANIONS and CATIONS have been added to output anion and divalent cation concentrations to the print file.
- The RPTSCHED mnemonics ESALSUR and ESALPLY have been added to output surfactant and polymer effective salinities to the print file.
- The WSALT keyword now specifies the injection concentration of multiple salts when the Multi-Component Brine model is active.
- Item 12 has been added to the second record of the TUNING keyword specifying a relative threshold parameter for damping in the ion exchange calculations.
- The RPTRST mnemonic CONV outputs the worst converging cells. Use CONV=integer to specify the number of cells to output.
- It is now possible for the user to specify a production rate target or upper limit for any linear combination of oil, water and gas - referred to as a linearly combined target/limit. A new keyword LINCOM has been added to allow input of the oil, water and gas coefficients used in the linear sum. The following keywords now accept a linearly combined control mode: WCONPROD, GCONPROD, WCONHIST (these 3 keywords have a new item to specify the linearly combined rate target/limit), WHISTCTL, WELCNTL, WELTARG, WLISTARG,

WTADD, WTMULT, GTADD, GTMULT, GRUPTARG. The following keywords now accept a linearly combined control phase: WCUTBACK, WCUTBACT, GCUTBACK, GCUTBACT, WGRUPCON, WBHGLR, GUIDERAT. Keyword GCONPRI has two new items: one to specify a linearly combined target/limit and another to specify the procedure on exceeding this limit.

- The keyword FILEUNIT can be used to confirm the units of the data set.
- The MULTFLT keyword can be used to modify transmissibilities across named faults.
- Item 13 has been added to the second record of the TUNING keyword specifying a weighting factor for updating active tracer concentrations when the tracers are solved in each Newton iteration. See also OPTIONS(126).
- The UDT keyword can be used to specify user defined tables, for use in user defined quantities.
- The UDQ keyword has some additional syntax to allow the use of user defined tables.
- If OPTIONS(133) is set to a non-zero value, well and group guide rates are allowed to change for the first NUPCOL iterations instead of remaining fixed throughout the timestep.
- The MESSAGE keyword has been added to output messages to the screen and print file.
- A new option exists to change the way that the mixture density is calculated for stagnant connections at the bottom of the well and cross-flowing connections. Instead of estimating the density from the average well phase fractions, stagnant connections at the bottom of the well can have their density set equal to that of the lowest open connection and the segregated density calculation can be forced for cross-flowing connections. This option is activated with OPTIONS(139). It is only applicable to non-friction wells using the SEG density calculation method (see item 12 of WELSPECS).

New facilities in ECLIPSE 300

RUNSPEC section

- The PETOPTS keyword has two new mnemonics: (1) NOPINCHC which suppresses calculation of pinchout NNCs; (2) TRANPORV which informs ECLIPSE 300 that transmissibilities and pore volumes are contained in the .GSG file. The INITNNC mnemonic has been extended and now also applies to CORSNUM information. If a grid file is also being output, INITNNC adds the requested information to the INIT file but does not suppress its output to the grid file.
- The START keyword has a new argument for the time at the start of the run.
- The TRPLPORO and NMATRIX keywords can now be used with LGRs.
- The VISAGE keyword has been added to activate the VISAGE external geomechanics workflow.
- The GASSOL keyword has been added to allow multi-component gas to dissolve in the aqueous phase.
- An alternative memory allocation algorithm can be enabled by setting item 163 of OPTIONS3 to 1.
- An alternative viscosity calculation in CO2SOL can be enabled by setting the item 164 of OPTIONS3 to greater than 0.

- An alternative water density calculation in CO2SOL/H2SSOL using Ezrokhi's formula can be enabled by setting item 165 of OPTIONS3 to greater than 0.
- The WSEGSOLV keyword has been added to control parameters related to the multi-segmented well iterative linear solver.
- An item has been added to the WSEGDIMS keyword to specify the maximum number of chord segment links per well.
- The LAGTRAC keyword has been added to facilitate the Lagrangian Tracer Solver. This is a new method of calculating tracer concentrations using streamlines.
- An alternative handling of the Palmer-Mansoori rock model can be enabled by setting item 179 of OPTIONS3.
- The UDTDIMS keyword has been added to specify dimensions when using user defined tables in the UDT keyword
- The default behavior for equilibration option 3 is modified when the reference depth is not set to GOC. The 2008.1 behavior can be restored by setting item 185 of OPTIONS3 to a non-zero value. Note that the pre-2008.1 behavior can be restored by setting item 162 of OPTIONS3 to 1.
- The default behavior for temperature variation with depth in the flash calculation is modified. The pre-2009.1 behavior can be restored by setting item 188 of OPTIONS3 to a non-zero value.
- Options for rigorous density inversion check and correction in flash have been implemented. Currently 2 options are available. Option 1: check is performed post-scalar flash and can be activated by setting item 188 of OPTIONS3 to 1; while in Option 2 during vector flash as well as post-scalar flash and can be activated by setting item 188 of OPTIONS3 to 2.
- The JALS solver has a new mode that improves performance in the majority of cases. It is specifically recommended that mode 2 is used in cases where wells cross parallel domains. The JALS solver in mode 2 is now the default for thermal simulations.
- A new mnemonic NEWTRAN has been added to the PETOPTS keyword. This forces fault transmissibilities to be calculated even for corner point geometry, which is necessary for correct treatment of coarsened cells containing faults.
- A new item 4 has been added to the JALS keyword. This can be used to switch on an extra residual check ("ON").
- If item 214 of OPTIONS3 is set to a non-zero value, this enables the pre-2009.2 John Appleyard Linear Solver (JALS).

GRID section

- The ROCKCONL keyword has been added to allow rocks to be connected to local grid refinements.
- A new item has been added to analytic aquifer keyword AQUCT to specify the temperature in the aquifer. This is only for use with either the thermal or temperature options.

- A new option has been added to specify the thermal conductivities. This involves using new keywords THCROCK, THCOIL, THCGAS, THCWATER and THCSOLID.
  - The THCROCK keyword will specify the rock thermal conductivities.
  - The THCOIL keyword will specify the oil phase thermal conductivities.
  - The THCWATER keyword will specify the water phase thermal conductivities.
  - The THCGAS keyword will specify the gas phase thermal conductivities.
  - The THCSOLID keyword will specify the solid phase thermal conductivities.
- A new mnemonic THCAVE has been added to RPTGRID and RPTGRIDL to output the average cell thermal conductivities to the .PRT file.
- A new mnemonic THCROCK has been added to RPTGRID and RPTGRIDL to output the cell rock thermal conductivities to the .PRT file.
- A new mnemonic THCOIL has been added to RPTGRID and RPTGRIDL to output the cell oil phase thermal conductivities to the .PRT file.
- A new mnemonic THCWATER has been added to RPTGRID and RPTGRIDL to output the cell water phase thermal conductivities to the .PRT file.
- A new mnemonic THCGAS has been added to RPTGRID and RPTGRIDL to output the cell gas phase thermal conductivities to the .PRT file
- A new mnemonic THCSOLID has been added to RPTGRID and RPTGRIDL to output the cell solid phase thermal conductivities to the .PRT file.
- The Flux boundary option can now be used with the Thermal Option. Two options are available: (1) Running a thermal reduced run from a thermal base run; (2) Running a thermal reduced run from a compositional or an ECLIPSE 300 black-oil base run. For option (2), the keyword THFLXTF needs to be used in the USEFLUX model.
- The keyword FILEUNIT can be used to confirm the units of the data set.
- The Vinsome heat loss model has been extended to include a user defined threshold temperature from which the heat loss calculation starts. Refer to item 7 of keyword, ROCKPROP.
- The MULTREGT keyword is now functional in parallel.
- The COPYBOX keyword is now functional in parallel.
- The keyword PETGRID has been extended to import local grids for corner point geometry with curved or truncated pillars. In addition, if this keyword is present in the dataset, no grid file will be output.
- The keyword ROCKFRAC enables the rock volume to be computed as a fraction of the grid cell bulk volume. This can be useful for coal bed, shale gas and thermal runs.

EDIT section

- The keyword FILEUNIT can be used to confirm the units of the data set.

PROPS section

- A new item has been added to analytic aquifer keyword AQUCT to specify the temperature in the aquifer. This is only for use with either the thermal or temperature options.
- The OPTODD3P keyword is used to provide control over the choice of model and the method of detecting saturation direction changes for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.

- The SOLUAQA keyword has been added to specify properties of a water multi-component gas system.
- The VISCAQA keyword has been added to specify coefficients of the Ezrokhi's method for aqua viscosity. This keyword is compatible with the CO2STORE, GASWAT, GASSOL and the thermal KWTABTn options.
- The DENAQA keyword has been added to specify coefficients of the Ezrokhi's method for aqua density. This keyword is compatible with the CO2STORE, GASWAT, GASSOL and the thermal KWTABTn options.
- The keyword RSWVD is now compatible with the keyword GASSOL.
- The keyword DIFFCWAT is now compatible with the keyword GASSOL as well as GASWAT and CO2STORE.
- The keyword FILEUNIT can be used to confirm the units of the data set.
- The PEDTUNE keyword can be used to specify a single set of tuning parameters which act as multipliers on the coefficients used in the mixture molecular weight calculation which forms part of the Pedersen et al. viscosity correlation.
- The PEDTUNER keyword can be used to specify separate sets of tuning parameters for each equation of state region which act as multipliers on the coefficients used in the mixture molecular weight calculation which forms part of the Pedersen et al. viscosity correlation.
- Keyword LCUNIT has been added to allow specification of the units for the linear combination of phases.
- The PCODD3P keyword, which is used with the ODD3P option, has been extended to permit to additional capillary pressure models to be selected using item 1.
- The KWTABTn and KWTEMP keywords can be used with THERMAL and with THERMAL plus CO2STORE runs to define components that dissolve in an aqua phase.
- The keywords SPECHW1 and SPECHW2 can be used to specify aqua enthalpy.
- AMFVD is used to enter the initial aqua composition versus depth for THERMAL runs.
- Equilibrium deviation reactions can now be limited to forward or reverse directions via the 6th argument of the EQLDREAC keyword.
- A correction has been made to the calculation of the minimum oil saturation for the Stone 1 relative permeability model with end-point scaling and with either the SOMGAS or the SOMWAT keyword specified. This correction is only applicable if the critical oil-in-gas (SOGCR) or critical oil-water (SOWCR) end-point saturations are zero. The pre-2009.2 behavior can be restored by setting item 193 of the OPTIONS3 keyword to a non-zero value.

REGIONS section

- The keyword FILEUNIT can be used to confirm the units of the data set.

SOLUTION section

- A new mnemonic ROCKCON has been added to the RPTSOL keyword to output rock connection data within global grid and local grid refinements.
- The analytic aquifer keywords AQUCHGAS, AQUCHWAT, AQUFETP and AQUFLUX are now compatible with the THERMAL option.

- New items have been added to analytic aquifer keywords AQUCHGAS, AQUCHWAT, AQUCT, AQUFETP and AQUFLUX to specify the temperature in the aquifer. This is only for use with either the thermal or temperature options.
- The VISDATES keyword has been added to specify report times at which stress step calculations should be performed in the VISAGE external geomechanics workflow.
- The VISOPTS keyword has been added to specify options for the VISAGE external geomechanics workflow.
- A new mnemonic TCBULK has been added to RPTRST to output the cell average thermal conductivities multiplied by the saturation dependent multiplier.
- New mnemonics BTFORG and BTFORO have been added to the RPTRST keyword to output Forchheimer beta factors.
- The RPTSOL and RPTRST mnemonic PORV outputs the reservoir pore volumes at reference conditions.
- The keyword FILEUNIT can be used to confirm the units of the data set.
- The mnemonics FLOOIL and FLOGAS for the RPTRST and RPTSOL keywords can now be used to output inter-block oil and gas flow rates for compositional models. Also note that FLOWAT is now compatible with the GASWAT and CO2STORE options.
- The mnemonic TSUB has been added to the RPTRST keyword to output sub-cool conditions in a THERMAL simulation.
- The keyword AMF can be used to specify the initial aqua mole fraction in thermal runs with the CO2STORE or KWTABTn keyword.
- The keyword AMF is now compatible with the keyword GASSOL.
- The keyword RSWVD is now compatible with the keyword GASSOL.
- The keyword RSW is now compatible with the keyword GASSOL.
- The keyword AMF is now compatible with the THERMAL option.
- The default behavior of equilibration option 3 is modified when the reference depth is not set to GOC. The 2008.1 behavior can be restored by setting item 185 of OPTIONS3 to a non-zero value. The pre-2008.1 behavior can be restored by setting item 162 of OPTIONS3 to 1.

SUMMARY section

- New mnemonic BTCN has been added to output block tracer concentrations.
- New mnemonics CTFR, CTPR, CTPT, CTPC, CTIR, CTIT and CTIC have been added to output well connection data for tracer tracking option.
- New mnemonics CSFR, CSPR, CSPT, CSPC, CSIR, CSIT and CSIC have been added to output well connection data for the brine tracking option.
- New mnemonics AAQER and AAQET have been added to output thermal energy influx rates and totals for analytic aquifers.
- New mnemonics FAQER and FAQET have been added to output field thermal energy influx rates and totals for analytic aquifers.
- New mnemonic AAQTEMP has been added to output the temperature in analytic aquifers.
- New mnemonic AAQENTH has been added to output the molar enthalpy in analytic aquifers.

- The new mnemonics BIFTWO, BIFTOG and BIFTWG have been added as aliases for the existing mnemonics BIFTOW, BIFTGO and BIFTGW respectively for the ODD3P three-phase relative permeability and capillary pressure hysteresis model option.
- New mnemonics BCFR, BFLOO, BFLOW and BFLOG have been added for the output of inter-block flow rates of components, oil, water and gas, respectively.
- New mnemonics BBTFORG and BBTFORO have been added to output Forchheimer beta factors.
- New mnemonics FCGMI, FCGMM, FCWM and FCOM have been added to output the molar amount of a component, in the specified phase.
- New mnemonics FTLM, GTLM, WTLM, FTVM, GTVM, WTVM, FTML, GTML, WTML, FTMV, GTMV and WTMV have been added to give data on traced mass rate and total for liquid and vapor.
- The new mnemonics WTSUB, WTOFF and WPOFF have been added to report well sub-cool conditions and saturation temperature and pressure offsets in a THERMAL simulation.
- New mnemonics BPORVMOD and BPERMMOD have been added to output the pore volume and transmissibility multipliers when rock compaction is activated.
- Mnemonics with separation stages index: FXMFn, GXMFn, WXMFn, FYMFn, GYMFn, WYMFn, FGnPR, GGnPR, WGnPR, FOnPR, GOnPR, WOnPR, FGnPT, GGnPT, WGnPT, FOnPT, GOnPT and WOnPT have been extended so that the number of stages n can be given values up to 99.
- It is now possible to request a field/group/well component in the oil/gas phase molar rate for stage numbers greater than 9 using the mnemonics FCORn, GCOPn, WCORn, FCGRn, GCGRn and WCGRn.

Note The previous two facilities are not available for local wells.

- New mnemonic CZMF has been added to output the connection component mole fractions.
- Recovery efficiency mnemonics (FOE, FOEW, FOEIW, FOEWW, FOEIG, FOEWG, ROE, ROEW, ROEIW, ROEWW, ROEIG, ROEWG) are now available in ECLIPSE 300.
- New mnemonics WJPR, GJPR, FJPR, WJPT, GJPT, FJPT, WJPRH, GJPRH, FJPRH, WJPTH, GJPTH, FJPTH and GJPRL have been added to give data on linearly combined production rates and totals.
- New mnemonics BAMF, WAMF, GAMF and FAMF have been added for the summary output of the aqueous mole fractions of aqua components.
- New mnemonic FAGIP has been added for the summary output of field gas in place for the aqueous phase. This mnemonic is compatible with the CO2SOL, H2SSOL and GASSOL options.

SCHEDULE section

- The analytic aquifer keywords AQUCHGAS, AQUCHWAT, AQUFETP and AQUFLUX are now compatible with the thermal option.
- New items have been added to analytic aquifer keywords AQUCHGAS, AQUCHWAT, AQUCT, AQUFETP and AQUFLUX to specify the temperature in the aquifer. This is only for use with either the thermal or temperature options.
- User Defined Arguments (UDA) have been extended to the following keywords: WELDRAW, WSALT, WTRACER, WTMULT, WTADD, GRUPFUEL, GRUPSALE.

- The DATES keyword has a new argument for the time of the report.
- A new mnemonic, TCBULK, has been added to RPTRST to output the cell average thermal conductivities multiplied by the saturation dependent multiplier.
- New mnemonics BTFORG and BTFORO have been added to the RPTSCHED keyword to output Forchheimer beta factors.
- New mnemonics BTFORG and BTFORO have been added to RPTRST keyword to output Forchheimer beta factors.
- New options have been added to the keyword WLIFTOPT to permit the inclusion of a weighted gas rate term in the denominator of the lift gas gradient calculation and to permit wells to continue to be allocated lift gas after a group gas target has been achieved but whilst an oil rate limit has not been reached.
- The RPTRST and RPTSCHED mnemonic PORV outputs the reservoir pore volumes at reference conditions.
- The keywords WELLSTRW and WINJW are now activated for options CO2STORE, GASWAT and GASSOL.
- New keyword WSEGSICD has been added which can be used to model the Spiral Inflow Control Device.
- New keyword WSEGAICD has been added which can be used to model the Autonomous Inflow Control Device.
- The keyword FILEUNIT can be used to confirm the units of the data set.
- The mnemonics FLOOIL and FLOGAS for the RPTRST and RPTSCHED keywords can now be used to output inter-block oil and gas flow rates for compositional models. Also note that FLOWAT is now compatible with the GASWAT and CO2STORE options.
- The mnemonic TSUB has been added to the RPTRST and RPTSCHED keywords to output sub-cool conditions in a THERMAL simulation.
- The WSEGLINK keyword has been added to allow looped flowpaths to be specified in multi-segmented wells.
- The MULTFLT keyword can be used to modify transmissibilities across named faults.
- The LAGTRUPD, LTRACE, NOLTRACE and RPTSLN keywords have been added to facilitate the Lagrangian Tracer Solver. This is a new method of calculating tracer concentrations using streamlines.
- The analytic aquifer keywords AQUCT, AQUCHWAT, AQUFETP and AQUFLUX are now compatible with the GASSOL option.
- The UDT keyword can be used to specify user defined tables, for use in user defined quantities.
- The UDQ keyword has some additional syntax to allow the use of user defined tables.
- It is now possible for the user to specify a production rate target or upper limit for any linear combination of oil, water and gas - referred to as a linearly combined target/limit. A new keyword LINCOM has been added to allow input of the oil, water and gas coefficients used in the linear sum. The following keywords now accept a linearly combined control mode: WCONPROD, GCONPROD, WCONHIST, GRUPPROD, WELLPROD (these 5 keywords have a new item to specify the linearly combined rate target/limit), WHISTCTL, WELCNTL,

WELTARG, WLISTARG, WTADD, WTMULT, GTADD, GTMULT and GRUPTARG. The following keywords now accept a linearly combined control phase: WCUTBACK, WGRUPCON, GUIDERAT, WELLGR and GRUPGR. Keyword GCCONPRI has two new items: one to specify a linearly combined target/limit and another to specify the procedure on exceeding this limit.

OPTIMIZE section

- A new SCSA control mode 'ORIG' has been added to OPTPARS item 7. This instructs the optimizer to use the original well control settings from the base run in order to control wells with SCSA parameters in optimization runs.

Aquifers

- New summary mnemonics AAQER and AAQET have been added to output thermal energy influx rates and totals for analytic aquifers.
- New summary mnemonics FAQER and FAQET have been added to output field thermal energy influx rates and totals for analytic aquifers.
- New mnemonic AAQTEMP has been added to output the temperature in analytic aquifers.
- The analytic aquifer keywords AQUCHGAS, AQUCHWAT, AQUFETP and AQUFLUX are now compatible with the thermal option.
- New mnemonic AAQENTH has been added to output the molar enthalpy in analytic aquifers.
- New items have been added to analytic aquifer keywords AQUCHGAS, AQUCHWAT, AQUCT, AQUFETP and AQUFLUX to specify the temperature in the aquifer. This is only for use with either the thermal or temperature options.
- The analytic aquifer keywords AQUCT, AQUCHWAT, AQUFETP and AQUFLUX are now compatible with the GASSOL option.

Thermal

- The ROCKCONL keyword has been added to allow rocks to be connected to local grid refinements.
- A new mnemonic ROCKCON has been added to RPTSOL keyword to output rock connection data within global grid and local grid refinements.
- New summary mnemonics AAQER and AAQET have been added to output thermal energy influx rates and totals for analytic aquifers.
- New summary mnemonics FAQER and FAQET have been added to output field thermal energy influx rates and totals for analytic aquifers.
- New mnemonic AAQTEMP has been added to output the temperature in analytic aquifers.
- The analytic aquifer keywords AQUCHGAS, AQUCHWAT, AQUFETP and AQUFLUX are now compatible with the THERMAL option.
- New mnemonic AAQENTH has been added to output the molar enthalpy in analytic aquifers.
- New items have been added to analytic aquifer keywords AQUCHGAS, AQUCHWAT, AQUCT, AQUFETP and AQUFLUX to specify the temperature in the aquifer. This is only for use with either the thermal or temperature options.

- A new option has been added to specify the thermal conductivities. This involves using new keywords THCROCK, THCOIL, THCGAS, THCWATER and THCSOLID.
  - The THCROCK keyword will specify the rock thermal conductivities.
  - The THCOIL keyword will specify the oil phase thermal conductivities.
  - The THCWATER keyword will specify the water phase thermal conductivities.
  - The THCGAS keyword will specify the gas phase thermal conductivities.
  - The THCSOLID keyword will specify the solid phase thermal conductivities.
- A new mnemonic THCAVE has been added to RPTGRID and RPTGRIDL to output the average cell thermal conductivities to the .PRT file.
- A new mnemonic THCROCK has been added to RPTGRID and RPTGRIDL to output the cell rock thermal conductivities to the .PRT file.
- A new mnemonic THCOIL has been added to RPTGRID and RPTGRIDL to output the cell oil phase thermal conductivities to the .PRT file.
- A new mnemonic THCWATER has been added to RPTGRID and RPTGRIDL to output the cell water phase thermal conductivities to the .PRT file.
- A new mnemonic THCGAS has been added to RPTGRID and RPTGRIDL to output the cell gas phase thermal conductivities to the .PRT file.
- A new mnemonic THCSOLID has been added to RPTGRID and RPTGRIDL to output the cell solid phase thermal conductivities to the .PRT file.
- A new mnemonic TCBULK has been added to RPTRST to output the cell average thermal conductivities multiplied by the saturation dependent multiplier.
- The FLUXOPTS keyword should be used in a full field non-Thermal model (either Blackoil or Compositional) in order to output essential information required to simulate a Thermal flux boundary model.
- The Flux boundary option can now be used with the Thermal Option. Two options are available: (1) Running a thermal reduced run from a thermal base run; (2) Running a thermal reduced run from a compositional or an ECLIPSE 300 black-oil base run. For option (2), the keyword THFLXTF needs to be used in the USEFLUX model.
- The mnemonic TSUB has been added to the RPTRST and RPTSCHED keywords to output sub-cool conditions in a THERMAL simulation.
- The new mnemonics WTSUB, WTOFF and WPOFF have been added to report well sub-cool conditions and saturation temperature and pressure offsets in a THERMAL simulation.
- The Vinsome heat loss model has been extended to include a user defined threshold temperature from which the heat loss calculation starts. Refer to item 7 of keyword, ROCKPROP.
- Solubility of hydrocarbons in an aqua phase can be defined using the keyword KWTABTn and the density and viscosity of the aqua phase can be modeled using the keywords DENAQA and VISCAQA.
- The CO2STORE option is made compatible with THERMAL.
- An alternative solution procedure for simulations with rock-filled cells has been added. Mathematically this is a null change but variations due to numerical rounding may occur. The algorithm can be activated by setting item 161 of OPTIONS3 to 1.
- Equilibrium deviation reactions can now be limited to forward or reverse directions using the 6th argument of the EQLDREAC keyword.

- The JALS solver has a new mode that improves performance in the majority of cases. It is specifically recommended that mode 2 is used in cases where wells cross parallel domains. The JALS solver in mode 2 is now the default for thermal simulations.
- A new item 4 has been added to the JALS keyword. This can be used to switch on an extra residual check (“ON”).

Behavioral changes in ECLIPSE 100

These may cause changes to the results of a simulation, or alter the form or content of the output.

RUNSPEC section

- For the surfactant model enabled using the SURFACT keyword, capillary pressure calculations have been changed. See "The Surfactant Model" in the "ECLIPSE Technical Description". This change may be reverted by entering a non-zero value for the 119th item of the OPTIONS keyword.
- The default value of item 3 of WSEGSOLV has been changed from 1.0E-6 to 1.0E-8.
- If item 134 of the OPTIONS keyword is set to 1, an improved method is used to calculate the local grid geometry for export to the EGRID file. This can make a difference to the size and shape of LGR grid blocks at large depths, especially if the model contains a thin top layer of global grid blocks and especially for LGR grid blocks which are very small relative to the global grid blocks.

GRID section

- For the MULTNUM, FLUXNUM, OPERNUM and PINCHNUM keywords, for dual porosity, OPTIONS option number 117 allows these to be specified only for the matrix or for the fracture instead of automatically applying to both matrix and fracture. The option applies either to all of these keywords (option value 2) or only to MULTNUM (option value 1).
- For the JFUNC and JFUNCR keywords, invalid entries for items 1 and 6 will result in an error condition.
- If OPTIONS switch 141 is set to a non-zero value, the pre-2009.2 behavior for adjusting non-neighbor connections within coarsened cells is restored. For 2009.2 onwards, the NNC ends may be moved to a different coarsened cell to avoid the situation where, after coarsening, a fault NNC no longer crosses its fault. This new algorithm may also cause slight changes in the transmissibility of other NNCs.

PROPS section

- A correction has been made to the calculation of relative permeability for Stone’s Second Model when end-point scaling has also been specified. Pre-2008.2 behavior can be restored by setting the OPTIONS option number 121 to a non-zero value.
- Modifications have been made to the calculation of the residual oil saturation for the Stone I relative permeability model to correct a problem associated with the end-point scaling of the residual saturations. Pre-2009.1 behavior may be restored by entering a non-zero value for the 128th option of the OPTIONS keyword.
- A new model for Polymer Dead Pore Volume has been added. The pre-2008.2 model can be restored by setting switch 122 in the OPTIONS keyword to a positive number.

- An alternative handling of the Palmer-Mansoori rock model has been introduced. The pre-2009.1 model can be restored by setting switch 96 in the OPTIONS keyword to 1.
- An alternative interpolation method for the SOF32D saturation functions family has been added. The pre-2009.1 method can be restored setting switch 135 of the OPTIONS keyword to a positive number.
- A correction to the shape of the curvature of the relative permeability curves has been made when the DPKRMOD keyword is specified for dual porosity models. The correction only applies if the modification parameters defined by items 1 and 2 of this keyword are specified as positive value. The pre-2009.1 behavior can be restored by setting switch 138 of the OPTIONS keyword to a non-zero value.
- When using the STONE1EX keyword with hysteresis, if the exponent values assigned to a given grid block are different for drainage and imbibition, for version 2009.2 onwards a warning is issued. For versions up to and including 2009.1, this resulted in an error condition.
- A correction has been made to the calculation of the interpolated PVTO tables, used for the API tracking option, to avoid the creation of duplicate Rs nodes which has been found to occur for some combinations of Rs nodes and user specified minimum separations between successive Rs nodes. Pre-2009.2 behavior may be restored by setting item 140 of the OPTIONS keyword to a non-zero value.
- If output of the PVTO tables is selected using the RPTPROPS keyword, the interpolated versions of the tables will also be printed if API tracking has been specified.

REGIONS section

- Except for the MISCNUM keyword, the documentation states that all REGIONS section keywords should be assigned a minimum value of 1. However, it is possible that pre-processor or manual edits to a keyword definition, can set a value of 0, even for inactive cells. To be compatible with the treatment of REGIONS section keywords using EQUALS/ADD/MULTIPLY keywords, if REGIONS data is specified using a REGIONS keyword directly, then error messages can be output for both active and inactive cells. To do this, set OPTIONS switch number 131 to a non-zero value.

SOLUTION section

- The calculation of inter-block velocities which is controlled by the RPTRST mnemonics VELOCITY, VELOIL, VELWAT and VELGAS has been changed. Pre-2009.1 behavior can be restored by using item 137 of the OPTIONS keyword.

SUMMARY section

- Memory allocation problems associated with the OFM keyword have been corrected.
- The calculation of the summary quantities FTPC, GTPC, WTPC, CTPC, FTIC, GTIC, WTIC and CTIC for concentrations of partitioned tracers has been changed. The pre-2009.1 calculation can be restored by setting switch 124 in the OPTIONS keyword to a non-zero value.
- The calculation of inter-block velocity summary quantities BVELO, BVELW and BVELG has been changed. The pre-2009.1 calculation can be restored by setting switch 137 in the OPTIONS keyword to a non-zero value.

SCHEDULE section

- In a system containing both dissolved gas and vaporized oil, the maximum rates of increase of R_s and R_v may now be specified using the DRSDT and DRVDT keywords respectively. If the VAPPARS keyword is specified subsequently, the limits imposed by the DRSDT and DRVDT keywords will be disabled. Similarly, the constraints imposed by the specification of the VAPPARS keyword will be disabled by the subsequent specification of either the DRSDT or DRVDT keywords. Pre-2008.2 behavior can be restored by entering a non-zero value for the 118th option of the OPTIONS keyword.
- Previously if a new artificial lift quantity for lift switching was set in WLIFT (item 5) with an increment specified (item 10) a well would only have its lift increased the first time it died on THP control. Now, as long as the maximum lift has not been reached, the lift will be increased each time the well dies. The old behavior can be restored by setting OPTIONS item 132 to a non-zero value.
- COMPFLSH is no longer banned or ignored with multi-segment wells. The mixing tank assumptions in the flash transformation functionality of COMPFLSH is inconsistent with the discretized well bore of multi-segment wells, hence a warning is still output when these features are used together.
    - In the 2008.1 and 2008.2 versions of the code, COMPFLSH records on connections in multi-segment wells were ignored (with a warning), whilst runs were stopped with an error if a WELSEGS (or WELSEGSL) keyword was specified for a well which already had connections with flash transformation ratios. Pre-2008.1, no errors or warnings were produced if these features were used together.
- The calculation of inter-block velocities which is controlled by the RPTRST mnemonics VELOCITY, VELOIL, VELWAT and VELGAS has been changed. Pre-2009.1 behavior can be restored by using item 137 of the OPTIONS keyword.
- A bug was fixed in the calculation of well potentials when the generalized pseudo-pressure special inflow equation is used (see WELSPECS item 8). Pre-2009.1 behavior can be restored by using item 101 of the OPTIONS keyword.
- The calculation of the flowrate limit for wells subject to a drawdown limit (WELDRAW) has been improved. Pre-2009.2 behavior may be restored by using item 144 of the OPTIONS keyword.

Tracers

- The convergence criterion for the standard solver used for tracer tracking has been changed. The Pre-2008.2 behavior can be restored by setting OPTIONS item 120 to 1.
- Prior to 2009.1 the adsorped tracer amount was not properly reset following a convergence failure. If OPTIONS(127) is set to a positive value, the pre-2009.1 behavior for tracer adsorption, using a non-cascade tracer solver, is restored.
- Prior to 2009.1 the adsorped tracer amount was assumed not to decay when using a non-cascade tracer solver. The behavior has been changed to include the decay of adsorped tracer. The pre-2009.1 behavior can be restored by setting OPTIONS(136) to a non-zero value.

Aquifers

- The interpolation of influence function tables for Carter-Tracy analytic aquifers has been improved. The Pre-2009.1 behavior can be restored by setting OPTIONS item 123 to a non-zero value.

Temperature option

- Prior to 2009.1 the conversion from specific heat capacity to energy in place was not correctly calculated for the FIELD unit system. The Pre-2009.1 behavior can be restored by setting `OPTIONS` item 130 to a non-zero value.

Behavioral changes in ECLIPSE 300

RUNSPEC section

- The `JALS` solver in mode 2 is now the default for thermal simulations.In addition, a new item 4 has been added in order to switch on an extra residual check (“ON”). By default, this item will be set to “OFF”
- If `OPTIONS3` switch 197 is set to a non-zero value, the pre-2009.2 behavior for adjusting non-neighbor connections within coarsened cells is restored. For 2009.2 onwards, the NNC ends may be moved to a different coarsened cell to avoid the situation where, after coarsening, a fault NNC no longer crosses its fault. This new algorithm may also cause slight changes in the transmissibility of other NNCs.
- If `OPTIONS3` switch 208 is set to a non-zero value, the pre-2009.2 behavior is restored whereby the `SPECHS` keyword need not be specified for thermal models with components exhibiting solid phases.

GRID section

- For the `JFUNC` and `JFUNCR` keywords, invalid entries for items 1 and 6 will result in an error condition.
- For coal bed methane models, using the multi porosity option and the instant sorption model, each porosity adds on the rock volume. This behavior has changed so that the total rock volume of the pore systems is reset to the average total rock volume. It is possible to change the default by explicit assignment of rock fractions using the `ROCKFRAC` keyword. The pre-2009.2 behavior can be obtained by setting item 212 in the `OPTIONS3` keyword to 1.

PROPS section

- An alternative handling of the Palmer-Mansoori rock model has been added. The pre-2009.1 model can be restored by setting switch 179 in the `OPTIONS3` keyword to 1.
- When using the `STONE1EX` keyword with hysteresis, if the exponent values assigned to a given grid block are different for drainage and imbibition, for version 2009.2 onwards a warning is issued. For versions up to and including 2009.1, this resulted in an error condition.
- The `SPECHS` keyword must now be specified for thermal models in which one or more components exhibit solid phases. Pre 2009.2 behavior can be restored by setting item 208 of the `OPTIONS3` keyword to a non-zero value.

SOLUTION section

- The Li factor calculation during equilibration has been improved. Pre-2009.1 behavior can be restored by setting item 172 of the `OPTIONS3` keyword to a non-zero value.

- The Li factor calculation during solution initialization is now based only on single-phase cells. Pre-2009.1 behavior can be restored by setting item 174 of the OPTIONS3 keyword to a non-zero value.
- For black-oil models, the calculations for mnemonics FLOOIL and FLOGAS for the RPTRST and RPTSOL keywords are now consistent with those for inter-region flow rates. For backward compatibility, option 182 of keyword OPTIONS3 ensures that the previous calculations are used instead.
- For multi-component water models, the calculations for mnemonic FLOWAT for the RPTRST and RPTSOL keywords (and also summary mnemonic RWFR) have been updated to take into account all water components. For backward compatibility with the previous calculations, option 183 of keyword OPTIONS3 may be used.
- For the RPTRST and RPTSOL keywords, LAMB was previously used to output the saturation dependent multiplier for thermal conductivity for the Thermal option. This has now been replaced by TCMULT.
- The default behavior for equilibration option 3 is modified when the reference depth is not set to GOC. The 2008.1 behavior can be restored by setting item 185 of OPTIONS3 to a non-zero value. Note that the pre-2008.1 behavior can restored by setting item 162 of OPTIONS3 to 1.

SUMMARY section

- Region phase production totals (ROPT, RWPT and RGPT) were previously calculated at the beginning of each timestep. They are now calculated at the end as all other summary mnemonics. Setting OPTIONS3 item 176 to a non-zero value will revert to the previous behavior.
- Pre-2009.1 group level sum-then-separate mnemonics (such as GGPRG for example) cause a termination with a BUG message if the group had no relevant open wells and the GSEPREPT keyword had not been used. This is not the case from 2009.1 onwards.
- Prior to the 2008.3 version shut and stopped well potentials were sometimes included in the reported group potential. The pre-2008.3 behavior can be restored by setting item 178 of the OPTIONS3 keyword to a non-zero value.
- In previous versions, BLAMB was used to output the block saturation dependent multiplier for thermal conductivity for the Thermal option. This has now been replaced by BTCMULT.
- Prior to the 2009.1 version, in segments where the drift flux model is not used, segment phase velocities were set equal to the segment mixture velocity. Now, if the phase saturation within such a segment is zero, the segment phase velocity is also zero. The pre-2009.1 behavior can be restored by setting item 187 of the OPTIONS3 keyword to a non-zero value.
- For a coal bed methane model, the FGIPG mnemonics do not include adsorbed gas.

SCHEDULE section

- For simulations using the chemical reactions feature the Appleyard chop is now applied to components when they appear or disappear. The Appleyard chop forces the solution to pass through these points in a more controlled fashion in order to stabilize the solution. The pre 2009.1 behavior can be restored by setting item 158 of the OPTIONS3 keyword to -1.
- Reinjection of gas with well composition under group control (such as GCONINJE with WINJGAS) or group composition under well control now functions correctly. This change may be reversed using item 159 of the OPTIONS3 keyword.

- A warning is now issued if a group has a REIN target set using GCONINJE element 6 and make-up gas set using GINJGAS. Make-up gas will be switched off for this group. This change may be reversed using item 159 of the OPTIONS3 keyword.
- Gas accounting now accounts for the GEFAC keyword. Pre-2008.2 behavior may be recovered using item 159 of the OPTIONS3 keyword.
- If advance gas and the WEFAC and/or GEFAC keywords are both selected advance gas is now calculated correctly. Pre-2008.2 behavior may be recovered using item 159 of the OPTIONS3 keyword.
- If WAVAILIM and the WEFAC and/or GEFAC keywords are both selected the availability limit calculation has been improved. Pre-2008.2 behavior may be recovered using item 159 of the OPTIONS3 keyword.
- If WELSPECS element 5 is defaulted, the bottom hole reference depth now accounts for the COMPORD keyword. Pre-2008.2 behavior may be recovered using item 142 of the OPTIONS3 keyword.
- If there is insufficient gas to meet sales, fuel and reinjection targets then sales and fuel gas will only be reduced if the WAVAILIM keyword has been specified. Pre-2008.2 behavior may be recovered using item 95 of the OPTIONS3 keyword.
- The non-linear damping algorithm for thermal simulations has been improved. The pre 2008.2 default behavior can be restored using item 160 of the OPTIONS3 keyword.
- The RPTSCHED mnemonic FIP now gives the correct material balance when satellite keywords GSATPROD and GSATINJE are used. Pre-2008.2 behavior can be restored using item 168 of the OPTIONS3 keyword.
- Previously item 4 of CVCRIT could only be used to specify a maximum number of linear iterations greater than NSTACK. A value less than NSTACK can now be specified if desired.
- If a reinjection target of greater than 1 is set using GCONINJE any sales or fuel gas targets set using GRUPSALE or GRUPFUEL will now be respected. This change can be reversed using item 173 of the OPTIONS3 keyword.
- History matched and injection wells with zero rates and banned crossflow are now automatically shut. Item 177 of the OPTIONS3 keyword can turn this behavior off.
- For black-oil models, the calculations for mnemonics FLOOIL and FLOGAS for the RPTRST and RPTSCHED keywords are now consistent with those for inter-region flow rates. For backward compatibility, option 182 of keyword OPTIONS3 ensures that the previous calculations are used instead.
- For multi-component water models, the calculations for mnemonic FLOWAT for the RPTRST and RPTSCHED keywords (and also summary mnemonic RWFR) have been updated to take into account all water components. For backward compatibility with the previous calculations, option 183 of keyword OPTIONS3 may be used.
- For the RPTRST and RPTSCHED keywords, LAMB was previously used to output the saturation dependent multiplier for thermal conductivity for the Thermal option. This has now been replaced by TCMULT.
- Oil injection wells have been banned. The pre-2009.1 behavior of allowing oil injectors can be restored by setting item 180 of keyword OPTIONS3 to a non-zero value.
- From 2009.1 onwards, if a well is under availability control (see WAVAILIM) when an unlimited injection fluid source (for example stream) is specified (see WINJGAS), then the control mode is reset to the most recent one specified explicitly for that well. The pre-2009.1 behavior (which leaves the well on availability control) can be restored by setting OPTIONS3 item 186 to a non-zero multiple of 2.

- From 2009.1 onwards, if a group is under availability control (see WAVAILIM) when more than NUPCOL Newton iterations are required, the appropriate phase availability is restored for the group from the last NUPCOL calculated value. Previously the group phase availability limit would always have been the last NUPCOL calculated availability for the **water** phase. The pre-2009.1 behavior can be restored by setting OPTIONS3 item 186 to a non-zero multiple of 3.
- From 2009.1 onwards when an action is triggered in a thermal deck the well enthalpies are set. Previously this was only done at the beginning of each report step. The pre-2009.1 behavior can be restored by setting OPTIONS3 item 190 to a non-zero value.
- The default maximum number of linear iterations has been reduced to 20 when using the JALS solver.
- The gravity density calculation for MSWs has changed. The pre-2009.1 behavior can be restored by setting OPTIONS3, item 191 to a non-zero value.
- The calculation of tracer concentration has been changed if a well or group efficiency factor has been set (WEFAC or GEFAC). The pre-2009.1 behavior can be restored by setting OPTIONS3, item 192 to a non-zero value.
- If item 9 of the OPTIONS3 keyword is set to a non-zero value a zero THP will be assigned for zero rate wells. If set to -1 the pre-2009.1 behavior of this option switch will be obtained.
- The Lagrangian tracer solver (LAGTRAC) gives an improved solution for cases where no water is present in the reservoir.
- The Lagrangian tracer solver (LAGTRAC) calculates when to update and retrace streamlines more efficiently. If item 194 of the OPTIONS3 keyword is positive and divisible by 2, pre-2009.2 behavior is recovered.
- The Lagrangian tracer solver (LAGTRAC) uses an improved calculation of porosity to derive the time-of-flight. If item 194 of the OPTIONS3 keyword is positive and divisible by 3 and 5, pre-2009.2 behavior is recovered.
- The Lagrangian tracer solver (LAGTRAC) will now remain active after a fast restart if it is enabled when the SAVE keyword is used.
- Actions defined using ACTIONX with a dynamic well list defined using WDYNLIST have improved functionality. This change may be reverted to pre-2009.2 behavior by setting OPTIONS3 item 205 to a positive value.
- A change in behavior introduced in DYNAMICR in 2009.1 has been reverted to pre-2009.1 behavior for version 2009.2. 2009.1 could give zero for some block mnemonics where the value should have been non-zero.
- The calculation of well IPR (Inflow Performance Relation) has been improved. The new method improves the results for injecting connections, leading to better results from Open ECLIPSE queries ending IPRA, IPRB and IPRC.
- If item 196 of the OPTIONS3 keyword is set to a non-zero value, this restores the pre-2009.2 behavior for the well flow-dependent skin calculation for VELDEP models using radial geometry.
- Lift gas applied to a well using WCONPROD is now applied if the well is opened using QDRILL. Pre-2009.2 behavior may be recovered by setting OPTIONS3 item 211 to a non-zero value.

OPTIMIZE section

- The default for OPTPARS item 7 (SCSA parameter control mode) has been changed from 'BHP' to 'ORIG'.

ODD3P Relative Permeability and Capillary Pressure Hysteresis Model

- The 2008.2, 2009.1 and 2009.2 versions of the ODD3P relative permeability and capillary pressure hysteresis model include a considerable number of extensions and refinements as compared to the 2008.1 version. These include an alternative method of detecting saturation switching (2008.2), an additional capillary pressure function (2009.1) and corrections to the derivatives associated with the scaling of the saturation end-points (2009.2). Pre-2009.2 behavior may be restored using the OPTODD3P keyword.

Equation of State

- An alternative single-phase splitting initialization algorithm for Gibbs free energy minimization in flash is implanted. This behavior can be activated using a non-zero value for item 167 of the OPTIONS3 keyword.
- The default behavior for temperature variation with depth in the flash calculation is modified. The pre-2009.1 behavior can be restored by setting item 188 of OPTIONS3 to a non-zero value.
- Options for rigorous density inversion check and correction in flash have been implemented. Currently 2 options are available. Option 1: check is performed post-scalar flash and can be activated by setting item 188 of OPTIONS3 to 1; while in Option 2 during vector flash as well as post-scalar flash and this can be activated by setting item 188 of OPTIONS3 to 2.

Multi-Segmented Wells

- Energy initialization for MSW has been changed. Pre-2009.1 behavior can be restored using item 171 of the OPTIONS3 keyword.

Aquifers

- The interpolation of influence function tables for Carter-Tracy analytic aquifers has been improved. The Pre-2009.1 behavior can be restored by setting OPTIONS3 item 170 to a non-zero value.

New keywords in ECLIPSE 100

RUNSPEC section

The new RUNSPEC keywords are:

Table G.1 RUNSPEC keywords

Keyword	Description
ALKALINE	Activates the alkaline (high pH) model.
LOWSALT	Activates the low-salinity option within the Brine option.
VISAGE	Activates the VISAGE geomechanics workflow.
ECLMC	Activates the new Multi-Component Brine model if used in conjunction with keyword BRINE.
TRPLPORO	Activates triple porosity in ECLIPSE 100 and sets the number of vug porosities. This keyword is already in ECLIPSE 300.
UDTDIMS	Sets maximum dimensions for the UDT (user defined tables) keyword.
MESSAGE	Outputs a message to the screen and print file.

GRID section

The new GRID section keywords are:

Table G.2 GRID keywords

Keyword	Description
IONROCK	Specifies rock ion exchange capacity.
FILEUNIT	Confirms the units of the data set.
MESSAGE	Outputs a message to the screen and print file.

EDIT section

The new EDIT section keywords are:

Table G.3 EDIT keywords

Keyword	Description
FILEUNIT	Confirms the units of the data set.
MESSAGE	Outputs a message to the screen and print file.

PROPS section

The new PROPS section keywords are:

Table G.4 PROPS keywords

Keyword	Description
ALSURFST	Tables of multipliers to oil/water surface tension as a function of alkaline concentration.
ALPOLADS	Tables of multipliers to polymer adsorption as a function of alkaline concentration.
ALSURFAD	Tables of multipliers to surfactant adsorption as a function of alkaline concentration.
ALKADS	Alkaline adsorption functions.
ALKROCK	Alkaline-rock interaction properties.
LSALTFNC	Low salinity tables weighting fractions as a function of the salt concentration.
IONXROCK	Ion exchange equilibrium data for rock ion exchanges.
IONXSURF	Ion exchange equilibrium data for surfactant micelle ion exchanges.
SURFESAL	Effective salinity data for surfactant.
PLYESAL	Effective salinity data for polymer.
ADSORP	Adsorption data for generic adsorption functions.
FILEUNIT	Confirms the units of the data set.
LCUNIT	Allow specification of the units for the linear combination of phases.
MESSAGE	Outputs a message to the screen and print file.

REGIONS section

The new REGIONS keywords are

Table G.5 REGIONS keywords

Keyword	Description
LWSLTNUM	Regions for low salinity saturation functions.
LSLTWNUM	Regions for water-wet low-salinity saturation functions.
FILEUNIT	Confirms the units of the data set.
MESSAGE	Outputs a message to the screen and print file.

SOLUTION section

The new SOLUTION keywords are:

Table G.6 SOLUTION keywords

Keyword	Description
VISDATES	Specify VISAGE geomechanics stress step dates.
VISOPTS	Specify VISAGE geomechanics workflow options.
FILEUNIT	Confirms the units of the data set.
MESSAGE	Outputs a message to the screen and print file.

SUMMARY section

The new SUMMARY keywords are:

Table G.7 SUMMARY keywords

	Description
FTIRF	Field Tracer Injection Rate (Free)
FTIRS	Field Tracer Injection Rate (Solution)
FTPRF	Field Tracer Production Rate (Free)
FTPRS	Field Tracer Production Rate (Solution)
FTITF	Field Tracer Injection Total (Free)
FTITS	Field Tracer Injection Total (Solution)
FTPTF	Field Tracer Production Total (Free)
FTPTS	Field Tracer Production Total (Solution)
FTICF	Field Tracer Injection Concentration (Free)
FTICS	Field Tracer Injection Concentration (Solution)
FTPCF	Field Tracer Production Concentration (Free)
FTPCS	Field Tracer Production Concentration (Solution)
GTIRF	Group Tracer Injection Rate (Free)
GTIRS	Group Tracer Injection Rate (Solution)
GTPRF	Group Tracer Production Rate (Free)
GTPRS	Group Tracer Production Rate (Solution)
GTITF	Group Tracer Injection Total (Free)
GTITS	Group Tracer Injection Total (Solution)
GTPTF	Group Tracer Production Total (Free)
GTPTS	Group Tracer Production Total (Solution)
GTICF	Group Tracer Injection Concentration (Free)
GTICS	Group Tracer Injection Concentration (Solution)
GTPCF	Group Tracer Production Concentration (Free)
GTPCS	Group Tracer Production Concentration (Solution)
WTIRF	Well Tracer Injection Rate (Free)

Table G.7 SUMMARY keywords

	Description
WTIRS	Well Tracer Injection Rate (Solution)
WTPRF	Well Tracer Production Rate (Free)
WTPRS	Well Tracer Production Rate (Solution)
WTITF	Well Tracer Injection Total (Free)
WTITS	Well Tracer Injection Total (Solution)
WTPTF	Well Tracer Production Total (Free)
WTPTS	Well Tracer Production Total (Solution)
WTICF	Well Tracer Injection Concentration (Free)
WTICS	Well Tracer Injection Concentration (Solution)
WTPCF	Well Tracer Production Concentration (Free)
WTPCS	Well Tracer Production Concentration (Solution)
CTIRF	Connection Tracer Injection Rate (Free)
CTIRS	Connection Tracer Injection Rate (Solution)
CTPRF	Connection Tracer Production Rate (Free)
CTPRS	Connection Tracer Production Rate (Solution)
CTITF	Connection Tracer Injection Total (Free)
CTITS	Connection Tracer Injection Total (Solution)
CTPTF	Connection Tracer Production Total (Free)
CTPTS	Connection Tracer Production Total (Solution)
CTICF	Connection Tracer Injection Concentration (Free)
CTICS	Connection Tracer Injection Concentration (Solution)
CTPCF	Connection Tracer Production Concentration (Free)
CTPCS	Connection Tracer Production Concentration (Solution)
FTPRALK	Field alkaline production rate
FTIRALK	Field alkaline injection rate
FTPTALK	Field alkaline production total
FTITALK	Field alkaline injection total
GTPRALK	Group alkaline production rate
GTIRALK	Group alkaline injection rate
GTPTALK	Group alkaline production total
GTITALK	Group alkaline injection total
WTPRALK	Well alkaline production rate
WTIRALK	Well alkaline injection rate
WTPTALK	Well alkaline production total
WTITALK	Well alkaline injection total
CTFRALK	Connection alkaline flow rate
CTPTALK	Connection alkaline production total
CTITALK	Connection alkaline injection total
BTCNFALK	Block alkaline concentration
BTADSALK	Block alkaline adsorption

Table G.7 SUMMARY keywords

	Description
BTSTMALK	Block multiplier to water/oil surface tension
BTPADALK	Block multiplier to Polymer adsorption
BTSADALK	Block multiplier to Surfactant adsorption
CSPR	Connection salt production flow rate
CSIR	Connection salt injection flow rate
SRRHEAD	Hydraulic head for river reach
FTPRANI	Field anion production rate
FTIRANI	Field anion injection rate
FTPTANI	Field anion production total
FTITANI	Field anion injection total
GTPRANI	Group anion production rate
GTIRANI	Group anion injection rate
GTPTANI	Group anion production total
GTITANI	Group anion injection total
WTPRANI	Well anion production rate
WTIRANI	Well anion injection rate
WTPTANI	Well anion production total
WTITANI	Well anion injection total
CTPTANI	Connection anion production total
CTITANI	Connection anion injection total
CTFRANI	Connection anion flow rate
BTCNFANI	Block anion concentration
FTPRCAT	Field divalent cation production rate
FTIRCAT	Field divalent cation injection rate
FTPTCAT	Field divalent cation production total
FTITCAT	Field divalent cation injection total
GTPRCAT	Group divalent cation production rate
GTIRCAT	Group divalent cation injection rate
GTPTCAT	Group divalent cation production total
GTITCAT	Group divalent cation injection total
WTPRCAT	Well divalent cation production rate
WTIRCAT	Well divalent cation injection rate
WTPTCAT	Well divalent cation production total
WTITCAT	Well divalent cation injection total
CTPTCAT	Connection divalent cation production total
CTITCAT	Connection divalent cation injection total
CTFRCAT	Connection cation flow rate
BTCNFCAT	Block divalent cation concentration
BTSADCAT	Block divalent cation surfactant associated concentration
BTRADCAT	Block divalent cation rock associated concentration

Table G.7 SUMMARY keywords

	Description
BESALSUR	Block effective salinity for phase behavior and surfactant properties
BESALPLY	Block effective salinity for polymer properties
BPORVMOD	Block pore volume multiplier associated with rock compaction
BPERMMOD	Block transmissibility multiplier associated with rock compaction
BEPVIS (alias BVPOLY)	Block effective polymer viscosity
BEMVIS	Block effective mixture (water/polymer) viscosity
BEWV_POL	Block effective water viscosity (Polymer option)
BEWV_SUR	Block effective water viscosity due to surfactant concentration (Surfactant option)
BEWV_SAL	Block effective water viscosity due to salt concentration (Brine option)
WJPR	Well linearly combined production rate
GJPR	Group linearly combined production rate
FJPR	Field linearly combined production rate
WJPT	Well linearly combined production total
GJPT	Group linearly combined production total
FJPT	Field linearly combined production total
WJPRH	Well linearly combined production rate history
GJPRH	Group linearly combined production rate history
FJPRH	Field linearly combined production rate history
WJPTH	Well linearly combined production total history
GJPTH	Group linearly combined production total history
FJPTH	Field linearly combined production total history
GJPRL	Group linearly combined rate production limit (for reservoir coupling)
MESSAGE	Outputs a message to the screen and print file.

SCHEDULE section

The new SCHEDULE keywords are:

Table G.8 SCHEDULE Keywords

Keyword	Description
WALKALIN	Specifies the alkaline concentration for a water injector.
WSEGAICD	Specifies segments to represent an Autonomous Inflow Control Device.
LINCOM	Specifies the coefficients for a linear combination of oil, water and gas.
FILEUNIT	Confirms the units of the data set.
MULTFLT	Used to modify transmissibilities across named faults.
UDT	Specifies a user defined table.
MESSAGE	Outputs a message to the screen and print file.

New keywords in ECLIPSE 300

RUNSPEC section

The new RUNSPEC keywords are:

Table G.9 RUNSPEC keywords

Keyword	Description
VISAGE	Activates the VISAGE geomechanics workflow
GASSOL	Allows multi-component gas to dissolve in the aqueous phase.
FLUXOPTS	Instructs ECLIPSE to output necessary information to the flux file in order to simulate a reduced thermal run from a non-thermal base case.
WSEGSOLV	Sets parameters for the multi-segment well iterative linear solver.
LAGTRAC	Activates the Lagrangian Tracer Solver.
UDTDIMS	Sets maximum dimensions for the UDT (user defined tables) keyword.

GRID section

The new GRID section keywords are:

Table G.10 GRID keywords

Keyword	Description
ROCKCONL	This keyword has been added to allow rocks to be connected to local grid refinements.
THCROCK	Specifies the rock thermal conductivities for the Thermal option.
THCOIL	Specifies the oil phase thermal conductivities for the Thermal option.
THCWATER	Specifies the water phase thermal conductivities for the Thermal option.
THCGAS	Specifies the gas phase thermal conductivities for the Thermal option.
THCSOLID	Specifies the solid phase thermal conductivities for the Thermal option.
THFLXTF	Specifies the ratio between the heat transmissibility and the flow transmissibility for the use with both the Thermal option and the Flux boundary condition option.
FILEUNIT	Confirms the units of the data set.
ROCKFRAC	Set rock volume from a fractional value multiplied with the grid cell bulk volume.

EDIT section

The new EDIT section keywords are:

Table G.11 EDIT keywords

Keyword	Description
FILEUNIT	Confirms the units of the data set.

PROPS section

The new PROPS section keywords are;

Table G.12 PROPS keywords

Keyword	Description
OPTODD3P	Options for control of the ODD3P three-phase relative permeability and capillary pressure calculation.
PEDTUNE	Provides a single set of multiplicative tuning parameters to modify the mixture molecular weight calculation used in the Pedersen et al. viscosity calculation.
PEDTUNER	Provides multiplicative tuning parameters to modify the mixture molecular weight calculation used in the Pedersen et al. viscosity calculation specified separately for each equation of state region.
SOLUAQA	Specify properties of a water multi-component gas system.
VISCAQA	Specify coefficients of the Ezrokhi's method for aqua viscosity. This keyword is compatible with the CO2STORE, GASWAT, GASSOL and the thermal KWTABTn options.
DENAQA	Specify coefficients of the Ezrokhi's method for aqua density. This keyword is compatible with the CO2STORE, GASWAT, GASSOL and the thermal KWTABTn options.
FILEUNIT	Confirms the units of the data set.
LCUNIT	Allow specification of units for linear combination of phases.
KWTABTn	Tables of aqua Kw-values.
KWTEMP	Temperatures for Kw-value tables.
SPECHW1	Component aqua specific heat, 1st coefficient.
SPECHW2	Component aqua specific heat, 2nd coefficient.
AMFVD	Aqueous mole fractions versus depth for thermal.

REGIONS section

The new REGIONS section keywords are:

Table G.13 REGIONS keywords

Keyword	Description
FILEUNIT	Confirms the units of the data set.

SOLUTION section

The new SOLUTION section keywords are:

Table G.14 SOLUTION section keywords

Keyword	Description
VISDATES	Specify VISAGE geomechanics stress step dates.
VISOPTS	Specify VISAGE geomechanics workflow options.
FILEUNIT	Confirms the units of the data set.

SUMMARY section

The new SUMMARY keywords are:

Table G.15 SUMMARY keywords

Keyword	Description
BTCN	Block Tracer Concentration
CTFR	Well Connection Tracer Flow Rate (+ or -)
CTPR	Well Connection Tracer Production Rate
CTPT	Well Connection Tracer Production Total
CTPC	Well Connection Tracer Production Concentration
CTIR	Well Connection Tracer Injection Rate
CTIT	Well Connection Tracer Injection Total
CTIC	Well Connection Tracer Injection Concentration
CSFR	Well Connection Salt Flow Rate (+ or -)
CSPR	Well Connection Salt Production Rate
CSPT	Well Connection Salt Production Total
CSPC	Well Connection Salt Production Concentration
CSIR	Well Connection Salt Injection Rate
CSIT	Well Connection Salt Injection Total
CSIC	Well Connection Salt Injection Concentration
AAQER	Analytic aquifer thermal energy influx rate
AAQET	Analytic aquifer thermal energy influx total
FAQER	Field analytic aquifer thermal energy influx rate
FAQET	Field analytic aquifer thermal energy influx total
AAQTEMP	Analytic aquifer temperature
AAQENTH	Analytic aquifer molar enthalpy
BIFTWO	Water-oil interfacial tension (alias for BIFTOW and only available for the ODD3P relative permeability and capillary pressure hysteresis model)
BIFTOG	Oil-gas interfacial tension (alias for BIFTGO and only available for the ODD3P relative permeability and capillary pressure hysteresis model)
BIFTWG	Water-gas interfacial tension (alias for BIFTGW and only available for the ODD3P relative permeability and capillary pressure hysteresis model)

Table G.15 SUMMARY keywords

Keyword	Description
BCFR	Inter-block component flow rate
BFLOO	Inter-block oil flow rate
BFLOW	Inter-block water flow rate
BFLOG	Inter-block gas flow rate
BBTFORG	Forchheimer beta factor for gas phase
BBTFORO	Forchheimer beta factor for oil phase
FCGMI	Component molar amount trapped in the gas phase
FCGMM	Component molar amount mobile in the gas phase
FCWM	Component molar amount dissolved in water
FCOM	Component molar amount dissolved in oil
FTLM	Field traced mass liquid total
FTVM	Field traced mass vapor total
FTML	Field traced mass liquid rate
FTMV	Field traced mass vapor rate
GTLM	Group traced mass liquid total
GTVM	Group traced mass vapor total
GTML	Group traced mass liquid rate
GTMV	Group traced mass vapor rate
WTLM	Well traced mass liquid total
WTVM	Well traced mass vapor total
WTML	Well traced mass liquid rate
WTMV	Well traced mass vapor rate
WJPR	Well linearly combined production rate
GJPR	Group linearly combined production rate
FJPR	Field linearly combined production rate
WJPT	Well linearly combined production total
GJPT	Group linearly combined production total
FJPT	Field linearly combined production total
WJPRH	Well linearly combined production rate history
GJPRH	Group linearly combined production rate history
FJPRH	Field linearly combined production rate history
WJPTH	Well linearly combined production total history
GJPTH	Group linearly combined production total history
FJPTH	Field linearly combined production total history
GJPRL	Group linearly combined rate production limit (for reservoir coupling)
CZMF	Connection component mole fractions
WPOFF	Difference between the bottom hole pressure and the highest water saturation pressure in the grid blocks connected to the well
WTOFF	Difference between the water saturation temperature at the bottom hole pressure and the hottest grid block connected to the well

Table G.15 SUMMARY keywords

Keyword	Description
WTSUB	Difference between the water saturation temperature at the bottom hole pressure and the well bore temperature at the bottom hole reference point
FCORn	Field component oil molar rate (n=1 to 99)
GCORn	Group component oil molar rate (n=1 to 99)
WCORn	Well component oil molar rate (n=1 to 99)
FCGRn	Field component gas molar rate (n=1 to 99)
GCGRn	Group component gas molar rate (n=1 to 99)
WCGRn	Well component gas molar rate (n=1 to 99)
FOE	(OIP (initial)-OIP (now))/OIP (initial)
FOEW	Oil production from wells/OIP (initial)
FOEIW	(OIP (initial)-OIP (now))/Initial mobile oil wrt water
FOEWW	Oil production from wells/Initial mobile oil wrt water
FOEIG	(OIP (initial)-OIP (now))/Initial mobile oil wrt gas
FOEWG	Oil production from wells/Initial mobile oil wrt gas
ROE	(OIP (initial)-OIP (now))/OIP (initial)
ROEW	Oil production from wells/OIP (initial)
ROEIW	(OIP (initial)-OIP (now))/Initial mobile oil wrt water
ROEWW	Oil production from wells/Initial mobile oil wrt water
ROEIG	(OIP (initial)-OIP (now))/Initial mobile oil wrt gas
ROEWG	Oil production from wells/Initial mobile oil wrt gas
BAMF	Aqueous mole fractions for aqua components (for blocks)
WAMF	Aqueous mole fractions for aqua components (for wells)
GAMF	Aqueous mole fractions for aqua components (for groups)
FAMF	Aqueous mole fractions for aqua components (for the field)
FAGIP	Field gas in place (aqueous phase) for CO2SOL/H2SSOL/ GASSOL.

SCHEDULE section

The new SCHEDULE section keywords are:

Table G.16 SCHEDULE keyword

Keyword	Description
WSEGSICD	Specifies segments to represent a Spiral Inflow Control Device.
WSEGAICD	Specifies segments to represent an Autonomous Inflow Control Device.
FILEUNIT	Confirms the units of the data set.
WSEGLINK	Defines the chord segment links for specifying looped flowpaths.
MULTFLT	Used to modify transmissibilities across named faults.
LAGTRUPD	Controls when the Lagrangian Tracer Solver updates streamlines.
LTRACE	Forces the Lagrangian Tracer Solver to trace new streamlines.
NOLTRACE	Prevents the Lagrangian Tracer Solver from tracing new streamlines.

Table G.16 SCHEDULE keyword

Keyword	Description
RPTSLN	Controls the output of streamlines.
UDT	Specifies a user defined table.
LINCOM	Specifies the coefficients for a linear combination of oil, water and gas.

Altered keywords in ECLIPSE 100

RUNSPEC section

The altered RUNSPEC keywords are:

Table G.17 Altered RUNSPEC keywords

Keyword	Description
PETOPTS	The INITNNC mnemonic has been extended and now also applies to CORSNUM information. If a grid file is also being output, INITNNC adds the requested information to the INIT file but does not suppress its output to the grid file. The NOPINCHC mnemonic has been added to suppress the calculation of pinchout NNCs. The TRANPORV mnemonic has been added to tell ECLIPSE 100 that the .GSG file contains transmissibilities and pore volumes. A new NEWTRAN mnemonic has been added for compatibility with ECLIPSE 300. It has no effect.
START	A fourth argument has been added for the time at the start of the run.
BRINE	Activates the new Multi-Component Brine model if used in conjunction with keyword ECLMC.
NMATRIX	As triple porosity is now usable in ECLIPSE 100, this keyword will conflict with the triple porosity option and cannot be used in the same deck.
PSTEADY	A twelfth argument has been added which can be used to speed-up the simulator's advance to steady-state.
WSEGSOLV	The default value of item 3 has been changed from 1.0E-6 to 1.0E-8.
ROCKCOMP	The BOBERG rock compaction hysteresis can be activated using the ROCKCOMP keyword.

GRID section

The altered GRID keywords are:

Table G.18 Altered GRID keywords

Keyword	Description
PETGRID	This keyword has been extended to import local grids for corner point geometry with curved or truncated pillars. In addition, if this keyword is present in the dataset, no grid file will be output.
RPTGRID	New mnemonic IONROCK has been added to output the rock ion exchange capacity.

PROPS section

The altered PROPS keywords are:

Table G.19 Altered PROPS keywords

Keyword	Description
PLYROCK	The first argument can now be set to a value between 0 and 1, but strictly less than 1.
ROCKOPTS	The ELASTIC option can be activated using the ROCKOPTS keyword to determine the initial conditions for the HYSTER and BOBERG options.
RPTPROPS	If output of the PVTO tables is selected, the interpolated versions of the tables will also be printed if API tracking has been specified

SOLUTION section

The altered SOLUTION keywords are:

Table G.20 Altered SOLUTION keywords

Keyword	Description
SALT	Specifies initial salt concentration for multiple salts when the Multi-Component Brine model is active.
SALTVD	Specifies tables of salt concentration versus depth for multiple salts when the Multi-Component Brine model is active.
RPTSOL	Mnemonics ANIONS and CATIONS have been added to output initial anion and divalent cation concentrations.
RPTSOL	Mnemonics ESALSUR and ESALPLY have been added to output initial surfactant and polymer effective salinities.
RPTSOL	Mnemonic SALTVD now outputs tables of initial anion and divalent cation equivalent molar concentrations versus depth when the Multi-Component Brine model is active.

Table G.20 Altered SOLUTION keywords

Keyword	Description
RPTRST	Mnemonic CONV outputs indicators for worst converging cells.
RPTRST	Mnemonic PORV outputs the reservoir pore volumes at reference conditions.
RPTRST	Mnemonic VISC allows to output extra viscosities when Polymer or Surfactant or Brine option(s) is/are active: - SHWATVIS: effective polymer viscosity due to shear thinning (PLYSHEAR) - EPVIS: effective polymer viscosity (Polymer option) - EMVIS: effective mixture (water/polymer) viscosity (Polymer option) - EWV_POL: effective water viscosity (Polymer option) - EWV_SUR: effective water viscosity due to surfactant concentration (Surfactant option) - EWV_SAL: effective water viscosity due to salt concentration (Brine option)

SCHEDULE section

The altered SCHEDULE keywords are:

Table G.21 Altered SCHEDULE keywords

Keyword	Description
DATES	A fourth argument has been added to this keyword for the time of the report.
NEXTSTPL	A new item has been added to the keyword NEXTSTPL so that the maximum length of the next time-step can now be applied to every subsequent report step.
GCONSALE	Items 2, 3 and 4 of GCONSALE have been modified to allow UDAs.
RPTSCHED	New mnemonic ALKALINE to output Alkaline concentrations and adsorption (if activated).
WELDRAW	Item 2 of WELDRAW has been modified to allow UDAs.
WLIFTOPT	New item 6 to permit the inclusion of a weighted gas rate production term in the denominator of gas lift optimization gradient calculation. New item 7 to permit wells to continue to be allocated lift gas after a group gas target has been achieved but whilst an oil rate limit has not been reached.
WSEGSICD	New items 11 and 12 to specify flow scaling calculation method and maximum valid flow rate.
GRDREACH	New item 16 has been added to remove the capillary pressure term from the river inflow equation.
GRDREACH	Items 17, 18 and 19 have been added to specify the choice of river infiltration equation.
WSALT	The keyword now specifies the injection concentration of multiple salts when the Multi-Component Brine model is active.
RPTSCHED	New mnemonics ANIONS and CATIONS have been added to output anion and divalent cation concentrations.
RPTSCHED	New mnemonics ESALSUR and ESALPLY have been added to output surfactant and polymer effective salinities.
TUNING	Item 12 has been added to the second record specifying a relative threshold parameter for damping in the ion exchange calculations.

Table G.21 Altered SCHEDULE keywords

Keyword	Description
RPTRST	Mnemonic CONV outputs indicators for the worst converging cells.
WCONPROD	New CRAT control mode and new item to specify associated target/limit.
GCONPROD	New CRAT control mode and new item to specify associated target/limit.
WCONHIST	New CRAT control mode and new item to specify associated target/limit.
WHISTCTL	New CRAT control mode.
WELCNTL	New CRAT control mode.
WELTARG	New CRAT control mode.
WLISTARG	New CRAT control mode.
WTADD	New CRAT control mode.
WTMULT	New CRAT control mode.
GTADD	New CRAT control mode.
GTMULT	New CRAT control mode.
GRUPTARG	New CRAT control mode.
WCUTBACK	New linearly combined control phase.
WCUTBACT	New linearly combined control phase.
GCUTBACK	New linearly combined control phase.
GCUTBACT	New linearly combined control phase.
WGRUPCON	New linearly combined control phase.
GUIDERAT	New linearly combined control phase.
WBHGLR	New linearly combined control phase.
GCONPRI	New item to specify linearly combined target/limit and new item to specify procedure on exceeding this limit.
WAPI	Item 2 of WAPI has been modified in to allow UDAs.
WFOAM	Item 2 of WFOAM has been modified to allow UDAs.
WPOLYMER	Items 2 and 3 of WPOLYMER has been modified to allow UDAs.
WSALT	Item 2 of WSALT has been modified in to allow UDAs.
WSOLVENT	Item 2 of WSOLVENT has been modified in to allow UDAs.
WSURFACT	Item 2 of WSURFACT has been modified to allow UDAs.
WTRACER	Items 3 and 4 of WTRACER has been modified to allow UDAs.
WTMULT	Item 3 of WTMULT has been modified to allow UDAs.
WTADD	Item 3 of WTADD has been modified to allow UDAs.
TUNING	Item 13 has been added to the second record specifying a weighting factor for updating active tracer concentrations when the tracers are solved in each Newton iteration.
RPTSCHED	Mnemonic PORV outputs the reservoir pore volumes at reference conditions.
UDQ	Additional syntax has been added to allow the use of user defined tables (UDT).

OPTIONS keyword

The altered OPTIONS are:

Table G.22 Altered OPTIONS

Switch	Description
Switch 88	The default behavior is for the emulsion viscosity model to be activated in keyword WSEGSICD and deactivated in keyword WSEGAICD. If this switch is set equal to 1 the emulsion viscosity model is deactivated in keyword WSEGSICD. If set equal to 2 the emulsion viscosity model is activated in keyword WSEGAICD. If set equal to 3 the emulsion viscosity model is both deactivated in keyword WSEGSICD and activated in keyword WSEGAICD.
Switch 96	From 2009.1 onwards, the derivative of the pore volume with respect to pressure in the Palmer-Mansoori rock model (see the ROCKPAMA keyword) has been modified for stability. This can help if you encounter convergence problems running the Palmer-Mansoori model.
Switch 101	A bug was fixed in the calculation of well potentials when the generalized pseudo-pressure special inflow equation is used (see WELSPECS item 8). The pre-2009.1 behavior can be restored by setting this item to a non-zero multiple of 7.
Switch 117	In dual porosity, this allows MULTNUM regions to be specified for only the matrix or the fracture if this switch is greater than 0; without this switch, matrix specifications will automatically be copied to the fracture and vice versa.Additionally, if the switch is greater than 1, this behavior will also apply to the PINCHNUM, FLUXNUM and OPERNUM keywords.
Switch 118	If set to a non-zero value, the pre-2008.2 calculation behavior for the use of the DRSDT and DRVDT keywords in the presence of both vaporized oil and dissolved gas is restored.
Switch 119	If set to a non-zero value, the pre-2009.1 capillary pressure calculations for the surfactant model are restored.
Switch 120	If set to 1, the pre-2008.2 convergence criterion for the standard solver used for tracer tracking is restored.
Switch 121	If set to a non-zero value, the pre-2008.2 calculation behavior for the calculation of relative permeabilities using Stone's Second Model with end-point scaling is restored.
Switch 122	If set to 1, the pre-2008.2 Polymer Dead Pore Volume model is restored.
Switch 123	If set to a non-zero value, the pre-2009.1 behavior for the interpolation of the Carter-Tracy aquifer influence function tables is restored.
Switch 124	If set to a non-zero value, the pre-2009.1 calculation of the summary quantities FTPC, GTPC, WTPC, CTPC, FTIC, GTIC, WTIC and CTIC for concentrations of partitioned tracers will be restored.
Switch 125	If set to a positive value, this changes the tolerance used when checking whether the current simulation time is a report time. The new tolerance is 10**(-OPTIONS(125)). The default tolerance is 10**(-3) which is equivalent to setting this value to 3.

Table G.22 Altered OPTIONS

Switch	Description
Switch 126	If set to 1, all active tracers will be solved in each Newton iteration instead of at the end of a time step. This option can be used to reduce the time lag in tracer dependent properties. If set to 2, all active tracers will be solved at the end of a time step. The default behavior is to solve all tracers at the end of a time step, unless `ECLMC` is specified in the `RUNSPEC` section, in which case the default behavior is to solve Brine, Surfactant and Alkaline tracers in each Newton iteration, but all other tracers at the end of the time step.
Switch 127	If set to a positive value, the pre-2009.1 behavior for tracer adsorption, using a non-cascade tracer solver, is restored. Prior to 2009.1, the adsorped tracer amount was not properly reset following a convergence failure.
Switch 128	If set to a non-zero value, the pre-2009.1 behavior for the calculation of residual oil saturation for the Stone I relative permeability model with end-point scaling is restored.
Switch 129	For developer use.
Switch 130	If set to a non-zero value, the pre-2009.1 behavior for specific heat capacity conversion is restored.
Switch 131	Except for the `MISCNUM` keyword, the documentation states that all `REGIONS` section keywords should be assigned a minimum value of 1. However, it is possible that pre-processor or manual edits to a keyword definition, can set a value of 0, even for inactive cells. To be compatible with the treatment of `REGIONS` section keywords using `EQUALS/ADD/MULTIPLY` keywords, if `REGIONS` data is specified using a `REGIONS` keyword directly, then error messages can be output for both active and inactive cells. To do this, set `OPTIONS` switch number 131 to a non-zero value.
Switch 132	If set to a non-zero value, the pre-2009.1 behavior for `WLIFT` is restored.
Switch 133	If set to a non-zero value, well and group guide rates are allowed to change for the first `NUPCOL` iterations instead of remaining fixed throughout the timestep.
Switch 134	If set to 1, a different algorithm is used to calculate the local grid geometry. In some cases this can give better output to the EGRID file for visualization purposes.
Switch 135	If set to a positive number, the pre-2009.1 interpolation method for the `SOF32D` saturation functions family is restored.
Switch 136	If set to a non-zero value, the pre-2009.1 behavior for the decay of an adsorped tracer is restored.
Switch 137	If set to a non-zero value, the pre-2009.1 behavior for inter-block velocity calculations is restored.
Switch 138	If set to a non-zero value, the pre-2009.1 behavior for the calculation of the modification to the curvature of the relative permeability curves using the keyword `DPKRMOD` is restored.
Switch 139	If set to a multiple of 2, the switch activates an alternative method for calculating the mixture density for stagnant connections at the bottom of a well. If set to a multiple of 3, the switch forces cross-flowing connections to use the segregated density calculation method.

Table G.22 Altered OPTIONS

Switch	Description
Switch 140	If set to a non-zero value, the pre-2009.2 behavior for the interpolation of PVTO tables for API tracking is restored. For the 2009.2 version onwards the interpolation procedure has been modified to avoid duplicate Rs nodes which could occur for some combinations of table data and minimum Rs node spacing.
Switch 141	If set to a non-zero value, the pre-2009.2 behavior for adjusting non-neighbor connections within coarsened cells is restored. For 2009.2 onwards, the NNC ends may be moved to a different coarsened cell to avoid the situation where, after coarsening, a fault NNC no longer crosses its fault. This new algorithm may also cause slight changes in the transmissibility of other NNCs.
Switch 144	If set to a non-zero value, the pre-2009.2 behavior for calculating flow rate limits for the WELDRAW keyword is used.

Altered keywords in ECLIPSE 300

RUNSPEC section

The altered RUNSPEC keywords are:

Table G.23 Altered RUNSPEC keywords in ECLIPSE 300

Keyword	Description
PETOPTS	The INITNNC mnemonic has been extended and now also applies to CORSNUM information. If a grid file is also being output, INITNNC adds the requested information to the INIT file but does not suppress its output to the grid file. The NOPINCHC mnemonic has been added to suppress the calculation of pinchout NNCs. The TRANPORV mnemonic has been added to tell ECLIPSE 300 that the .GSG file contains transmissibilities and pore volumes. A new NEWTRAN mnemonic has been added. This forces fault transmissibilities to be calculated even for corner point geometry, which is necessary for correct treatment of coarsened cells containing faults.
START	A fourth argument has been added for the time at the start of the run.
WSEGDIMS	A fourth item has been added to specify the maximum number of chord segment links per well.
NSTACK	The default value for the first argument has been modified to 20 when using the JALS solver.
JALS	The JALS solver has a new mode that can be selected via the 1st argument of the keyword. In addition, a new item 4 has been added in order to switch on an extra residual check (“ON”). By default, this item will be set to “OFF”.

GRID section

The altered GRID keywords are:

Table G.24 Altered GRID keywords in ECLIPSE 300

Keyword	Description
PETGRID	This keyword has been extended to import local grids for corner point geometry with curved or truncated pillars. In addition, if this keyword is present in the dataset, no grid file will be output.
AQUCT	A new item has been added to specify the temperature in the aquifer. This is only for use with either the thermal or temperature options.
RPTGRID	New mnemonics THCROCK, THCOIL, THCWATER, THCGAS and THCSOLID have been added to output respectively the rock, oil phase, water phase, gas phase and solid phase thermal conductivity.
RPTGRIDL	New mnemonics THCROCK, THCOIL, THCWATER, THCGAS and THCSOLID have been added to output respectively the rock, oil phase, water phase, gas phase and solid phase thermal conductivity.
INIT	The .INIT file may now contain THCROCK, THCOIL, THCWATER, THCGAS and THCSOLID data.
ROCKPROP	Item 7 has been added to specify a minimum temperature difference to start the Vinsome model heat loss calculation.
MULTREGT	The keyword is now functional in parallel.
COPYBOX	The keyword is now functional in parallel.

PROPS section

The altered PROPS keywords are:

Table G.25 Altered PROPS keywords in ECLIPSE 300

Keyword	Description
AQUCT	A new item has been added to specify the temperature in the aquifer. This is only for use with either the thermal or temperature options.
ODD3P	The three-phase relative permeability and capillary pressure hysteresis model calculation has been improved. Pre-2009.2 behavior may be restored using the OPTODD3P keyword.
RSWVD	New items can be added to specify the concentration of dissolved gases versus depth. This keyword is compatible with the GASSOL keyword.
DIFFCWAT	This existing keyword is used to specify the aqua component diffusion coefficients, and is now compatible with the GASSOL option as well as GASWAT and CO2STORE.
PCODD3P	Item 1 has been extended to permit a new capillary pressure model which incorporates saturation weighting based upon three phases via option 12.
EQLDREAC	A 6th argument has been added to enable reactions to be limited to forward or reverse directions only.
RPTPROPS	Mnemonics SWF3 and SGF3 now output, in addition to the input saturation function tables, the 3-phase relative permeabilities for water and gas, respectively, for the IKU3P model.

SOLUTION section

The altered SOLUTION keywords are:

Table G.26 Altered SOLUTION keywords in ECLIPSE 300

Keyword	Description
RPTSOL	A new mnemonic ROCKCON has been added to output rock connection data within global grid and local grid refinements
AMF	This keyword is compatible with the GASWAT and CO2STORE keywords.
AQUCHGAS	A new item has been added to specify the temperature in the aquifer. This is only for use with either the thermal or temperature options.
AQUCHWAT	A new item has been added to specify the temperature in the aquifer. This is only for use with either the thermal or temperature options.
AQUCT	A new item has been added to specify the temperature in the aquifer. This is only for use with either the thermal or temperature options.
AQUFETP	A new item has been added to specify the temperature in the aquifer. This is only for use with either the thermal or temperature options.
AQUFLUX	New items have been added to specify the temperature and pressure in the aquifer. This is only for use with either the thermal or temperature options.
RSWVD	New items can be added to specify the concentration of dissolved gases versus depth. This keyword is compatible with the GASSOL keyword.
RSW	New items can be added to specify the initial variation of aqueous gases-water ratio by enumeration. This keyword is compatible with the GASSOL keyword.
RPTRST	If flow output is requested, this now also includes NNC flows.
RPTSOL	In previous versions, LAMB was used to output the saturation dependent multiplier for thermal conductivity for the Thermal option. This has now been replaced by TCMULT.
RPTRST	In previous versions, LAMB was used to output the saturation dependent multiplier for thermal conductivity for the Thermal option. This has now been replaced by TCMULT.
RPTRST	A new mnemonic TCBULK has been added to output the cell average thermal conductivities multiplied by the saturation dependent multiplier.
RPTSOL	Two new mnemonics BTFORG and BTFORO have been added to output Forchheimer beta factors.
RPTRST, RPTSOL	The mnemonics FLOOIL and FLOGAS for the RPTRST and RPTSOL keywords can now be used to output inter-block oil and gas flow rates for compositional models. Note that, in such (compositional) cases, the values returned for FLOOIL and FLOGAS are, by default, based on the first FIP set defined. Also note that FLOWAT is now compatible with the GASWAT and CO2STORE options.
RPTRST	Two new mnemonics BTFORG and BTFORO have been added to output Forchheimer beta factors.
RPTRST	PORV outputs the reservoir pore volumes at reference conditions.
RPTSOL	PORV outputs the reservoir pore volumes at reference conditions.
RPTRST	TSUB outputs sub-cool temperatures in a THERMAL simulation.

SUMMARY section

The altered SUMMARY mnemonics are:

Table G.27 Altered SUMMARY mnemonics

Mnemonic	Description
BLAMB	In previous versions, BLAMB was used to output the block saturation dependent multiplier for thermal conductivity for the Thermal option. This has now been replaced by BTCMULT.
WXMFn	The number of separation stages n have been extended to cover up to 99 stages.
WYMFn	The number of separation stages n have been extended to cover up to 99 stages.
GXMFn	The number of separation stages n have been extended to cover up to 99 stages.
GYMFn	The number of separation stages n have been extended to cover up to 99 stages.
FXMFn	The number of separation stages n have been extended to cover up to 99 stages.
FYMFn	The number of separation stages n have been extended to cover up to 99 stages.
WOnPR	The number of separation stages n have been extended to cover up to 99 stages.
GOnPR	The number of separation stages n have been extended to cover up to 99 stages.
FOnPR	The number of separation stages n have been extended to cover up to 99 stages.
WOnPT	The number of separation stages n have been extended to cover up to 99 stages.
GOnPT	The number of separation stages n have been extended to cover up to 99 stages.
FOnPT	The number of separation stages n have been extended to cover up to 99 stages.
WGnPR	The number of separation stages n have been extended to cover up to 99 stages.
GGnPR	The number of separation stages n have been extended to cover up to 99 stages.
FGnPR	The number of separation stages n have been extended to cover up to 99 stages.
WGnPT	The number of separation stages n have been extended to cover up to 99 stages.
GGnPT	The number of separation stages n have been extended to cover up to 99 stages.
FGnPT	The number of separation stages n have been extended to cover up to 99 stages.

SCHEDULE section

The altered SCHEDULE keywords are:

Table G.28 Altered SCHEDULE keywords in ECLIPSE 300

Keyword	Description
AQUCHGAS	A new item has been added to specify the temperature in the aquifer. This is only for use with either the thermal or temperature options.
AQUCHWAT	A new item has been added to specify the temperature in the aquifer. This is only for use with either the thermal or temperature options.
AQUCT	A new item has been added to specify the temperature in the aquifer. This is only for use with either the thermal or temperature options.
AQUFETP	A new item has been added to specify the temperature in the aquifer. This is only for use with either the thermal or temperature options.
AQUFLUX	New items have been added to specify the temperature and pressure in the aquifer. This is only for use with either the thermal or temperature options.

Table G.28 Altered SCHEDULE keywords in ECLIPSE 300

Keyword	Description
CVCRIT	A value less than NSTACK can now be specified if desired
DATES	A fourth argument has been added to this keyword for the time of the report.
GRUPFUEL	Items 2 and 3 of GRUPFUEL have been modified to allow UDAs.
GRUPSALE	Items 2 and 3 of GRUPSALE have been modified to allow UDAs.
RPTRST	If flow output is requested, this now also includes NNC flows.
WELDRAW	Item 2 of WELDRAW has been modified to allow UDAs.
WLIFTOPT	New item 6 to permit the inclusion of a weighted gas rate production term in the denominator of the gas lift optimization gradient calculation. New item 7 to permit wells to continue to be allocated lift gas after a group gas target has been achieved but whilst an oil rate limit has not been reached.
RPTSCHED	In previous versions, LAMB was used to output the saturation dependent multiplier for thermal conductivity for the Thermal option. This has now been replaced by TCMULT.
RPTRST	In previous versions, LAMB was used to output the saturation dependent multiplier for thermal conductivity for the Thermal option. This has now been replaced by TCMULT.
RPTRST	A new mnemonic TCBULK has been added to output the cell average thermal conductivities multiplied by the saturation dependent multiplier.
RPTSCHED	Two new mnemonics BTFORG and BTFORO have been added to output Forchheimer beta factors.
RPTRST	Two new mnemonics BTFORG and BTFORO have been added to output Forchheimer beta factors.
WELLSTRW	Now activated for options CO2STORE, GASWAT and GASSOL.
WINJW	Now activated for options CO2STORE, GASWAT and GASSOL.
WSALT	Item 2 of WSALT has been modified to allow UDAs.
WTRACER	Items 3 and 4 of WTRACER have been modified to allow UDAs.
WTMULT	Item 3 of WTMULT has been modified to allow UDAs.
WTADD	Item 3 of WTADD has been modified to allow UDAs.
RPTRST, RPTSCHED	The mnemonics FLOOIL and FLOGAS for the RPTRST and RPTSCHED keywords can now be used to output inter-block oil and gas flow rates for compositional models. Note that, in such (compositional) cases, the values returned for FLOOIL and FLOGAS are, by default, based on the first FIP set defined. Also note that FLOWAT is now compatible with the GASWAT and CO2STORE options.
ACTIONW	The mnemonics WXMFnn, WYMFnn and WZMFnn should now be written WXMF_nn, WYMF_nn and WZMF_nn. Item 181 of keyword OPTIONS3 reverts this change.
UDQ	Additional syntax has been added to allow the use of user defined tables (UDT).
RPTSCHED	Mnemonic PORV outputs the reservoir pore volumes at reference conditions.
WCONPROD	New CRAT control mode and new item to specify associated target/limit.
GCONPROD	New CRAT control mode and new item to specify associated target/limit.
WCONHIST	New CRAT control mode and new item to specify associated target/limit.
WELLPROD	New CRAT control mode and new item to specify associated target/limit.
GRUPPROD	New CRAT control mode and new item to specify associated target/limit.

Table G.28 Altered SCHEDULE keywords in ECLIPSE 300

Keyword	Description
WHISTCTL	New CRAT control mode.
WELCNTL	New CRAT control mode.
WELTARG	New CRAT control mode.
WLISTARG	New CRAT control mode.
WTADD	New CRAT control mode.
WTMULT	New CRAT control mode.
GTADD	New CRAT control mode.
GTMULT	New CRAT control mode.
GRUPTARG	New CRAT control mode.
WCUTBACK	New linearly combined control phase.
WGRUPCON	New linearly combined control phase.
GUIDERAT	New linearly combined control phase.
WELLGR	New linearly combined control phase.
GRUPGR	New linearly combined control phase.
GCONPRI	New item to specify linearly combined target/limit and new item to specify procedure on exceeding this limit.
RPTRST, RPTSCHED	TSUB outputs sub-cool temperatures in a THERMAL simulation.
CVCRIT	The default maximum number of linear iterations when using the JALS solver has been reduced to 20.
TUNING	The default maximum number of linear iterations when using the JALS solver has been reduced to 20.

OPTIMIZE section

The altered OPTIMIZE keywords are:

Table G.29 Altered OPTIMIZE keywords in ECLIPSE 300

Keyword	Description
OPTPARS	A new item 'ORIG' is available when specifying the SCSA parameter control mode in item 7. The default for this item has been changed from 'BHP' to 'ORIG'.

OPTIONS3 keyword

The altered OPTIONS3 switches are:

Table G.30 Altered OPTIONS3 switches

Switch	Description
9	If set to a non-zero value, a zero THP will be assigned to zero rate wells. If set to -1 the pre-2009.1 behavior for this option switch will be obtained.
95	This switch controls the scale back of sales and fuel gas should insufficient gas be available to meet sales, fuel and reinjection targets. If set to 0 fuel and sales gas are scaled back equally only if WAVAILIM is set. If set to 3 fuel and sales gas are scaled back equally irrespective of WAVAILIM. If set to 4 the priority order specified in WTAKEGAS is used to decide which of the fuel and sales to scale back first, but no scale back takes place if WAVAILIM has not been specified.
137	This switch has been extended so that, if it is set to a negative value, an extra flash is not performed for property updates in TRACER runs (for pre-2008.1 back compatibility).
142	If set to 1, the pre-2008.1 behavior of WELSPECS element 5 is used. If set to 2 then the 2008.1 behavior is used.
158	If set to 1, the Appleyard chop is applied to components when they appear or disappear. This behavior is the default for simulations using the chemical reactions feature but can be disabled by setting to -1.
159	If set to a non-zero value, the pre-2008.2 behavior for gas accounting is restored according to the value set. If the value is divisible by 2 and if a well is set to reinject a group vapor source or a group is set to reinject a well vapor source (for example GINJGAS item 2 set to WV), then no gas will be reinjected. If the value is divisible by 3, then make-up gas is not switched off if a reinjection target is set for a group. If the value is divisible by 5, then the pre-2008.2 calculation of advance gas is used. This changes the calculation of advance gas if the WEFAC and/or GEFAC keywords are used. If the value is divisible by 7, then the pre-2008.2 calculation of availability limit is used. This changes the calculation of the availability limit if the WEFAC and/or GEFAC keywords are used. If the value is divisible by 11, then the pre-2008.2 calculation of gas accounting is used. This changes the calculation of excess gas and reinjected gas if the GEFAC keyword is used.
160	If set to 1, the pre-2008.2 thermal non-linear damping is restored.
161	If set to 1 an alternative procedure for simulations with rock-filled cells is activated. Mathematically this is a null change but variations due to numerical rounding may occur.
162	If set to 1, this restores the pre-2008.1 behavior for equilibration options 2 and 3 when the reference depth is not set to GOC.
163	If set to 1, an alternative memory allocation algorithm is enabled. It reduces the number of allocations/de-allocations while increasing the maximum amount of memory used.
164	If set > 0, this switch turns on an alternative viscosity calculation in CO2SOL.

Table G.30 Altered OPTIONS3 switches

Switch	Description
165	If set to 1, this switch turns on Ezrokhi's formula for the water density calculation in CO2SOL/H2SSOL.
166	If set to 1, well constraints may be weakly violated in order to improve the convergence of the well model.
167	If set to > 0, this switch activates the new single-phase splitting initialization algorithm for Gibbs free energy minimization in the flash.
168	If set to 1, the pre-2008.2 calculation of fluid in place is used, correctly accounting for satellite group injection and production fluids.
169	The default behavior is for the emulsion viscosity model to be activated in keyword WSEGSICD and deactivated in keyword WSEGAICD. If this switch is set equal to 1 the emulsion viscosity model is deactivated in keyword WSEGSICD. If set equal to 2 the emulsion viscosity model is activated in keyword WSEGAICD. If set equal to 3 the emulsion viscosity model is both deactivated in keyword WSEGSICD and activated in keyword WSEGAICD.
170	If set to a non-zero value, the pre-2009.1 behavior for the interpolation of the Carter-Tracy aquifer influence function tables is restored.
171	If set to 1, the pre-2009.1 multi segment well energy predictor is restored.
172	If set to a non-zero value, the pre-2009.1 equilibration FACTLI calculation is performed.
173	If set to a non-zero value, the pre-2009.1 behavior for gas accounting is restored. This affects how sales and fuel gas is calculated if a reinjection target greater than 1 is specified.
174	If set to a non-zero value, the pre-2009.1 FACTLI calculation during solution initialization is used. The new calculation updates FACTLI based only on the single-phase cells.
175	If set to a non-zero value, the pre-2006.2 defaulting of the D-factor (item 12 of the COMPDAT keyword, or item 13 of the COMPDATL and COMPDATM keywords) is restored. A bug was fixed in this defaulting at 2006.2. Prior to that version, the completion D-factor was re-computed each time one of these keywords was present with the D-factor defaulted.
176	This option must be set in the RUNSPEC section. If set to a non-zero value the pre-2009.1 method for calculating region phase production totals (ROPT, RWPT and RGPT) is used. Pre-2009.1 these quantities were reported for the beginning of the timestep rather than the end.
177	If a multiple of 2, then the closure of history matched wells with zero production rates and banned crossflow will not automatically occur. If a multiple of 3, then the closure of injection wells with zero injection rates and banned crossflow will not automatically occur.
178	If set to a non-zero value, the pre-2008.3 reporting of group potentials is restored. Prior to the 2008.3 version, shut and stopped well potentials were sometimes included in the reported group potential.
179	From 2009.1 onwards, the derivative of the pore volume with respect to pressure in the Palmer-Mansoori rock model (see the ROCKPAMA keyword) is modified for stability. This helps with some convergence problems encountered in previous releases. The default behavior is to ignore the temperature effect if it dominates the pressure effect. If a value of 1 is used, the pre-2009.1 behavior for Palmer-Mansoori rock model is restored. If a value of 2 is used then the temperature effect in the derivative is ignored in all cases.

Table G.30 Altered OPTIONS3 switches

Switch	Description
180	If set to a nonzero value, the pre-2009.1 behavior of allowing oil injectors is restored.
181	If set to a non-zero value, the mnemonics `WXMF`, `WYMF` and `WZMF` of component nn in keyword `ACTIONW` are handled in the pre-2007.1 way: they should be written `WXMFnn`, `WYMFnn` and `WZMFnn`. The post-2007.1 way of writing these mnemonics is `WXMF_nn`, `WYMF_nn` and `WZMF_nn`.
182	If set to 1, this option ensures that the pre-2009.1 calculations for `FLOOIL` and `FLOGAS` are used for black-oil models.
183	If set to 1, this option ensures that the pre-2009.1 calculations for `FLOWAT` and `RWFR` are used for multi-component water models.
184	If set to 1, this option ensures that the pre-2009.1 calculation of the well model for `CO2SOL`/`H2SSOL` case is used.
185	If set to a non-zero value, this option restores the 2008.1 behavior of equilibration option 3 when the reference depth is not set to GOC. Note that the pre-2008.1 behavior can be restored by setting item 162 of `OPTIONS3` to 1.
186	This affects various behavior related to the `WAVAILIM` keyword. • From 2009.1 onwards, if a well is under availability control when an unlimited injection fluid source (for example stream) is specified (see `WINJGAS`), then the control mode is reset to the most recent one specified explicitly for that well. The pre-2009.1 behavior (which leaves the well on availability control) can be restored by setting `OPTIONS3` item 186 to a non-zero value which is a multiple of 2. • From 2009.1 onwards, if a group is under availability control (see `WAVAILIM`) when more than `NUPCOL` Newton iterations are required, the appropriate phase availability is restored for the group from the last `NUPCOL` calculated value. Previously the group phase availability limit would always have been the last `NUPCOL` calculated availability for the **water** phase. The pre-2009.1 behavior can be restored by setting `OPTIONS3` item 186 to a non-zero multiple of 3.
187	If set to a non-zero value, the pre-2009.1 reporting of segment phase velocities is restored. Prior to 2009.1 the phase velocities in segments where the drift flux slip model was not used were set equal to the segment mixture velocity. Post-2009.1 the phase velocity is set to zero if the phase saturation within the segment is zero.
188	If set to a non-zero value, the pre-2009.1 calculation of temperature variation with depth in the flash calculation is restored.
189	If set to 1, this switch activates a rigorous density inversion check post-scalar flash If set to 2, this switch activates a rigorous density inversion check during the vector flash as well as post-scalar flash.
190	If set to a non-zero value, the pre-2009.1setting of well enthalpies in thermal decks is activated. From 2009.1 onwards the well enthalpies are set whenever an action is triggered.
191	If set to a non-zero value, the pre-2009.1 density gravity calculation method for MSWs is used.

Table G.30 Altered OPTIONS3 switches

Switch	Description
192	The calculation of tracer concentration has been changed if a well or group efficiency factor has been set (`WEFAC` or `GEFAC`). The pre-2009.1 behavior can be restored by setting `OPTIONS3`, item 192 to a non-zero value.
193	If set to a non-zero value, the pre-2009.2 behavior is restored for the calculation of the minimum oil saturation for the Stone 1 relative permeability model with end-point scaling and with either the `SOMGAS` or the `SOMWAT` keyword specified. This switch is only relevant if the critical oil-in-gas (`SOGCR`) or critical oil-water (`SOWCR`) end-point saturations are zero.
194	If set to a positive value, pre-2009.2 `LAGTRAC` calculations are recovered: • If divisible by 2, streamline update and retrace times are calculated as they were for 2009.1. • If divisible by 3, streamline tracing uses the 2009.1 cell porosity calculation. • If divisible by 5, streamline tracing ignores rock compressibility. • If divisible by 7, streamline flowrates are calculated using the 2009.1 well/group efficiency method.
196	If set to a non-zero value, this restores the pre-2009.2 behavior for the well flow-dependent skin calculation for `VELDEP` models using radial geometry.
197	If set to a non-zero value, the pre-2009.2 behavior for adjusting non-neighbor connections within coarsened cells is restored. For 2009.2 onwards, the NNC ends may be moved to a different coarsened cell to avoid the situation where, after coarsening, a fault NNC no longer crosses its fault. This new algorithm may also cause slight changes in the transmissibility of other NNCs.
204	If set to a non-zero value, this enables the pre-2009.2 calculation method for segments whose pressure drop is defined using VFP tables.
205	If set to a positive value, this enables the pre-2009.2 behavior of `ACTIONX` with `WDYNLIST`.
206	If set to a positive value, two successive Newton iterations are required to converge in order to achieve final convergence.
207	Alternative Jacobian construction for `CO2SOL`/`H2SSOL`/`GASSOL`
208	If set to a non-zero value, pre 2009.2 behavior is restored whereby the `SPECHS` keyword may be omitted for thermal models with one or more components exhibiting solid phases.
210	If set to a non-zero value, the pre-2009.2 behavior of the well inflow performance relationship calculations is restored. This mainly affects Open ECLIPSE queries ending `IPRA`, `IPRB` or `IPRC` for wells with injecting connections.

Table G.30 Altered OPTIONS3 switches

Switch	Description
211	If set to a non-zero value, the pre-2009.2 behavior of gas lift for wells drilled using the `QDRILL` keyword is used. If lift gas is specified for a closed well using `WCONPROD` prior to 2009.2 then no lift gas will be used if the well is opened using `QDRILL`.
212	If set to a positive value the old behavior when calculating default rock volumes for a coal bed methane, instant adsorption model is activated.
214	If set to a non-zero value, this enables the pre-2009.2 John Appleyard Linear Solver (`JALS`).

New developments in 2008.1 and 2008.2

New facilities in ECLIPSE 100

RUNSPEC section

- Dimensions for User Defined Arguments can be specified in `UDADIMS`.
- Dimensions for aquifer and block user defined quantities can be specified in `UDQDIMS`.
- The `WPOTCALC` keyword has been added to control potential calculations on shut and stopped wells. The use of keyword is equivalent to setting `OPTIONS` item 101 to a non-zero multiple of 3 and has been added to make the control of these potential calculations more visible.
- The `WSEGSOLV` keyword has been added to control parameters related to the multi-segmented well iterative linear solver.
- The `DYNRDIMS` keyword has been added to set up dimensions for the Dynamic Region facility.
- The `GRAVDRB` keyword can be used to apply the vertical discretised gravity drainage model.
- The `NNEWTF` keyword can be used to activate the non-Newtonian model when the Polymer flood option is active.
- An item has been added to the `WSEGDIMS` keyword to specify the maximum number of chord segment links per well
- The `PETOPTS` keyword has a new mnemonic `NOPINCHC` which suppresses calculation of pinchout NNCs.
- The `START` keyword has a new argument for the time at the start of the run.
- The `VISAGE` keyword has been added to activate the VISAGE external geomechanics workflow.

GRID section

- The `JFUNCR` keyword can be used to specify Leverett J-Functions for the scaling of capillary pressure on a per saturation table basis, with the option of selecting capillary pressures from either direct table look-up or via J-Function calculations.These can be output with the `RPTGRID` keyword.
- The `PETGRID` keyword can now be used to import global grid geometry, local grid geometry and properties from a Generic Simulation Grid (.GSG) file. This type of file format has to be imported via Petrel.
- The `EQUALREG` keyword has been extended to allow the user to choose the region (`MULTNUM`, `FLUXNUM` or `OPERNUM`) which will be used to modify a specific property.
- The `ADDREG` keyword has been extended to allow the user to choose the region (`MULTNUM`, `FLUXNUM` or `OPERNUM`) which will be used to modify a specific property.
- The `MULTIREG` keyword has been extended to allow the user to choose the region (`MULTNUM`, `FLUXNUM` or `OPERNUM`) which will be used to modify a specific property.

- The COPYREG keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
- The MULTREGT keyword has been extended to allow the user to choose the region (MULTNUM or FLUXNUM) which will be used to multiply the transmissibilities.
- The MULTREGD keyword has been extended to allow the user to choose the region MULTNUM or FLUXNUM) which will be used to multiply the diffusivities.
- The MULTREGH keyword has been extended to allow the user to choose the region MULTNUM or FLUXNUM) which will be used to multiply the rock thermal conductivities.
- The MULTREGP keyword has been extended to allow the user to choose the region MULTNUM or FLUXNUM) which will be used to multiply the pore volumes.
- The BTOBALFA and BTOBALFV keywords now set up block to block connections in ECLIPSE 100 (global grid only) as well as ECLIPSE 300.
- The NMATOPTS keywords now also accepts an argument VERTICAL to use the ECLIPSE 100 Discrete Matrix modeling to calculate dual porosity gravity drainage.

EDIT section

- The EQUALREG keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
- The ADDREG keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
- The MULTIREG keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM)) which will be used to modify a specific property.
- The COPYREG keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM)) which will be used to modify a specific property.
- The MULTREGT keyword has been extended to allow the user to choose the region (MULTNUM or FLUXNUM) which will be used to multiply the transmissibilities.
- The MULTREGD keyword has been extended to allow the user to choose the region (MULTNUM or FLUXNUM) which will be used to multiply the diffusivities.
- The MULTREGH keyword has been extended to allow the user to choose the region (MULTNUM or FLUXNUM) which will be used to multiply the rock thermal conductivities.
- The MULTREGP keyword has been extended to allow the user to choose the region (MULTNUM or FLUXNUM) which will be used to multiply the pore volumes.

PROPS section

- The STONE1EX keyword can be used to apply an exponent to the saturation terms in the STONE1 three-phase relative permeability model. Output of values can be obtained from the RPTPROPS keyword
- The FOAMOPTS keyword can be used to select the phase for foam transport and the type of model to be used for calculating the mobility reduction due to foam.
- The FOAMFCN, FOAMFRM, FOAMFSC, FOAMFSO, FOAMFSW, and FOAMFST keywords can be used to specify the properties of the functional model of mobility reduction due to foam.
- The EQUALREG keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.

- The ADDREG keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
- The MULTIREG keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
- The COPYREG keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
- The FHERCHBL keyword must be used in conjunction with the NNEWTF keyword to specify the properties of the non-Newtonian fluid as functions of the polymer concentration.
- The SWATINIT keyword is used to match an initial water distribution. Two new scaling options for capillary pressure have been added to OPTIONS(74). When the SWATINIT keyword is used to match an initial water distribution for mixed wettability rocks and positive capillary pressure dominated then the scaling on P_{cow} is more sensible if OPTIONS(74)=5. When the SWATINIT keyword is used to match an initial water distribution for mixed wettability rocks and negative capillary pressure dominated then the scaling on P_{cow} is more sensible if OPTIONS(74)=7.
- A new model for Polymer Dead Pore Volume has been added. The pre-2008.2 model can be restored by setting switch 122 in the OPTIONS keyword to one.
- The first argument of the PLYROCK keyword (Dead Pore Volume) can now be set to a value between 0 and 1, but strictly smaller than 1.

REGIONS section

- The EQUALREG keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
- The ADDREG keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
- The MULTIREG keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
- The COPYREG keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.

SOLUTION section

- The EQUALS keyword has been added to allow you to modify grid values within the current box.
- The AQANCONL keyword has been added to allow analytic aquifers to be connected to local grid refinements.
- The CELLINDX mnemonic has been added to the RPTRST keyword. This outputs the cell indices corresponding to the other output properties. If any cells are missing due to inactivity this allows determination of which cells properties are actually present in the restart file without the need for information in the INIT file.
- The DYNREG mnemonic has been added to the RPTRST keyword. This outputs the cell indices corresponding to the dynamic regions specified using the DYNAMICR keyword.

- The SFIP mnemonic has been added to the RPTRST keyword. This is a synonym for the FIP mnemonic and is intended to reduce possible confusion when using FIP in both ECLIPSE 100 and ECLIPSE 300, to indicate that for ECLIPSE 100 surface/separator conditions are used.
- The RFIP mnemonic has been added to the RPTRST keyword to output fluid in place information at reservoir conditions.
- The EQUALREG keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
- The ADDREG keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
- The MULTIREG keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
- The COPYREG keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
- The DYNAMICR keyword has been added to define regions based on when a set of conditions are satisfied. These regions may be static or allowed to change dynamically.
- The ENDDYN keyword has been added to indicate the end of a DYNAMICR keyword sequence.
- The RIVERSYS keyword has been extended to include an option of solving the time-dependent Saint-Venant equations.
- The FOAMCNM mnemonic has been added to the to RPTSOL keyword to output the logarithm base ten of the capillary number for the FOAM option when the functional form of the gas mobility reduction model has been selected and the FOAMFCN keyword has been specified.
- The VISDATES keyword has been added to specify report times at which stress step calculations should be performed in the VISAGE external geomechanics workflow.
- The VISOPTS keyword has been added to specify options for the VISAGE external geomechanics workflow.

SUMMARY section

- New mnemonics FGSTP, FGSPR, FGSSP, FGSRU, FGSRL, GGSTP, GGSPR, GGSSP, GGSRU and GGSRL have been added to give data on sustainable capacity test results.
- New mnemonics CRREXCH, CRRPROT, CRRINJT, SRTFR, SRTFC added to give data on river results.
- New mnemonics, SRSFR and SRSFC, to output brine flow and concentration in rivers reaches.
- New mnemonics WPWE0, WPWE1, WPWE2, WPWE3, WPWE4, WPWE5, WPWE6, WPWE7 and WPWEM added to give information on well events such as drilling, completion opening and function changes.
- A new mnemonic WNEWTON has been added to output the number of well Newton iterations used in the last global Newton iteration at each timestep. A negative value indicates that the well model did not converge during the last global Newton iteration.
- New mnemonics (F,G)API and RAPI added to give average API of produced oil or oil in place. These mnemonics are for use with the API tracking option (see "API Tracking" on page 69).

- The mnemonic AAQP can now be used to output aquifer pressure for all aquifer types (except for constant flux aquifer as given by AQUFLUX).
- New keyword SUMTHIN reduces the amount of data written to summary files. The keyword defines a time period; only one datum will be recorded for each of these time periods. Data are always recorded at report times.
- New mnemonic BTCNMFOA has been added to output the logarithm base ten of the capillary number for the FOAM option when the functional form of the gas mobility reduction model has been selected and the FOAMFCN keyword has been specified.
- New mnemonics CTPR, CTIR, CTPC and CTIC have been added to give information on tracer production/injection rates and concentrations for well connections.

SCHEDULE section

- The keyword WFOAM may now be used with water injectors if the foam transport phase has been specified as water using the FOAMOPTS keyword.
- Aquifer and Block user defined quantities can be defined in UDQ.
- The WSEGLINK keyword has been added to allow looped flowpaths to be specified in multi-segmented wells.
- The GSSCPTST keyword has been added to allow sustainable capacity tests. This allows the gas production rate which a group of wells is capable of maintaining for a period of time to be determined. It can also determine the period of time for which a gas production rate is sustainable.
- A well list name may be specified to trigger actions in the ACTIONX keyword.
- The CELLINDX mnemonic has been added to the RPTRST keyword. This outputs the cell indices corresponding to the other output properties. If any cells are missing due to inactivity this allows determination of which cells properties are actually present in the restart file without the need for information in the INIT file.
- The DYNREG mnemonic has been added to the RPTRST keyword. This outputs the cell indices corresponding to the dynamic regions specified using the DYNAMICR keyword.
- The SFIP mnemonic has been added to the RPTRST keyword. This is a synonym for the FIP mnemonic and is intended to reduce possible confusion when using FIP in both ECLIPSE 100 and ECLIPSE 300, to indicate that for ECLIPSE 100 surface/separator conditions are used.
- The RFIP mnemonic has been added to the RPTRST keyword to output fluid in place information at reservoir conditions.
- Aquifer and block mnemonics may be used to trigger actions in the ACTIONX keyword.
- The WELEVNT keyword has been added to allow the SUMMARY section mnemonic WPWEM to be set from the data file.
- The keyword RIVTRACE has been added to allow tracer injection in rivers.
- The keyword RIVSALT has been added to allow salt injection in rivers.
- The RIVERSYS keyword has been extended to include an option of solving the time-dependent Saint-Venant equations.
- The RIVDEBUG keyword has been added to control output of river debug output.
- The RIVRPROP keyword has been added to update river reach properties in the schedule section.

- The MULTREGT keyword has been extended to allow the user to choose the region (MULTNUM or FLUXNUM) which will be used to multiply the transmissibilities.
- The DYNAMICR keyword has been added to define regions based on when a set of conditions are satisfied. These regions may be static or allowed to change dynamically.
- The ENDDYN keyword has been added to indicate the end of a DYNAMICR keyword sequence.
- The new mnemonic FOAMCNM has been added to the to RPTSCHED keyword to output the logarithm base ten of the capillary number for the FOAM option when the functional form of the gas mobility reduction model has been selected and the FOAMFCN keyword has been specified.
- In records 2 and upwards of the COMPSEGS and COMPSEGL keyword, a new item (11 for COMPSEGS and 12 for COMPSEGL) has been added, this allows the segment to which each grid block completion is allocated to be specified explicitly. If no explicit segment is specified, then the previous mechanism is used (that is, each grid block connection is allocated to the segment whose nodal point lies nearest to the center of the connection).
- The DATES keyword has a new argument for the time of the report.
- New options have been added to the keyword WLIFTOPT to permit the inclusion of a weighted gas rate term in the denominator of the lift gas gradient calculation and to permit wells to continue to be allocated lift gas after a group gas target has been achieved but whilst an oil rate limit has not been reached.

Aquifers

- An extra item has been added to the AQUCHWAT keyword to allow the depth term in the aquifer inflow equation to be removed from the calculation.
- The AQANCONL keyword has been added to allow analytic aquifers to be connected to local grid refinements.
- An extra item has been added to the AQUCHWAT keyword to allow the user to impose the minimum and maximum flow rates before or after the flow rate has been scaled by the grid block saturation in cases when water is flowing into the aquifer from the grid.

Tracers

- The convergence criterion for the non-cascade solver used for tracer tracking has been changed. The Pre-2008.2 behavior can be restored by setting OPTIONS option number 120 to 1.

New Facilities in ECLIPSE 300

RUNSPEC section

- Dimensions for aquifer and block user defined quantities can be specified in UDQDIMS.
- Dimensions for User Defined Arguments can be specified in UDADIMS.
- The first argument of ACTPARAM can now be used.
- The DYNRDIMS keyword has been added to set up dimensions for the Dynamic Region facility.

- H2SSOL activates the H2S solubility option.
- The TRPLPORO keyword sets up a triple porosity calculation.
- The CO2STORE option is now compatible with the AIM option when solids are present. Solid components include NACL,CACL2 and CACO3.
- The GRAVDRB keyword can be used to apply the vertical discretised gravity drainage model.
- The PETOPTS keyword has a new mnemonic NOPINCHC which suppresses calculation of pinchout NNCs.
- The START keyword has a new argument for the time at the start of the run.
- The VISAGE keyword has been added to activate the VISAGE external geomechanics workflow.
- An alternative memory allocation algorithm can be enabled by setting the item 163 of OPTIONS3 to 1.
- An alternative single-phase splitting attempt for Gibbs free energy minimization in flash is defaulted. Pre-2008.2 behavior can be restored using item 167 of the OPTIONS3 keyword.

GRID section

- The JFUNCR keyword can be used to specify Leverett J-Functions for the scaling of capillary pressure on a per saturation table basis, with the option of selecting capillary pressures from either direct table look-up or via J-Function calculations. These can be output with the RPTGRID keyword.
- The BTOBALFA and BTOBALFV keywords now also set up block to block connections in and to LGRs in ECLIPSE 300.
- The PETGRID keyword can now be used to import global grid geometry, local grid geometry and properties from a Generic Simulation Grid (.GSG) file. This type of file format has to be imported via Petrel.
- The EQUALREG keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
- The ADDREG keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
- The MULTIREG keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
- The COPYREG keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
- The MULTREGT keyword has been extended to allow the user to choose the region (MULTNUM, or FLUXNUM) which will be used to multiply the transmissibilities.
- The MULTREGP keyword has been extended to allow the user to choose the region (MULTNUM, or FLUXNUM) which will be used to multiply the pore volumes.
- The NMATOPTS keywords now also accepts an argument VERTICAL to use the ECLIPSE 300 Multi porosity modelling to calculate dual porosity gravity drainage.
- The ROCKSPLV keyword allows the user to specify non-default distributions of rock between matrix and fracture for dual porosity thermal calculations.

EDIT section

- The EQUALREG keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
- The ADDREG keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
- The MULTIREG keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
- The COPYREG keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
- The MULTREGT keyword has been extended to allow the user to choose the region (MULTNUM, or FLUXNUM) which will be used to multiply the transmissibilities.
- The MULTREGP keyword has been extended to allow the user to choose the region (MULTNUM, or FLUXNUM) which will be used to multiply the pore volumes.

PROPS section

- Items 4 to 20 are now in use and new items 23 to 26 have been added to keyword EPSODD3P. These provide control over surface tension based capillary pressure, relative permeability and saturation end-point scaling in the ODD3P model.
- A new feature to model formation damage has been added for compositional simulations. Damage factors are specified with the KRGDM, KRODM and KRWDM keywords and are used to calculate damage multipliers for the gas, oil and water relative permeabilities respectively. The component concentrations at which damage initiates can be specified with the KRGDI, KRODI and KRWDI keywords and the damage multipliers can be output by including the arguments KRGDM, KRODM and KRWDM in the RPTRST keyword.
- The STONE1EX keyword can be used to apply an exponent to the saturation terms in the STONE1 three-phase relative permeability model. Output of values can be obtained from the RPTPROPS keyword
- The pressure and temperature values used to calculate chemical reaction rates can now be limited using the REACLIMS keyword.
- The RKTRMDIR keyword can now be used exactly as for ECLIPSE 100.
- The Baker three-phase relative permeability model can be used (keywords: BAKER1, BAKER2)
- Residual oil saturation for compositional runs can be set using SORMW1 keyword in conjunction with ENDSCALE keyword.
- Cross phase diffusion when using the molecular diffusion option is available by using the keywords DIFFCOG, DIFFCGO, DIFFCWG and DIFFCGW.
- Relative permeability and capillary pressure curves can now be scaled depending on composition. A component is nominated using the EPSCOMP keyword and the curve end-points found by interpolating the ENKRVC, ENPCVC and ENPTVC keywords.
- The WAGHYSTR keyword can be used to supply the data items required by the WAG Hysteresis Model.
- The EQUALREG keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
- The ADDREG keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.

- The MULTIREG keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
- The COPYREG keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
- The default coefficients for the water and steam viscosities in a thermal simulation may now be supplied by the user via the THWVC and THSVC keywords.
- The interpolation method used in conjunction with the OILVISCT data can now be selected using the OILVTIM keyword.
- An alternative correlation for the water K-value in a thermal simulation has been added and can be specified using the KVCRWAT keyword.
- The K-values in a compositional simulation can now be interpolated on the mole fraction of a specified component. The K-value tables are entered using the KVTABCn keyword, the interpolation points via the KVMF keyword and the component specified using the KVCOMP keyword.
- The mutual solubility of CO2 and H2S in water can be achieved by using the keywords SOLUCO2H and SOLUH2SC.
- The ASPP1P is set to define the variable the precipitation is to be set against (pressure or temperature or concentration of a specified component). This keyword needs to be used in conjunction with ASPREWG.
- The ASPP2P is set to define variables the precipitation is to set against (pressure and temperature, pressure and concentration, temperature and concentration). This keyword needs to be used with ASPPW2D.
- The ASPPW2D is set to provide a 2D table of the asphaltene precipitation as a function of the variables set in ASPP2P.
- The ASPLCRT is set to control the plugging as a function of volume fraction of asphaltene deposit and flocs concentration.
- The ASPKROW is set to provide the oil-wet relative permeability for water and oil as functions of the water saturation for the wettability change due to asphaltene deposition. This keyword must be used in conjunction with ASPWETF.
- The ASPWETF is set to provide the weighting factor used in the wettability change due to asphaltene deposition to model the transition from an water wet system to an oil wet system. This keyword must be used in conjunction with ASPKROW.
- The SWATINIT keyword is used to match an initial water distribution. Two new scaling options for capillary pressure have been added to OPTIONS3(143). When the SWATINIT keyword is used to match an initial water distribution for mixed wettability rocks and positive capillary pressure dominated then the scaling on P_{cow} is more sensible if OPTIONS3(143)=5. When the SWATINIT keyword is used to match an initial water distribution for mixed wettability rocks and negative capillary pressure dominated then the scaling on P_{cow} is more sensible if OPTIONS3(143)=7.
- The REACOPS keyword has been added to specify options specific to the chemical reaction model including convergence criteria.
- The EHYSTR keyword item 12 allows a threshold saturation of Killough's hysteresis model for both wetting and non-wetting phases. This threshold allows better control of the numerical sensitivity of the system, preventing it from being too unstable.
- The OPTODD3P keyword is used to provide control over the choice of model and the method of detecting saturation direction changes for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.

REGIONS section

- The EQUALREG keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
- The ADDREG keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
- The MULTIREG keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
- The COPYREG keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.

SOLUTION section

- A new mnemonic FIP has been added to the RPTSOL keyword to give fluid-in-place reports.
- The CELLINDX mnemonic has been added to the RPTRST keyword. This outputs the cell indices corresponding to the other output properties. If any cells are missing due to inactivity this allows determination of which cells properties are actually present in the restart file without the need for information in the INIT file.
- The DYNREG mnemonic has been added to the RPTRST keyword. This outputs the cell indices corresponding to the dynamic regions specified using the DYNAMICR keyword.
- The RFIP mnemonic has been added to the RPTRST keyword. This is a synonym for the FIP mnemonic and is intended to reduce possible confusion when using FIP in both ECLIPSE 100 and ECLIPSE 300, to indicate that for ECLIPSE 300 reservoir conditions are used. Note that this only allows for four significant characters of the component name to be added to the restart file, instead of the five for FIP. Oil, water and gas fluid in place information is also output.
- Three new mnemonics SFIPGAS, SFIPOIL and SFIPWAT have been added to the RPTRST keyword to output FIP data at separator conditions.
- The SFIP mnemonic has been added to the RPTRST keyword to output FIP data at separator conditions. Using this mnemonic will output SFIPGAS, SFIPOIL and SFIPWAT.
- The EQUALREG keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
- The ADDREG keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
- The MULTIREG keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
- The COPYREG keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
- The DYNAMICR keyword has been added to define regions based on when a set of conditions are satisfied. These regions may be static or allowed to change dynamically.
- The ENDDYN keyword has been added to indicate the end of a DYNAMICR keyword sequence.
- The AQANCONL keyword has been added to allow analytic aquifers to be connected to local grid refinements.

- A new mnemonic AQANCONL has been added to the RPTSOL keyword to output aquifer grid block connection data within local grid refinements.
- The mnemonics FLOOIL and FLOGAS for the RPTRST keyword can now be used to output inter-block flows for asphaltene models.
- A new mnemonic SGTRAP has been added to the RPTRST keyword to output the trapped gas saturation.
- New mnemonics SWU, SWL, SWCR, SGU, SGL, SGCR, SOWCR, SOGCR have been added to the RPTSOL keyword to output the scaled saturation end-points in the case of temperature or composition dependent scaling.
- New RPTSOL mnemonics PCOWR, PCWOR, PCOGR, PCGOR, PCGWR and PCWGR have been added for the ODD3P three-phase relative permeability and capillary pressure hysteresis model option.
- The VISDATES keyword has been added to specify report times at which stress step calculations should be performed in the VISAGE external geomechanics workflow.
- The VISOPTS keyword has been added to specify options for the VISAGE external geomechanics workflow.

SUMMARY section

- New mnemonics FAMR, GAMR, FAMT and GAMT added giving make-up gas rates and totals.
- New mnemonics FGLR, GGLR, WGLR and CGLR added giving gas-liquid ratios.
- New mnemonics FGSTP, FGSPR, FGSSP, FGSRU, FGSRL, GGSTP, GGSPR, GGSSP, GGSRU and GGSRL have been added to give data on sustainable capacity test results.
- New mnemonics ROPR,ROPT,RWPR,RWPT,RGPR,RGPT,RNLPR,RNLPT have been added to give rates and totals data.
- New mnemonics RNLFR,RNLFT have been added to give inter-region NGL flow rates and totals data.By default, the inter-region flowing fractions are calculated using the region-averaged values.If OPTIONS3(127) is set to a non-zero value the block-by-block inter-region flowing fractions calculation for the table GPTABLEN look up is activated.
- New mnemonics FWGPRH, GWGPRH and WWGPRH have been added to give observed wet gas production for history matching.
- For the CO2STORE option, new mnemonics FGCDI, FGCDM, FWCD and RGCDI, RGCDM, RWCD have been added to give on a field and regional basis the molar amount of CO_2 in the water and gas phase.
- For the CO2STORE option, new mnemonics BAQPH and BAQSP have been added to give on a block basis the pH level and the molar densities of the ions in the aqueous phase obtained by aqueous speciation.
- Asphaltene: New mnemonics have been added to output the asphaltene precipitate total fraction, dissolved limit, and dissolved fraction: ASPRET, ASPLIM, ASPFRD and their block value equivalent BASPRET, BASPLIM, and BASPFRD. Adsorption, plugging and entrainment outputs for the deposition are now also available for block values using BASPADS, BASPLUG and BASPENT respectively. In addition mnemonics ASPLUG and ASPENT should be used instead of ASPLU and ASPEN respectively.
- The summary mnemonics MLINEARP, MSUMLINP, NCPRLINS, NLINEARP have been added to report performance indicators from the JALS solver.
- The summary mnemonics MAXDE, MAXDT have been added to evaluate solution changes in thermal simulations.

- The summary mnemonics WCGIR, WCGPR, WCOPR have been added to output oil and gas well production rates on a component basis.
- New summary mnemonics AACMR and AACMT have been added to give the component molar influx rates and totals for analytic multi-component aquifers. The newly added summary mnemonics ANCMR and ANCMT provide the same functionality for numerical aquifers.
- The summary mnemonics FGPRG, FOPRG, FNLPRG, FXMFG, FYMFG, FCOMRG, FCGMRG, FCNMRG, GGPRG, GOPRG, GNLPRG, GXMFG, GYMFG, GCOMRG, GCGMRG and GCNMRG have been added to output oil and gas production rates and compositions as calculated by summing well outputs and then separating rather than separating and then summing. The keyword GSEPREPT has also been added to specify the separator conditions for these calculations.
- New mnemonics WPWE0, WPWE1, WPWE2, WPWE3, WPWE4, WPWE5, WPWE6, WPWE7 and WPWEM added to give information on well events such as drilling, completion opening and function changes.
- A new mnemonic WNEWTON has been added to output the number of well Newton iterations used in the last global Newton iteration at each timestep. A negative value indicates that the well model did not converge during the last global Newton iteration.
- The mnemonic AAQP can now be used to output aquifer pressure for all analytic aquifer types (except for constant flux aquifer as given by AQUFLUX).
- The OFM keyword is now available in ECLIPSE 300, allowing the user to create output files for the OFM program.
- New keyword SUMTHIN reduces the amount of data written to summary files. The keyword defines a time period; only one datum will be recorded for each of these time periods. Data are always recorded at report times.
- The mnemonics NTS, NTSPCL, NTSMCL, NTSECL have been added to report the number of timesteps taken and the number of timesteps in which the pressure, molar densities and energy variables converged last. The mnemonics NLRESSUM and NLRESMAX report the sum of the non-linear residual and the maximum element of the non-linear residual. This collection of mnemonics can be output by including TUNE in the SUMMARY section.
- New mnemonics BGTRP and BGTPD, to output trapped gas saturation and dynamic trapped gas saturation respectively (WAG hysteresis only).
- For GASWAT or CO2STORE, setting the 151st item of OPTIONS3 makes it possible to consider only the gas phase as hydrocarbon. This affects the calculation of the summary keywords FPR, FPRH and FPPO. By default, the water and gas phases are considered as hydrocarbon.
- New mnemonics BSWU, BSWL, BSWCR, BSGU, BSGL, BSGCR, BSOWC, BSOGC, to output the scaled saturation table end-points in the case of temperature or composition depending scaling.
- New mnemonics BPCOWR, BPCWOR, BPCOGR, BPCGOR, BPCGWR and BPCWGR have been added for the ODD3P three-phase relative permeability and capillary pressure hysteresis model option.
- A mnemonic BSMF has been added to output the component solid mole fractions when using the CO2STORE and SOLID option.
- The new mnemonics BIFTWO, BIFTOG and BIFTWG have been added as aliases for the existing mnemonics BIFTOW, BIFTGO and BIFTGW respectively for the ODD3P three-phase relative permeability and capillary pressure hysteresis model option.

SCHEDULE section

- The GSSCPTST keyword has been added to allow sustainable capacity tests. This allows the gas production rate which a group of wells is capable of maintaining for a period of time to be determined. It can also determine the period of time for which a gas production rate is sustainable.
- Aquifer and Block user defined quantities can be defined in UDQ.
- The CELLINDX mnemonic has been added to the RPTRST keyword. This outputs the cell indices corresponding to the other output properties. If any cells are missing due to inactivity this allows determination of which cells properties are actually present in the restart file without the need for information in the INIT file.
- The DYNREG mnemonic has been added to the RPTRST keyword. This outputs the cell indices corresponding to the dynamic regions specified using the DYNAMICR keyword.
- Three new mnemonics SFIPGAS, SFIPOIL and SFIPWAT have been added to the RPTRST keyword to output FIP data at separator conditions.
- The RFIP mnemonic has been added to the RPTRST keyword. This is a synonym for the FIP mnemonic and is intended to reduce possible confusion when using FIP in both ECLIPSE 100 and ECLIPSE 300, to indicate that for ECLIPSE 300 reservoir conditions are used. Note that this only allows for four significant characters of the component name to be added to the restart file, instead of the five for FIP. Oil, water and gas fluid in place information is also output.
- The SFIP mnemonic has been added to the RPTRST keyword to output FIP data at separator conditions. Using this mnemonic will output SFIPGAS, SFIPOIL and SFIPWAT.
- Aquifer and block mnemonics may be used to trigger actions in the ACTIONX keyword.
- Additional output for multi-segment wells is available in the RFT file including segment temperature, steam quality, energy, enthalpy and saturation pressure.
- The DELAYACT keyword has been added.
- The WELEVNT keyword has been added to allow the SUMMARY section mnemonic WPWEM to be set from the data file.
- LIFT can now be used in item 2 of the WTADD and WTMULT keywords. This removes an incompatibility with ECLIPSE 100 when using these keywords.
- The MULTREGT keyword has been extended to allow the user to choose the region (MULTNUM or FLUXNUM) which will be used to multiply the transmissibilities.
- The DYNAMICR keyword has been added to define regions based on when a set of conditions are satisfied. These regions may be static or allowed to change dynamically.
- The ENDDYN keyword has been added to indicate the end of a DYNAMICR keyword sequence.
- PICOND has been extended to allow the user to define under what conditions the generalized pseudo-pressure look up table is recalculated. See"Special inflow equations" on page 1183 for details on generalized pseudo-pressure
- PICOND also has an alternative quadrature option allowing the generalised pseudo-pressure table to generate more detail in nonlinear areas and less in linear areas.
- WCONHIST may now have wet gas rate as a control mode by setting item 3 as WGRA.
- WELLWAG may now be controlled by surface rate, BHP or THP. Separate VFP tables may also be set for each phase injected.

- GOPRH, GGPRH, GWPRH, GLPRH, GOPTH, GGPTH, GWPTH, GLPTH, GOIRH, GGIRH, GWIRH, GOITH, GGITH and GWITH added to the list of mnemonics which work with WLIST groups.
- The mnemonics FLOOIL and FLOGAS for the RPTRST keyword can now be used to output inter-block flows for asphaltene models.
- A new mnemonic SGTRAP has been added to the RPTRST keyword to output the trapped gas saturation.
- New mnemonics SWU, SWL, SWCR, SGU, SGL, SGCR, SOWCR, SOGCR have been added to the RPTRST and RPTSCHED keywords to output the scaled saturation end-points in the case of temperature or composition dependent scaling.
- New RPTSCHED mnemonics PCOWR, PCWOR, PCOGR, PCGOR, PCGWR and PCWGR have been added for the ODD3P three-phase relative permeability and capillary pressure hysteresis model option.
- A mnemonic SMF has been added to RPTRST and RPTSCHED keywords to output the component solid mole fractions when using the CO2STORE and SOLID option.
- In records 2 and upwards of the COMPSEGS and COMPSEGL keyword, a new item (11 for COMPSEGS and 12 for COMPSEGL) has been added, this allows the segment to which each grid block completion is allocated to be specified explicitly. If no explicit segment is specified, then the previous mechanism is used (that is, each grid block connection is allocated to the segment whose nodal point lies nearest to the center of the connection).
- The DATES keyword has a new argument for the time of the report.
- New options have been added to the keyword WLIFTOPT to permit the inclusion of a weighted gas rate term in the denominator of the lift gas gradient calculation and to permit wells to continue to be allocated lift gas after a group gas target has been achieved but whilst an oil rate limit has not been reached.

Aquifers

- An extra item has been added to the AQUCHWAT keyword to allow the depth term in the aquifer inflow equation to be removed from the calculation.
- For the GASWAT and CO2STORE options, AQSTREAM can be used to specified the water composition of an AQUCHWAT, AQUCT, AQUFETP or AQUFLUX aquifer.
- For the multi-component water option (COMPW), AQSTREAW can be used to specified the water composition of an AQUCHWAT, AQUCT, AQUFETP or AQUFLUX aquifer.
- The AQANCONL keyword has been added to allow analytic aquifers to be connected to local grid refinements.
- An extra item has been added to the AQUCHWAT keyword to allow the user to impose the minimum and maximum flow rates before or after the flow rate has been scaled by the grid block saturation in cases when water is flowing into aquifer from grid.
- Item 11 of the AQUANCON and AQUCON keywords are now supported in ECLIPSE 300.

Reservoir Optimization

- An extra item has been added to the OPTPARS keyword to allow the specification of the well control mode which should be used with each SCSA parameter. For all other parameters, the control mode used is based directly on the parameter type (as specified with OPTPARS item 1). Since there is no obvious choice of control mode with SCSA parameters this additional item allows the desired well control mode to be specified on a well by well basis.
- Pre-2007.1, wells with SCSA parameters were kept under whatever control mode was specified in the optimization base case simulation. At 2007.1, wells with SCSA parameters were always under BHP control.

Behavioral changes in ECLIPSE 100

- The list of available licenses is not output by default anymore. This output can be enabled by setting the LISTLICENSES keyword to TRUE in the configuration file.

GRID section

- Behavior of the RADFIN4 keyword has changed to ensure the correct inheritance of porosity and calculation of local grid pore volumes for the case where THETA=4. This change may be reverted using by entering 1as the 112th value in the OPTIONS keyword.

PROPS section

- A correction has been made to the calculation of relative permeability for Stone's Second Model when end-point scaling has also been specified. Pre-2008.2 behavior can be restored by setting the OPTIONS option number 121 to a non-zero value.

SOLUTION section

- Behavior of the AQUCHWAT water density calculation, which is used to compute the aquifer inflow, has changed. This change may be reverted using by entering a non-zero value as the 110th value in the OPTIONS keyword.

SUMMARY section

- The RPTONLY and RPTONLYO keywords are now ignored if they are in the SUMMARY section of restarted runs, and they have to be re-specified at or after the restart time in the SCHEDULE section. The 109th value in the OPTIONS keyword can be used to restore the previous behavior in which the SUMMARY section entries are not ignored in restart files.
- Memory allocation problems associated with the OFM keyword have been corrected.

SCHEDULE section

- The VFPPROD mnemonic for the RPTSCHED keyword now produces tables comparing maximum dimensions from the VFPPDIMS keyword to those observed in the input tables. This output is produced for VFPPROD=2 or VFPPROD=3.
- If a well has no completions in active blocks a warning message is now produced. In previous versions of ECLIPSE 100 an error message was produced.

- Scale deposition (controlled by WSCTAB) now applies to both injecting and producing connections instead of only producing connections. This change may be reverted by entering a non-zero value as the 108th value in the OPTIONS keyword.
- Behavior of the AQUCHWAT water density calculation, which is used to compute the aquifer inflow, has changed. This change may be reverted using by entering a non-zero value as the 110th value in the OPTIONS keyword.
- The WTEST keyword is no longer reset when a group of production wells is closed by a single event such as a GECON rate violation or certain gas field operations events. This change may be reverted by entering a non-zero value as the 111th value in the OPTIONS keyword.
- If API tracking and local grid refinement are used together, the calculation of oil density in global grid cells is now calculated from local grid values using an oil-in-place weighting instead of a pore volume weighting. Pre-2008.1 behavior may be recovered by entering a non-zero value as the 113th value of the OPTIONS keyword.
- An improved water and solvent saturation bound-checking is performed by setting the 114th value of OPTIONS keyword to a non-zero value.
- In a system containing both dissolved gas and vaporized oil, the maximum rates of increase of R_s and R_v may now be specified using the DRSDT and DRVDT keywords respectively. If the VAPPARS keyword is specified subsequently, the limits imposed by the DRSDT and DRVDT keywords will be disabled. Similarly, the constraints imposed by the specification of the VAPPARS keyword will be disabled by the subsequent specification of either the DRSDT or DRVDT keywords. Pre-2008.2 behavior can be restored by setting the OPTIONS option number 118 to a non-zero value.

Aquifers

- Analytical aquifers now correctly calculate the flow from reservoir to aquifer in the case of connate water or gas. The pre-2008.1 behavior can be obtained by setting item 115 of the OPTIONS keyword to a non-zero value.

Polymer/Foam

- If OPTIONS switch 116 is set to one it restores the 2008.1 behavior.
- If OPTIONS switch 122 is set to one it restores the pre-2008.2 Polymer Dead Pore Volume model.

Behavioral changes in ECLIPSE 300

- The list of available licenses is not output by default anymore. This output can be enabled by setting the LISTLICENSES keyword to TRUE in the configuration file.

RUNSPEC section

- The THERMAL and DUALPORO or DUALPERM keywords together now imply that for thermal dual porosity calculations the rock volumes will be distributed between the porosities so that rock volume in the fracture porosity is less than or equal to the fracture pore volume, with the remainder of the rock volume associated with the matrix. This may be overridden using the ROCKSPLV keyword. Previously bulk volume less pore volume was used for each porosity separately.

- The ASPHALTE keyword arguments' handling has been improved. Permeability and viscosity changes can be skipped by default or by setting the 2nd and 3rd argument to 'NO' respectively.

PROPS section

- The SWATINIT calculation has been improved. The pre 2008.1 behavior can be restored using OPTIONS3(148).
- The ODD3P three-phase relative permeability and capillary pressure hysteresis model calculation has been improved. Pre-2008.2 behavior may be restored using the OPTODD3P keyword.

SOLUTION section

- The RPTRST and RPTSOL mnemonic MLSC now only outputs the moles per reservoir volume for the hydrocarbon components. Similar data for the water component(s) can be requested with the MWAT mnemonic.
- The RPTRST and RPTSOL mnemonic RPORV now outputs the reservoir pore volumes at Reservoir conditions.

SUMMARY section

- FCMIR, GCMIR, WCMIR, FCMIT, GCMIT and WCMIT are now reported with the correct sign.
- If CHANDIMS and LOAD are both used well related keywords can now be output for all wells.
- FNPR, GNPR, WNPR are now changed onto FNLPR, GNLPR, WNLPR to avoid confusion with ECLIPSE 100 salt mnemonics.
- Asphaltene: the following block mnemonics BASPDO, BASPRW and BASPVM have been changed to BASPDOT, BASPREW and BASPVOM for easy use and consistency with their field counterparts. In addition the following field mnemonics ASPEN and ASPLU have changed to ASPENT and ASPLUG for more consistency, and are not available anymore.
- BPORV outputs the cell pore volumes at reference conditions.

SCHEDULE section

- The tolerances used in the calculation of immobile fluid derivatives have been modified to prevent large elements occurring in the Jacobian. The pre-2007.2 behavior can be restored with the 128th argument of the OPTIONS3 keyword.
- Scale deposition (controlled by WSCTAB) now applies to both injecting and producing connections instead of only producing connections.This change may be reverted using by entering a non-zero value as the 129th value in the OPTIONS3 keyword.
- Gas reinjection from separator stages (controlled by GINJGAS) now respects higher priority sales and fuel targets correctly (controlled by GRUPSALE, GRUPFUEL and WTAKEGAS) when the sales/fuel group has additional injection rate controls. This change may be reversed by setting item 130 of OPTIONS3 to either 2, 3, 6 or 7 depending on which other elements of this option are required.

- Gas reinjection from separator stages (controlled by GINJGAS) now has a more relaxed check on sales and fuel targets (controlled by GRUPSALE and GRUPFUEL). This change may be reversed by setting item 130 of OPTIONS3 to either 4, 5, 6 or 7 depending on which other elements of this option are required.
- Some minor improvements have been made in the calculation of blocking factor for generalized pseudo-pressure. These may be reversed or activated by setting OPTIONS3 keyword item 132 to an appropriate value.
- The flows output by the RPTRST, RPTSCHED and RPTSOL keywords are now calculated at the end of the timestep rather than the start. The pre-2008.1 behavior can be reversed using item 134 of the OPTIONS3 keyword.
- If advance gas is used (see WADVANCE and GADVANCE) with WINJORD or WINJMIX then advance gas is added for the source well or group once only instead of once per separator stage used. The injection composition calculation and advance gas priority have been corrected. The pre-2008.1 behavior may be recovered using item 139 of the OPTIONS3 keyword.
- Saturation chops, in EOS mode, have been modified. The components are now ordered by the critical temperature in order to determine the most volatile components. The pre-2008.1 behavior may be recovered using item 138 of the OPTIONS3 keyword.
- If BLACKOIL is selected the segregated well head calculation now uses the fluid density for the well's PVT region instead of using PVT region 1 for all wells. The pre-2007.2 behavior may be recovered using item 141 of the OPTIONS3 keyword.
- The calculation of lumped tracer connection flows has been modified to include a Li correlation factor in the wellbore flash. The pre-2007.2 behavior can be obtained by setting item 140 of the OPTIONS3 keyword to a non-zero value.
- If item 5 of WELSPECS is defaulted then the bottom hole pressure reference depth is set to the depth of the center of the first connected block as defined by COMPORD. The pre-2008.2 behavior may be recovered using item 142 of the OPTIONS3 keyword. Note that 2008.2 behavior differs from 2008.1 behavior, with 2008.1 behavior giving unexpected results in some circumstances.
- The RPTRST and RPTSCHED mnemonic MLSC now only outputs the moles per reservoir volume for the hydrocarbon components. Similar data for the water component(s) can be requested with the MWAT mnemonic.
- If sales gas (GRUPSALE), fuel gas (GRUPFUEL) and reinjection gas (GINJGAS or WINJGAS) are specified at different levels of the group hierarchy then sales and fuel gas calculations now correctly take account of gas reinjected at a higher level group. This only affects calculations not using mixture gas (WINJORD or WINJMIX). The pre-2008.1 behavior may be recovered using item 139 of the OPTIONS3 keyword.
- The one phase selection method (labeling) now considers the Gibbs free energy of the possible states (liquid or vapor) and chooses the state with the lowest Gibbs free energy. If equal, the Li-criteria is applied. The pre-2008.1 method using the Li-criteria only can be activated by OPTIONS3 item 146 put to 6.
- The calculation of pressure equivalent radius and kh (and therefore well connection factor) for grid blocks at the center of radial grids and LGRs has been changed to assume that the well is at the center of the grid. This affects some non-Darcy flow calculations, and may be reversed by setting item 149 of the OPTIONS3 keyword.
- The second argument of the WTEST keyword is now respected if GECON shuts in a group of production wells. The pre-2008.1 behavior may be recovered by setting item 150 of the OPTIONS3 keyword.

- If partial gas reinjection is defined using mixture gas (WINJORD or WINJMIX) the volume reinjected is now correctly calculated using the net available gas after sales, fuel and advance gas have been applied. This change may be reversed using item 139 of the OPTIONS3 keyword.
- Gas reinjection using mixture gas (WINJORD or WINJMIX) no longer causes QDRILL to drill new gas injection wells before they are required. This change may be reversed using item 139 of the OPTIONS3 keyword.
- An improved vapor versus liquid density check is performed. The pre-2008.1 behavior will be restored by setting item 153 of the OPTIONS3 keyword to a non-zero value.
- The RPTRST and RPTSCHED mnemonic RPORV now outputs the reservoir pore volumes at Reservoir conditions.
- Reinjection of gas with well composition under group control (for example GCONINJE with WINJGAS) or group composition under well control now functions correctly. This change may be reversed using item 159 of the OPTIONS3 keyword.
- A warning is now issued if a group has a REIN target set using GCONINJE element 6 and make-up gas set using GINJGAS as this combination has never been compatible. Make-up gas will be switched off for this group. This change may be reversed using item 159 of the OPTIONS3 keyword.
- Gas accounting now accounts for the GEFAC keyword correctly. Pre-2008.2 behavior may be recovered using item 159 of the OPTIONS3 keyword.
- If advance gas and the WEFAC and/or GEFAC keywords are both selected advance gas is now calculated correctly. Pre-2008.2 behavior may be recovered using item 159 of the OPTIONS3 keyword.
- If WAVAILIM and the WEFAC and/or GEFAC keywords are both selected the availability limit is now calculated correctly. Pre-2008.2 behavior may be recovered using item 159 of the OPTIONS3 keyword.
- If there is insufficient gas to meet sales, fuel and reinjection targets then sales and fuel gas will only be reduced if the WAVAILIM keyword has been specified. Pre-2008.2 behavior may be recovered using element 95 of the OPTIONS3 keyword.
- The non-linear damping algorithm for thermal simulations has been improved. The pre 2008.2 default behavior can be restored using item 160 of the OPTIONS3 keyword.
- An alternative single-phase splitting attempt for Gibbs free energy minimization in flash is defaulted. Pre-2008.2 behavior can be restored using item 167 of the OPTIONS3 keyword.
- The RPTSCHED mnemonic FIP now gives the correct material balance when satellite keywords GSATPROD and GSATINJE are used. Pre-2008.2 behavior can be restored using item 168 of the OPTIONS3 keyword.

Aquifers

- For the GASWAT and CO2STORE options, the water entering / leaving the reservoir from an analytic water aquifer can be made up of any components. The pre-2007.2 behavior can be restored with the 131th argument of the OPTIONS3 keyword.
- For the multi-component water option (COMPW), the water entering / leaving the reservoir from an analytic water aquifer can be made up of any water components. The pre-2008.1 behavior can be restored with the 131th argument of the OPTIONS3 keyword.
- For the CO2SOL option, any water component is allowed to leave the reservoir. If the item 131 of OPTIONS3 is set to 1, this reverts to the pre-2008.1 behavior where the water entering / leaving the reservoir is made up entirely of the “WATER” component.

- The handling of capillary pressure in analytic aquifers has been improved. The pre-2008.1 behavior can be enabled by setting the item 145 of `OPTIONS3` to 1.
- Analytical aquifers now correctly calculate the flow from reservoir to aquifer in the case of connate water or gas. The pre-2008.1 behavior can be obtained by setting item 154 of the `OPTIONS3` keyword to a non-zero value.

CO2STORE and GASWAT

- The phase stability of the `GASWAT` option is improved.The determination of a single phase as gas or water uses a simplified criteria. If the temperature is less than the pseudo critical temperature, as computed by the Li criteria, see keyword `FACTLI`, and the phase contains mostly H_2O, the phase is labeled water, otherwise gas.It is possible to activate the pre-2008.1 behavior by putting `OPTIONS3` switch 146 to 6. Also the flash will treat H_2O as the heaviest component unless `OPTIONS3` switch 125 is put to -1
- The component NACL_S in the `CO2STORE` option is obsolete, only NACL should be used.

Equation of State

- The phase determination of a single phase compares the Gibbs free energy of a possible liquid state and a possible vapor state. The phase with lowest Gibbs free energy is used to determine the state. If equal, the Li criteria is used, see keyword `FACTLI`. It is possible to activate the pre-2008.1 behavior by putting `OPTIONS3` switch 146 to 6.
- An alternative single-phase splitting attempt for Gibbs free energy minimization in flash is defaulted. Pre-2008.2 behavior can be restored using item 167 of the `OPTIONS3` keyword

Chemical Reactions

- For simulations containing chemical reactions an additional convergence criterion has been placed on the sum of the non-linear residual. Pre 2008.1 behavior can be restored by setting the first item of the `REACOPS` keyword negative.
- For simulations using the chemical reactions feature the Appleyard chop is now applied to components when they appear or disappear. The Appleyard chop force the solution to pass through these points in a more controlled fashion in order to stabilize the solution. The pre 2008.2 behavior can be restored by setting item 158 of the `OPTIONS3` keyword to -1.

Tilted Block Equilibration

- The 2008.1 equilibration calculation uses the aquifer block properties instead of the tilted block geometrical properties for numerical aquifers associated with grid blocks of zero thickness. Pre 2008.1 behavior can be restored by setting item 155 of the `OPTIONS3` keyword to a non-zero value.
- The 2008.1 equilibration calculation corrects the weighted sum calculation for tilted grid blocks, defined using corner point geometry, by adjusting the averaged inclination factors to prevent the planes representing the top and bottom faces based on these averaged inclination factors from intersecting within the grid block. This correction will typically apply to distorted grid blocks, for example, those for which one or more corners share the same or similar depth values for both the upper and lower faces. Pre 2008.1 behavior can be restored by setting item 156 of the `OPTIONS3` keyword to a non-zero value.

ODD3P Relative Permeability and Capillary Pressure Hysteresis Model

- The 2008.1 version of the `ODD3P` relative permeability and capillary pressure hysteresis model includes a considerable number of extensions and refinements as compared to the pre 2008.1 versions. Because of the nature of these changes, no backwards compatibility has been provided for pre 2008.1 versions.

New keywords in ECLIPSE 100

RUNSPEC section

The new `RUNSPEC` keywords are:

Table G.31 RUNSPEC keywords

Keyword	Description
`UDADIMS`	Dimensions for User Defined Arguments
`WPOTCALC`	Control calculation of shut and stopped well potentials
`WSEGSOLV`	Sets parameters for the multi-segment well iterative linear solver
`DYNRDIMS`	This keyword is used to set up dimensions for the Dynamic Region facility.
`GRAVDRB`	Vertical discretised gravity drainage model.
`NNEWTF`	Activates the non-Newtonian model (when the Polymer option is used)
`VISAGE`	Activates the VISAGE geomechanics workflow

GRID section

The new `GRID` section keywords are:

Table G.32 GRID keywords

Keyword	Description
`BTOBALFA`	Create block to block connections (single value multiplier)
`BTOBALFV`	Create block to block connections (vector of multipliers)
`JFUNCR`	Specification of Leverett J-Functions for the scaling of capillary pressure on a per saturation table basis.

PROPS section

The new PROPS section keywords are:

Table G.33 PROPS keywords

Keyword	Description
FOAMOPTS	Control over foam transport phase and method of modeling gas mobility reduction due to foam.
FOAMFCN	Specification of functional dependence of gas mobility reduction due to foam upon capillary number.
FOAMFRM	Specification of functional dependence of gas mobility reduction due to foam upon reference mobility reduction.
FOAMFSC	Specification of functional dependence of gas mobility reduction due to foam upon surfactant concentration.
FOAMFSO	Specification of functional dependence of gas mobility reduction due to foam upon oil saturation
FOAMFST	Specification of gas-water surface tension in the presence of surfactant generating foam as a function of concentration
FOAMFSW	Specification of functional dependence of gas mobility reduction due to foam upon water saturation.
STONE1EX	Optional exponent applied to the saturation terms used in the STONE1 three-phase relative permeability model to calculate the oil relative permeability.
FHERCHBL	Specifies the properties of the non-Newtonian model as functions of the polymer concentration

SOLUTION section

The new SOLUTION keywords are:

Table G.34 SOLUTION keywords

Keyword	Description
AQANCONL	This allows analytic aquifers to be connected to local grid refinements
EQUALS	Set an array to a constant value in the current box
DYNAMICR	This keyword defines regions based on when a set of conditions are satisfied. These regions may be static or allowed to change dynamically.
ENDDYN	This keyword indicates the end of a DYNAMICR keyword sequence.
VISDATES	Specify VISAGE geomechanics stress step dates
VISOPTS	Specify VISAGE geomechanics workflow options

SUMMARY section

The new SUMMARY keywords are:

Table G.35 SUMMARY keywords

Keyword	Description
BASPADS	Volume fraction of asphaltene deposit by adsorption
BASPENT	Volume fraction of asphaltene entrained
BASPFRD	Volume fraction of asphaltene precipitated dissolved
BASPLIM	Volume fraction of asphaltene content (see ASPREWG keyword)
BASPLUG	Volume fraction of asphaltene deposit by plugging
BASPRET	Volume fraction of asphaltene precipitate including flocs component
BTCNMFOA	Capillary number for FOAM option and FOAMFCN keyword.
FGSPR	Target sustainable rate for most recent sustainable capacity test for field.
FGSRL	Maximum sustainable rate found for the most recent sustainable capacity test for field.
FGSRU	Minimum un-sustainable rate found for the most recent sustainable capacity test for field.
FGSSP	Period for which target sustainable rate could be maintained for the most recent sustainable capacity test for field.
FGSTP	Test period for the most recent sustainable capacity test for field.
GGSPR	Target sustainable rate for most recent sustainable capacity test for group.
GGSRL	Maximum sustainable rate found for the most recent sustainable capacity test for group.
GGSRU	Minimum un-sustainable rate found for the most recent sustainable capacity test for group.
GGSSP	Period for which target sustainable rate could be maintained for the most recent sustainable capacity test for group
GGSTP	Test period for the most recent sustainable capacity test for group.
SRTFR	Tracer flow rate for reach
SRTFC	Tracer concentration for reach
SRSFR	Brine flow rate for reach.
SRSFC	Brine concentration for reach.
CRREXCH	Water rate exchange for connection to river
CRRPROT	Water cumulative production for connection to river
CRRINJT	Water cumulative injection for connection to river
WNEWTON	Number of well Newton iterations used in the last global Newton iteration at each timestep. A negative value indicates that the well model did not converge during the last global Newton iteration
WPWE0	Well drilled indicator.
WPWE1	Connection opened indicator.
WPWE2	Connection shut indicator

Table G.35 SUMMARY keywords

Keyword	Description
WPWE3	Connection shut to bottom indicator.
WPWE4	Well stopped indicator.
WPWE5	Injector set to producer indicator.
WPWE6	Producer set to injector indicator.
WPWE7	Well shut indicator.
WPWEM	WELEVNT output mnemonic.
GAPI	Average API for oil produced by a group of wells when API tracking enabled.
FAPI	Average API for oil produced by all production wells when API tracking enabled
RAPI	Average API for oil in place by region when API tracking enabled.
SUMTHIN	Reduces the amount of data written to summary files.
CTPR	tracer production rate for well connection.
CTIR	tracer injection rate for well connection.
CTPC	tracer production concentration for well connection.
CTIC	tracer injection concentration for well connection.

SCHEDULE section

The new SCHEDULE keywords are:

Table G.36 SCHEDULE Keywords

Keyword	Description
GSSCPTST	This allows the gas production rate which a group of wells is capable of maintaining for a period of time to be determined. It can also determine the period of time for which a gas production rate is sustainable.
WSEGLINK	Defines the chord segment links for specifying looped flowpaths
RIVTRACE	Allow tracer injection for rivers
RIVSALT	Allow salt injection for rivers
RIVDEBUG	Controls output of debug information for rivers
RIVRPROP	Allows update of river reach data in SCHEDULE section
WELEVNT	Sets the value of the SUMMARY section mnemonic WPWEM
DYNAMICR	This keyword defines regions based on when a set of conditions are satisfied. These regions may be static or allowed to change dynamically.
ENDDYN	This keyword indicates the end of a DYNAMICR keyword sequence.

New keywords in ECLIPSE 300

RUNSPEC section

The new RUNSPEC keywords are:

Table G.37 RUNSPEC keywords

Keyword	Description
DYNRDIMS	This keyword is used to set up dimensions for the Dynamic Region facility.
H2SSOL	Activates H2S solubility in water.
GRAVDRB	Vertical discretised gravity drainage model.
H2SSOL	Activates the H2S solubility option.
TRPLPORO	Specify a triple porosity calculation.
UDADIMS	Dimensions for User Defined Arguments
VISAGE	Activates the VISAGE geomechanics workflow

GRID section

The new GRID section keywords are:

Table G.38 GRID keywords

Keyword	Description
JFUNCR	Specification of Leverett J-Functions for the scaling of capillary pressure on a per saturation table basis.
ROCKSPLV	The matrix and fracture rock volume weights for dual porosity thermal calculations.

PROPS section

The new PROPS section keywords are;

Table G.39 PROPS keywords

Keyword	Description
ASPP1P	Specify the variable the asphaltene precipitation is set against. Must be used in conjunction with ASPREWG.
ASPP2P	Specify the variables the asphaltene precipitation is set against. Must be used in conjunction with ASPPW2D.
ASPPW2D	Specify the 2D table of asphaltene precipitation against the variables chosen with ASPP2P.
ASPLCRT	Control the asphaltene plugging by setting a critical volume of asphaltene deposit and a critical volumetric concentration for the flocs.

Table G.39 PROPS keywords

Keyword	Description
ASPKROW	Specify the oil-wet relative permeability for the wettability change due to asphaltene deposition. Must be used in conjunction with ASPWETF.
ASPWETF	Specify the weighting factor in the wettability transition due to asphaltene deposition as a function of the volume fraction of asphaltene deposit. Must be used in conjunction with ASPKROW.
BAKER1	Three phase relative permeability to be calculated using Baker's general formulation
BAKER2	Three phase relative permeability to be calculated using Baker's simplified formulation
DIFFCOG	Cross phase diffusion oil-gas
DIFFCGO	Cross phase diffusion gas-oil
DIFFCGW	Cross phase diffusion gas-water (CO2STORE or GASWAT option)
DIFFCWG	Cross phase diffusion water-gas (CO2STORE or GASWAT option)
ENKRVC	Relative permeabilities at saturation end points versus composition
ENPCVC	Capillary pressures at saturation end points versus composition
ENPTVC	Saturation end points versus composition
EPSCOMP	Specifies the component with which to perform compositional dependent end-point scaling
KRGDI	Gas relative permeability damage initiation
KRODI	Oil relative permeability damage initiation
KRWDI	Water relative permeability damage initiation
KRGDM	Gas relative permeability damage multipliers
KRODM	Oil relative permeability damage multipliers
KRWDM	Water relative permeability damage multipliers
KVMF	Interpolation mole fractions for composition dependent K-values
KVCOMP	Interpolation component for composition dependent K-values
KVTABCn	K-value tables for composition dependent K-values
KVCRWAT	Water K-value correlation (THERMAL option)
OPTODD3P	Options for control of the ODD3P three-phase relative permeability and capillary pressure calculation
OILVTIM	Interpolation method for OILVISCT data
REACLIMS	The pressure and temperature values used to calculate chemical reaction rates can now be limited.
REACOPS	Specify options specific to the chemical reactions model including convergence criteria
RKTRMDIR	Allows directional transmissibility multipliers to be used.
SOLUCO2H	CO2 solubility versus pressure and H2S concentration.
SOLUH2SC	H2S solubility versus pressure and CO2 concentration.
SORMW1	Residual oil saturation for compositional runs can be set using SORMW1 keyword in conjunction with ENDSCALE keyword.

Table G.39 PROPS keywords

Keyword	Description
STONE1EX	Optional exponent applied to the saturation terms used in the STONE1 three-phase relative permeability model to calculate the oil relative permeability.
THSVC	Steam viscosity coefficients (THERMAL option)
THWVC	Water viscosity coefficients (THERMAL option)
WAGHYSTR	Now allows WAG Hysteresis model to be used.

SOLUTION section

The new SOLUTION section keywords are:

Table G.40 SOLUTION section keywords

Keyword	Description
AQANCONL	This allows analytic aquifers to be connected to local grid refinements.
AQSTREAW	Set composition of multi-component water analytic aquifer.
DYNAMICR	This keyword defines regions based on when a set of conditions are satisfied. These regions may be static or allowed to change dynamically.
ENDDYN	This keyword indicates the end of a DYNAMICR keyword sequence.
VISDATES	Specify VISAGE geomechanics stress step dates
VISOPTS	Specify VISAGE geomechanics workflow options

SUMMARY section

The new SUMMARY keywords are:

Table G.41 SUMMARY keywords

Keyword	Description
AACMR	Component molar influx rates for analytic multi-component aquifers.
AACMT	Component molar influx totals for analytic multi-component aquifers.
ANCMR	Component molar influx rates for numerical multi-component aquifers.
ANCMT	Component molar influx totals for numerical multi-component aquifers.
BAQPH	Aqueous pH level (CO2STORE option only);
BAQSP	Molar densities of ions in the aqueous phase (CO2STORE option only)
CGLR	Gas-liquid ratio for connections.
FAMR	Rate that make-up gas is added to gas produced by the field for reinjection.
FAMT	Total make-up gas added to gas produced by the field for reinjection.
FGCDI	CO_2 trapped (immobile) in gas phase (CO2STORE option only)

Table G.41 SUMMARY keywords

Keyword	Description
FGCDM	CO_2 mobile in gas phase (CO2STORE option only)
FGLR	Gas-liquid ratio for the field.
FGSPR	Target sustainable rate for most recent sustainable capacity test for field.
FGSRL	Maximum sustainable rate found for the most recent sustainable capacity test for field.
FGSRU	Minimum un-sustainable rate found for the most recent sustainable capacity test for field.
FGSSP	Period for which target sustainable rate could be maintained for the most recent sustainable capacity test for field.
FGSTP	Test period for the most recent sustainable capacity test for field.
FWCD	CO_2 dissolved in water phase (CO2STORE option only)
FWGPRH	Observed wet gas production rate for history matching.
GAMR	Rate that make-up gas is added to gas produced by this group for reinjection.
GAMT	Total make-up gas added to gas produced by this group for reinjection.
GGLR	Gas-liquid ratio for groups.
GGSPR	Target sustainable rate for most recent sustainable capacity test for group.
GGSRL	Maximum sustainable rate found for the most recent sustainable capacity test for group.
GGSRU	Minimum un-sustainable rate found for the most recent sustainable capacity test for group.
GGSSP	Period for which target sustainable rate could be maintained for the most recent sustainable capacity test for group.
GGSTP	Test period for the most recent sustainable capacity test for group.
GWGPRH	Observed wet gas production rate for history matching.
MAXDE	Maximum change in energy density for each timestep.
MAXDT	Maximum change in temperature for each timestep.
MLINEARP	Number of pressure iterations for each timestep (JALS solver only).
MSUMLINP	Total number of pressure iterations since the start of the run (JALS solver only).
NCPRLINS	Average number of pressure iterations per linear iteration for each timestep (JALS solver only).
NLINEARP	Average number of pressure iterations per Newton iteration for each timestep (JALS solver only).
OFM	Produce output files of summary data for the OFM program.
RGCDI	CO2 trapped (immobile) in gas phase (CO2STORE option only)
RGCDM	CO2 mobile in gas phase (CO2STORE option only)
ROPR	Oil production rate
ROPT	Oil production total

Table G.41 SUMMARY keywords

Keyword	Description
RWCD	CO_2 dissolved in water phase (CO2STORE option only)
RWPR	Water production rate
RWPT	Water production total
RGPR	Gas production rate
RGPT	Gas production total
RNLPR	NGL Production Rate
RNLPT	NGL Production Total
RNLFR	Inter-region NGL Flow Rate
RNLFT	Inter-region NGL Flow Total
WCGIR	Component Gas Injection Rate
WCGPR	Component Gas Production Rate
WCOPR	Component Oil Production Rate
WWGPRH	Observed wet gas production rate for history matching.
FGPRG	Gas production rate (production summed then separated)
FOPRG	Oil production rate (production summed then separated)
FNLPRG	NGL production rate (production summed then separated)
FXMFG	Liquid mole fraction (production summed then separated)
FYMFG	Vapor mole fraction (production summed then separated)
FCOMRG	Component molar rates in the oil phase (production summed then separated)
FCGMRG	Component molar rates in the gas phase (production summed then separated)
FCNMRG	Component molar rates in the NGL phase (production summed then separated)
GGPRG	Gas production rate (production summed then separated)
GOPRG	Oil production rate (production summed then separated)
GNLPRG	NGL production rate (production summed then separated)
GXMFG	Liquid mole fraction (production summed then separated)
GYMFG	Vapor mole fraction (production summed then separated)
GCOMRG	Component molar rates in the oil phase (production summed then separated)
GCGMRG	Component molar rates in the gas phase (production summed then separated)
GCNMRG	Component molar rates in the NGL phase (production summed then separated)
WGLR	Gas-liquid ratio for wells.
WNEWTON	Number of well Newton iterations used in the last global Newton iteration at each timestep. A negative value indicates that the well model did not converge during the last global Newton iteration.

Table G.41 SUMMARY keywords

Keyword	Description
WPWE0	Well drilled indicator.
WPWE1	Connection opened indicator.
WPWE2	Connection shut indicator.
WPWE3	Connection shut to bottom indicator.
WPWE4	Well stopped indicator.
WPWE5	Injector set to producer indicator.
WPWE6	Producer set to injector indicator.
WPWE7	Well shut indicator.
WPWEM	WELEVNT output mnemonic.
SUMTHIN	Reduces the amount of data written to summary files.
NTS	Number of timesteps taken.
NTSPCL	Number of timesteps in which the pressure solution variable converged last.
NTSMCL	Number of timesteps in which the molar density solution variables converged last.
NTSECL	Number of timesteps in which the energy density solution variable converged last (THERMAL option)
NLRESSUM	Sum of the non-linear residual
NLRESMAX	Maximum element of the non-linear residual
FNLPR	Field NGL production rate
GNLPR	Group NGL production rate
WNLPR	Well NGL production rate
TUNE	Adds the NTS, NTSPCL, NTSMCL, NTSECL, NLRESSUM and NLRESMAX mnemonics to the SUMMARY section.
BGTRP	Trapped gas saturation (WAG hysteresis only).
BGTPD	Dynamic trapped gas saturation (WAG hysteresis only).
BSWL	Minimum water saturation.
BSWU	Maximum water saturation.
BSWCR	Critical water saturation.
BSGL	Minimum gas saturation.
BSGU	Maximum gas saturation.
BSGCR	Critical gas saturation.
BSOWC	Critical oil-in-water saturation.
BSOGC	Critical oil-in-gas saturation.
BSMF	(CO2STORE with SOLID option only) Solid component mole fractions

Table G.41 SUMMARY keywords

Keyword	Description
BIFTWO	Water-oil interfacial tension (alias for BIFTOW and only available for the ODD3P relative permeability and capillary pressure hysteresis model)
BIFTOG	Oil-gas interfacial tension (alias for BIFTGO and only available for the ODD3P relative permeability and capillary pressure hysteresis model)
BIFTWG	Water-gas interfacial tension (alias for BIFTGW and only available for the ODD3P relative permeability and capillary pressure hysteresis model)

SCHEDULE section

The new SCHEDULE section keywords are:

Table G.42 SCHEDULE keyword

Keyword	Description
AQSTREAW	Set composition of multi-component water analytic aquifer.
DATES	A fourth argument has been added to this keyword for the time of the report.
DELAYACT	Now allows to initiate a set of keywords to be processed a certain time after another ACTION is triggered.
DYNAMICR	This keyword defines regions based on when a set of conditions are satisfied. These regions may be static or allowed to change dynamically.
ENDDYN	This keyword indicates the end of a DYNAMICR keyword sequence.
GSEPREPT	This specifies separator conditions for the group level separation production rate and composition summary vector calculations.
GSSCPTST	This allows the gas production rate which a group of wells is capable of maintaining for a period of time to be determined. It can also determine the period of time for which a gas production rate is sustainable.
WELEVNT	Sets the value of the SUMMARY section mnemonic WPWEM

Altered keywords in ECLIPSE 100

RUNSPEC section

The altered RUNSPEC keywords are:

Table G.43 Altered RUNSPEC keywords

Keyword	Description
UDQDIMS	Aquifer and Block options have been added to allow aquifer and block user defined quantities to be specified.
WSEGDIMS	A fourth item has been added to specify the maximum number of chord segment links per well.
PETOPTS	The INITNNC mnemonic has been extended and now also applies to CORSNUM information. If a grid file is also being output, INITNNC adds the requested information to the INIT file but does not suppress its output to the grid file. The NOPINCHC mnemonic has been added to suppress the calculation of pinchout NNCs.
START	A fourth argument has been added for the time at the start of the run.

GRID section

The altered GRID keywords are:

Table G.44 Altered GRID keywords

Keyword	Description
PETGRID	This keyword can now be used to import global grid geometry, local grid geometry and properties from a Generic Simulation Grid (.GSG) file. This type of file format has to be imported via Petrel. If this keyword is present in the dataset, no grid file will be output.
RPTGRID	This keyword has been extended to include the reporting of the JFUNCR keyword.
EQUALREG	This keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
ADDREG	This keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
MULTIREG	This keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.

Table G.44 Altered GRID keywords

Keyword	Description
COPYREG	This keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
MULTREGT	This keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM) which will be used to multiply the transmissibilities.
MULTREGD	This keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM) which will be used to multiply the diffusivities.
MULTREGH	This keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM) which will be used to multiply the rock thermal conductivities.
MULTREGP	This keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM) which will be used to multiply the pore volumes.
NMATOPTS	New option VERTICAL to support discretised matrix gravity drainage.

EDIT section

The altered EDIT keywords are:

Table G.45 Altered EDIT keywords

Keyword	Description
EQUALREG	This keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
ADDREG	This keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
MULTIREG	This keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
COPYREG	This keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
MULTREGT	This keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM) which will be used to multiply the transmissibilities.
MULTREGD	This keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM) which will be used to multiply the diffusivities.
MULTREGH	This keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM) which will be used to multiply the rock thermal conductivities.
MULTREGP	This keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM) which will be used to multiply the pore volumes.

PROPS section

The altered PROPS keywords are:

Table G.46 Altered PROPS keywords

Keyword	Description
RPTPROPS	This keyword has been extended to include the reporting of the Stone 1 exponent values specified using the STONE1EX keyword.
EQUALREG	This keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
ADDREG	This keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
MULTIREG	This keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
COPYREG	This keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.

REGIONS section

The altered REGIONS keywords are

Table G.47 Altered REGIONS keywords

Keyword	Description
EQUALREG	This keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
ADDREG	This keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
MULTIREG	This keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
COPYREG	This keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.

SOLUTION section

The altered SOLUTION keywords are:

Table G.48 Altered SOLUTION keywords

Keyword	Description
AQUCHWAT	New item 13 has been added to allow the depth term in the aquifer inflow equation to be removed from the calculation.
RPTRST	The new CELLINDX mnemonic outputs the cell indices corresponding to the other output properties. If any cells are missing due to inactivity this allows determination of which cells properties are actually present in the restart file without the need for information in the INIT file.
RPTRST	The DYNREG mnemonic has been added. This mnemonic outputs the cell indices corresponding to the dynamic regions specified using the DYNAMICR keyword.
RPTRST	The SFIP mnemonic has been added. This is a synonym for the FIP mnemonic and is intended to reduce possible confusion when using FIP in both ECLIPSE 100 and ECLIPSE 300, to indicate that for ECLIPSE 100 surface/separator conditions are used
RPTRST	The RFIP mnemonic has been added to output fluid in place at reservoir conditions.
RPTSOL	The FOAMCNM mnemonic has been added to output the logarithm base ten of the capillary number for the FOAM option when the functional form of the gas mobility reduction model has been selected and the FOAMFCN keyword has been specified.
EQUALREG	This keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
ADDREG	This keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
MULTIREG	This keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
COPYREG	This keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
RIVERSYS	The RIVERSYS keyword has been extended to include an option of solving the time-dependent Saint-Venant equations.
AQUCHWAT	An extra item has been added to the AQUCHWAT keyword to allow the user to impose the minimum and maximum flow rates before or after the flow rate has been scaled by the grid block saturation in cases when water is flowing into aquifer from grid.

SUMMARY section

The altered SUMMARY keywords are

Table G.49 Altered SUMMARY keywords

Keyword	Description
AAQP	This mnemonic can now be used to output aquifer pressure for all aquifer types (except for constant flux aquifer as given by AQUFLUX)

SCHEDULE section

The altered SCHEDULE keywords are:

Table G.50 Altered SCHEDULE keywords

Keyword	Description
ACTIONX	A well list name may be specified to trigger actions in the ACTIONX keyword.
ACTIONX	Aquifer and Block mnemonics may be used to trigger actions in the ACTIONX keyword.
AQUCHWAT	New item 13 has been added to allow the depth term in the aquifer inflow equation to be removed from the calculation.
COMPSEGL	A new item 12 has been added to records 2 and upwards. This allows the segment to which each grid block completion is allocated to be specified explicitly.
COMPSEGS	A new item 11 has been added to records 2 and upwards. This allows the segment to which each grid block completion is allocated to be specified explicitly.
DATES	A fourth argument has been added to this keyword for the time of the report.
RIVERSYS	The RIVERSYS keyword has been extended to include an option of solving the time-dependent Saint-Venant equations.
RPTRST	The new CELLINDX mnemonic outputs the cell indices corresponding to the other output properties. If any cells are missing due to inactivity this allows determination of which cells properties are actually present in the restart file without the need for information in the INIT file.
RPTRST	The DYNREG mnemonic has been added. This mnemonic outputs the cell indices corresponding to the dynamic regions specified using the DYNAMICR keyword.
RPTRST	The SFIP mnemonic has been added. This is a synonym for the FIP mnemonic and is intended to reduce possible confusion when using FIP in both ECLIPSE 100 and ECLIPSE 300, to indicate that for ECLIPSE 100 surface/separator conditions are used
RPTRST	The RFIP mnemonic has been added to output fluid in place at reservoir conditions.

Table G.50 Altered SCHEDULE keywords

Keyword	Description
RPTSCHED	The VFPPROD mnemonic now produces tables comparing maximum dimensions from the VFPPDIMS keyword to those observed in the input tables. This output is produced for VFPPROD=2 or VFPPROD=3
RPTSCHED	The FOAMCNM mnemonic has been added to output the logarithm base ten of the capillary number for the FOAM option when the functional form of the gas mobility reduction model has been selected and the FOAMFCN keyword has been specified.
UDQ	Aquifer and Block user defined quantities can be defined.
WSCTAB	Scale deposition now applies to both injecting and producing connections instead of only producing connections.
WFOAM	This keyword may now be used with water injectors if the foam transport phase has been specified as water using the FOAMOPTS keyword.
MULTREGT	This keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM) which will be used to multiply the transmissibilities.
AQUCHWAT	An extra item has been added to the AQUCHWAT keyword to allow the user to impose the minimum and maximum flow rates before or after the flow rate has been scaled by the grid block saturation in cases when water is flowing into the aquifer from the grid.
WLIFTOPT	New item 6 to permit the inclusion of a weighted gas rate production term in the denominator of the gas lift optimization gradient calculation. New item 7 to permit wells to continue to be allocated lift gas after a group gas target has been achieved but whilst an oil rate limit has not been reached.

OPTIONS keyword

The altered OPTIONS are:

Table G.51 Altered OPTIONS

Switch	Description
Switch **74**	This controls aspects of the mixed wettability rocks when the SWATINIT keyword is used to match an initial water distribution. When the SWATINIT keyword is used to match an initial water distribution for mixed wettability rocks and positive capillary pressure dominated then the scaling on P_{cow} is more sensible if OPTIONS(74)=5. When the SWATINIT keyword is used to match an initial water distribution for mixed wettability rocks and negative capillary pressure dominated then the scaling on P_{cow} is more sensible if OPTIONS(74)=7.
Switch **108**	This controls aspects of the scale deposition behavior. If set to a non-zero value this restores pre-2008.1 behavior: injecting connections will not be subject to scale damage.
Switch **109**	If set to a non-zero value, the pre-2007.1 behavior of the RPTONLY and RPTONLYO keywords in the SUMMARY section of restarted runs is restored. At 2007.1 this behavior was changed so that these keywords were ignored if they were present in the SUMMARY section in restarted runs, and they had to be re-specified at or after the restart time in the SCHEDULE section.
Switch **110**	If set to a non-zero value this restores pre-2007.3 behavior for the AQUCHWAT aquifer water density calculation
Switch **111**	If set to a non-zero value this restores pre-2008.1 behavior for the WTEST keyword. This prevents the reset of WTEST if a group of production wells are closed by a single event
Switch **112**	If set to 1 this restores pre-2008.1 behavior for the RADFIN4 keyword. The new behavior ensures the correct inheritance of porosity and calculation of local grid pore volumes for the case where THETA=4.
Switch **113**	If set to 1 this restores pre-2008.1 behavior for API tracking in local grid refinements. The new behavior calculates oil density in global grid cells from local grid cells using an oil-in-place weighting.
Switch **114**	If set to a non-zero value, an improved water and solvent saturation bound-checking will be performed.
Switch **115**	If set to a non-zero value the pre-2008.1 behavior of analytic aquifer inflow equations is used. The 2008.1 behavior correctly calculates the flow from the reservoir to the aquifer in the case of connate water/gas. This change affects all analytical aquifers.
Switch **116**	Affects models with the POLYMER or FOAM option. If set to one it restores the 2008.1 behavior.
Switch **117**	
Switch **118**	If set to a non-zero value pre-2008.2 calculation behavior for the use of the DRSDT and DRVDT keywords in the presence of both vaporized oil and dissolved gas is restored.
Switch **119**	Reserved for future use.

Table G.51 Altered OPTIONS

Switch	Description
Switch **120**	If set to 1 the pre-2008.2 convergence criterion for the non-cascade solver used for tracer tracking is restored.
Switch **121**	If set to a non-zero value pre-2008.2 calculation behavior for the calculation of relative permeabilities using Stone's Second Model with end-point scaling is restored.
Switch **122**	If set to one it restores the pre-2008.2 Polymer Dead Pore Volume model

DEBUG keyword

The altered switches for the `DEBUG` keyword are:

Table G.52 Altered DEBUG switches

Switch	Description
Switch **76**	Outputs debug from the river model

Altered keywords in ECLIPSE 300

RUNSPEC section

The altered `RUNSPEC` keywords are:

Table G.53 Altered RUNSPEC keywords in ECLIPSE 300

Keyword	Description
`ACTPARAM`	First argument can be defined
`ASPHALTE`	Argument handling has been improved. Permeability and viscosity changes can be skipped by default or by setting the 2nd and 3rd argument to 'NO' respectively.
`UDQDIMS`	Aquifer and Block options have been added to allow aquifer and block user defined quantities to be specified.
`PETOPTS`	The `INITNNC` mnemonic has been extended and now also applies to `CORSNUM` information. If a grid file is also being output, `INITNNC` adds the requested information to the INIT file but does not suppress its output to the grid file. The `NOPINCHC` mnemonic has been added to suppress the calculation of pinchout NNCs.
`START`	A fourth argument has been added for the time at the start of the run.

GRID section

The altered GRID keywords are:

Table G.54 Altered GRID keywords in ECLIPSE 300

Keyword	Description
BTOBALFA	Create block to block connections (single value multiplier); now available for LGRs
BTOBALFV	Create block to block connections (vector of multipliers); now available for LGRs
NMATOPTS	New option VERTICAL to support discretised matrix gravity drainage.
PETGRID	This keyword can now be used to import global grid geometry, local grid geometry and properties from a Generic Simulation Grid (.GSG) file. This type of file format has to be imported via Petrel. In addition, the root extension of the file to be imported must be provided - this applies for importing both.GSG files and.OPF files. If this keyword is present in the dataset, no grid file will be output.
RPTGRID	This keyword has been extended to include the reporting of the JFUNCR keyword.
EQUALREG	This keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
ADDREG	This keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
MULTIREG	This keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
COPYREG	This keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
MULTREGT	This keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM) which will be used to multiply the transmissibilities.
MULTREGP	This keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM) which will be used to multiply the pore volumes.
AQUCON	Item 11 is now supported

EDIT section

The altered EDIT keywords are:

Table G.55 Altered EDIT keywords in ECLIPSE 300

Keyword	Description
EQUALREG	This keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
ADDREG	This keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
MULTIREG	This keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
COPYREG	This keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
MULTREGT	This keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM)) which will be used to multiply the transmissibilities
MULTREGP	This keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM) which will be used to multiply the pore volumes.

PROPS section

The altered PROPS keywords are:

Table G.56 Altered PROPS keywords in ECLIPSE 300

Keyword	Description
EPSODD3P	Items 4 to 20 are now in use and new items 23 to 26 have been added. These provide control over surface tension based capillary pressure, relative permeability and saturation end-point scaling.
RPTPROPS	This keyword has been extended to include the reporting of the Stone 1 exponent values specified using the STONE1EX keyword.
EQUALREG	This keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
ADDREG	This keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
MULTIREG	This keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.

Table G.56 Altered PROPS keywords in ECLIPSE 300

Keyword	Description
COPYREG	This keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
EHYSTR	Item 12 allows a threshold saturation of Killough's hysteresis model for both wetting and non-wetting phases. This threshold allows better control of the numerical sensitivity of the system, preventing it from being too unstable.
ODD3P	The three-phase relative permeability and capillary pressure hysteresis model calculation has been improved. Pre-2008.2 behavior may be restored using the OPTODD3P keyword.

REGIONS section

The altered REGIONS keywords are:

Table G.57 Altered REGIONS keywords in ECLIPSE 300

Keyword	Description
EQUALREG	This keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property
ADDREG	This keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
MULTIREG	This keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
COPYREG	This keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.

SOLUTION section

The altered SOLUTION keywords are:

Table G.58 Altered SOLUTION keywords in ECLIPSE 300

Keyword	Description
ADDREG	This keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
AQSTREAM	For the GASWAT and CO2STORE options, this keyword can be used to specified the water composition of an AQUCHWAT, AQUCT, AQUFETP or AQUFLUX aquifer.
AQUANCON	Item 11 is now supported
AQUCHWAT	An extra item has been added to the AQUCHWAT keyword to allow the user to impose the minimum and maximum flow rates before or after the flow rate has been scaled by the grid block saturation in cases when water is flowing into aquifer from grid.
AQUCHWAT	New item 13 has been added to allow the depth term in the aquifer inflow equation to be removed from the calculation.
COPYREG	This keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
EQUALREG	This keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
MULTIREG	This keyword has been extended to allow the user to choose the region (MULTNUM, FLUXNUM or OPERNUM) which will be used to modify a specific property.
RPTRST	The new CELLINDX mnemonic outputs the cell indices corresponding to the other output properties. If any cells are missing due to inactivity this allows determination of which cells properties are actually present in the restart file without the need for information in the INIT file.
RPTRST	The DYNREG mnemonic has been added. This mnemonic outputs the cell indices corresponding to the dynamic regions specified using the DYNAMICR keyword.
RPTRST	The RFIP mnemonic has been added. This is a synonym for the FIP mnemonic and is intended to reduce possible confusion when using FIP in both ECLIPSE 100 and ECLIPSE 300, to indicate that for ECLIPSE 300 reservoir conditions are used. Note that this only allows for four significant characters of the component name to be added to the restart file, instead of the five for FIP.
RPTRST	The FIP (and also RFIP) mnemonic has been extended to output FIPGAS, FIPOIL and FIPWAT at reservoir conditions.
RPTRST	Three new mnemonics 'SFIPGAS', 'SFIPOIL' and 'SFIPWAT' have been added to output FIP data at separator conditions.

Table G.58 Altered SOLUTION keywords in ECLIPSE 300

Keyword	Description
RPTRST	The SFIP mnemonic has been added to output FIP data at separator conditions. Using the mnemonic will output SFIPGAS, SFIPOIL and SFIPWAT.
RPTRST	The mnemonic MWAT has been added to output the moles of water per reservoir volume.
RPTRST	The mnemonics KRGDM, KRODM and KRWDM have been added to output the damage multipliers generated using the KRGDM, KRODM and KRWDM keywords.
RPTRST	The mnemonics FLOOIL and FLOGAS for the RPTRST keyword can now be used to output inter-block flows for asphaltene models.
RPTRST	A new mnemonic SGTRAP has been added to output the trapped gas saturation.
RPTRST	New mnemonics SWL, SWU, SWCR, SGL, SGU, SGCR, SOWCR, SOGCR, have been added to output the scaled saturation table end-points, only in the case of temperature or composition dependent scaling (ENPTVT and ENPTVC keywords).
RPTRST	If flow output is requested, this now also includes NNC flows.
RPTSOL	New mnemonics SWL, SWU, SWCR, SGL, SGU, SGCR, SOWCR, SOGCR, have been added to output the scaled saturation table end-points, only in the case of temperature or composition dependent scaling (ENPTVT and ENPTVC keywords).
RPTSOL	A new mnemonic 'FIP' has been added to give fluid-in-place reports.
RPTSOL	The mnemonic MWAT has been added to output the moles of water per reservoir volume
RPTSOL	A new mnemonic AQANCONL has been added to output aquifer grid block connection data within local grid refinements.
RPTRST	The SMF mnemonic has been added to output the solid component mole fractions for CO2STORE and SOLID runs.

SUMMARY section

The altered SUMMARY mnemonics are:

Table G.59 Altered SUMMARY mnemonics

Mnemonic	Description
AAQP	This mnemonic can now be used to output aquifer pressure for all aquifer types (except for constant flux aquifer as given by AQUFLUX)
FCMIR	This is now reported with the correct sign.
FCMIT	This is now reported with the correct sign.
GCMIR	This is now reported with the correct sign.

Table G.59 Altered SUMMARY mnemonics

Mnemonic	Description
GCMIT	This is now reported with the correct sign.
WCMIR	This is now reported with the correct sign.
WCMIT	This is now reported with the correct sign.

SCHEDULE section

The altered SCHEDULE keywords are:

Table G.60 Altered SCHEDULE keywords in ECLIPSE 300

Keyword	Description
ACTIONX	Aquifer and Block mnemonics may be used to trigger actions in the ACTIONX keyword.
AQSTREAM	For the GASWAT and CO2STORE options, this keyword can be used to specified the water composition of an AQUCHWAT, AQUCT, AQUFETP or AQUFLUX aquifer.
AQUCHWAT	New item 13 has been added to allow the depth term in the aquifer inflow equation to be removed from the calculation.
AQUCHWAT	An extra item has been added to the AQUCHWAT keyword to allow the user to impose the minimum and maximum flow rates before or after the flow rate has been scaled by the grid block saturation in cases when water is flowing into aquifer from grid.
COMPSEGL	A new item 12 has been added to records 2 and upwards. This allows the segment to which each grid block completion is allocated to be specified explicitly.
COMPSEGS	A new item 11 has been added to records 2 and upwards. This allows the segment to which each grid block completion is allocated to be specified explicitly.
DATES	A fourth argument has been added to this keyword for the time of the report.
GADVANCE	Advance gas for a given source is now only added once to mixtures with contributions from multiple separators from the same source.
GINJGAS	This keyword now correctly respects the priority set by WTAKEGAS when separator stage gas is reinjected.
GRUPFUEL	This keyword now correctly respects the priority set by WTAKEGAS when separator stage gas is reinjected.
GRUPSALE	This keyword now correctly respects the priority set by WTAKEGAS when separator stage gas is reinjected.
MULTREGT	This keyword has been extended to allow the user to choose the region (MULTNUM or FLUXNUM) which will be used to multiply the transmissibilities.

Table G.60 Altered SCHEDULE keywords in ECLIPSE 300

Keyword	Description
PICOND	Two new items (items 5 and 6) have been added to the PICOND keyword to allow you to specify the lower and upper bounds for use in constructing the generalized pseudo-pressure integrals. These bounds are specified as dimensionless factors which are applied to the completion cell pressure to generate the lower and upper pressures for the table. They allow the use of smaller quadrature intervals by allowing control over the pressure range of the table. A further 4 new items (items 7, 8, 9 and 10) have been added to allow you to specify when a new integral table will be generated. Finally another 2 items (items 11 and 12) have been added to control the new adaptive quadrature feature, giving the generalized pseudo pressure table more detail in nonlinear areas and removing less useful points from the table.
QDRILL	If QDRILL is used with mixture reinjection (WINJORD or WINJMIX) new gas injection wells are now only drilled when required. This change may be reversed using item 139 of the OPTIONS3 keyword.
RPTRST	The mnemonics KRGDM, KRODM and KRWDM have been added to output the damage multipliers generated using the KRGDM, KRODM and KRWDM keywords.
RPTRST	The new CELLINDX mnemonic outputs the cell indices corresponding to the other output properties. If any cells are missing due to inactivity this allows determination of which cells properties are actually present in the restart file without the need for information in the INIT file
RPTRST	The DYNREG mnemonic has been added. This mnemonic outputs the cell indices corresponding to the dynamic regions specified using the DYNAMICR keyword.
RPTRST	The RFIP mnemonic has been added. This is a synonym for the FIP mnemonic and is intended to reduce possible confusion when using FIP in both ECLIPSE 100 and ECLIPSE 300, to indicate that for ECLIPSE 300 reservoir conditions are used. Note that this only allows for four significant characters of the component name to be added to the restart file, instead of the five for FIP.
RPTRST	The FIP (and also RFIP) mnemonic has been extended to output FIPGAS, FIPOIL and FIPWAT at reservoir conditions.
RPTRST	Three new mnemonics 'SFIPGAS', 'SFIPOIL' and 'SFIPWAT' have been added to output FIP data at separator conditions.
RPTRST	The SFIP mnemonic has been added to output FIP data at separator conditions. Using the mnemonic will output SFIPGAS, SFIPOIL and SFIPWAT.
RPTRST	The mnemonic SGTRAP has been added to output the trapped gas saturations.
RPTRST	The mnemonic MWAT has been added to output the moles of water per reservoir volume.

Table G.60 Altered SCHEDULE keywords in ECLIPSE 300

Keyword	Description
RPTRST	New mnemonics SWL, SWU, SWCR, SGL, SGU, SGCR, SOWCR, SOGCR, have been added to output the scaled saturation table end-points, only in the case of temperature or composition dependent scaling (ENPTVT and ENPTVC keywords).
RPTRST	If flow output is requested, this now also includes NNC flows.
RPTSCHED	New mnemonics SWL, SWU, SWCR, SGL, SGU, SGCR, SOWCR, SOGCR, have been added to output the scaled saturation table end-points, only in the case of temperature or composition dependent scaling (ENPTVT and ENPTVC keywords).
RPTSCHED	The mnemonic MWAT has been added to output the moles of water per reservoir volume.
UDQ	Aquifer and Block user defined quantities can be defined.
WADVANCE	Advance gas for a given source is now only added once to mixtures with contributions from multiple separators from the same source.
WCONHIST	History matching production wells may now have wet gas rate as a control mode by setting item 3 of this keyword as WGRA.
WELLWAG	WAG wells may now be controlled by surface rate, BHP or THP. Separate VFP tables may also be set for each phase injected.
WELSPECS	The default behavior of item 5 (reference depth for bottom hole pressure) has been altered to take the depth of the center of the first completed cell as defined by COMPORD.
WINJORD	Advance gas for a given source is now only added once to mixtures with contributions from multiple separators from the same source.
WLIFTOPT	New item 6 to permit the inclusion of a weighted gas rate production term in the denominator of the gas lift optimization gradient calculation. New item 7 to permit wells to continue to be allocated lift gas after a group gas target has been achieved but whilst an oil rate limit has not been reached.
WLIST	List of mnemonics that work with WLIST has been increased The mnemonics FLOOIL and FLOGAS for the RPTRST keyword can now be used to output inter-block flows for asphaltene models.
WSCTAB	Scale deposition now applies to both injecting and producing connections instead of only producing connections.
WTADD	LIFT has been added to the list available for item 2 to allow the artificial lift quantity value to be modified.
WTMULT	LIFT has been added to the list available for item 2 to allow the artificial lift quantity value to be modified.
RPTRST	The SMF mnemonic has been added to output the solid component mole fractions for CO2STORE and SOLID runs.
RPTSCHED	The SMF mnemonic has been added to output the solid component mole fractions for CO2STORE and SOLID runs.

OPTIONS3 keyword

The altered OPTIONS3 switches are:

Table G.61 Altered OPTIONS3 switches

Switch	Description
Switch **95**	This switch controls the scale back of sales and fuel gas should insufficient gas be available to meet sales, fuel and reinjection targets. If set to 0 fuel and sales gas are scaled back equally only if WAVAILIM is set. If set to 3 fuel and sales gas are scaled back equally irrespective of WAVAILIM. If set to 4 the priority order specified in WTAKEGAS is used to decide which of the fuel and sales to scale back first, but no scale back takes place if WAVAILIM has not been specified.
Switch **119**	If set to 1, the water properties of water analytic aquifers are calculated using the PVT tables for the GASWAT, CO2STORE, CO2SOL or COMPW options. For GASWAT or CO2STORE, this was the pre-2007.1 method. By default, the water properties are currently calculated using the EOS. For the multi-component water option COMPW), this was the pre-2008.1 method. By default, the water properties are currently calculated using the water components data. For the CO2SOL option, this was the pre-2008.1 method. By default, the water properties are currently calculated using the solubility table.
Switch **124**	If set to a positive number, a better hysteresis model for water capillary pressure is activated
Switch **127**	If set to a non-zero value the block-by-block inter-region flowing fractions calculation for the table GPTABLEN look up is activated. By default, the inter-region flowing fractions are calculated using the region-averaged values.
Switch **128**	If set to a non-zero value the pre-2007.2 tolerances used to calculate the derivatives of immobile fluids are restored. This can lead to large elements in the Jacobian and affect the convergence of the solution.
Switch **129**	This controls aspects of the scale deposition behavior. If set to a non-zero value this restores pre-2008.1 behavior: injecting connections will not be subject to scale damage.
Switch **130**	If set to a non-zero value, the pre-2004A behavior for gas accounting is used when there is re-injection of a mixture fluid (that is, one defined using the WINJMIX or WINJORD keywords) in which a mixture item comes from a separator stage.
Switch **131**	If set to 1, the pre-2007.2 behavior for water analytic aquifers for the GASWAT or CO2STORE, options is used where the water leaving / entering the reservoir is made up entirely of the “H2O” component. For the multi-component water option (COMPW), if set to 1, the pre-2008.1 behavior is used where the water leaving / entering the reservoir is made up entirely of the last water component. For the CO2SOL option, if set to 1, the pre-2008.1 behavior is used where the water leaving / entering the reservoir is made up entirely of the “WATER” component.

Table G.61 Altered OPTIONS3 switches

Switch	Description
Switch **132**	If non-zero and divisible by 2 then pre-2007.2 behavior of generalized pseudo-pressure integration calculation is used. If non-zero and divisible by 3 then the blocking factor denominator mobility value is calculated in the same way as the integral mobility value. The default behavior takes this value from the bulk cell properties. If non-zero and divisible by 5 then pre-2008.1 behavior of generalized pseudo-pressure calculation is used. This changes the behavior of velocity dependent relative permeability effects in for pressures below the connection pressure and above the well drainage radius pressure at the time the integral is calculated. This only affects the calculation if capillary number model 1 is used: see `VELDEP`.
Switch **133**	If set to 1 the residual in the WARP linear solver is calculated with greater numerical accuracy.
Switch **134**	If set to 1 the flows output by the `RPTRST`, `RPTSCHED` and `RPTSOL` keywords are calculated at the start of the timestep rather than the end. This was the pre-2008.1 behavior.
Switch **135**	If set to 1 an alternative implementation for predicting the energy density in the `HEATER` algorithm is activated.
Switch **136**	If set to a nonzero value apply restrictive well-reservoir convergence check.
Switch **137**	If set to a nonzero value do an extra flash at the end of an accepted time step (updates the fluid properties for reporting).
Switch **138**	Saturation chops, in EOS mode, have been modified. The components are now ordered by the critical temperature in order to determine the most volatile components. The pre-2008.1 behavior may be recovered using this item.
Switch **139**	If non-zero and divisible by 2 then the pre-2008.1 well reinjection calculation is used. If non-zero and divisible by 3 then the pre-2008.1 calculation of advance gas with `WINJORD` or `WINJMIX` is used. If non-zero and divisible by 5 then the pre-2008.1 calculation of reinjection, fuel and sales gas on different levels of the group structure is recovered. If non-zero and divisible by 7 then the pre-2008.1 calculation of the reinjection target for partial reinjection of mixture gas (`WINJORD` or `WINJMIX`) is applied. If non-zero and divisible by 11 then the pre-2008.1 behavior of `QDRILL` with `WINJORD` or `WINJMIX` is used.
Switch **140**	If set to a non-zero value the pre-2007.2 calculation for the lumped tracer connection phase flow, which does not include a Li correlation factor in the wellbore flash, will be used
Switch **141**	If non-zero the pre-2007.2 segregated well head calculation is used for `BLACKOIL` models
Switch **142**	If set to 1 the pre-2008.1 behavior of `WELSPECS` element 5 is used. If set to 2 then the 2008.1 behavior is used.
Switch **143**	This option allows the scaling of the water-oil capillary pressure curves for oil-wet, water-wet and mixed wet cases such that water distribution defined by `SWATINIT` is honored in the equilibrated initial solution.

Table G.61 Altered OPTIONS3 switches

Switch	Description
Switch **144**	If set to 1 any errors resulting from checking the Z-factors are down graded to warnings.
Switch **145**	If set to 1 the pre-2008.1 handling of capillary pressure in analytic aquifers will be used.
Switch **146**	Switch to activate one phase selection method.
Switch **148**	If set to 1 the pre-2008.1 handling of the `SWATINIT` calculation will be used.
Switch **149**	If set to 1 the pre-2008.1 calculation of pressure equivalent radius for grid blocks at the center of radial grids is used.
Switch **150**	If set to 1 the pre-2008.1 behavior of `WTEST` after `GECON` shuts a group of production wells is used.
Switch **151**	If set to 1, only the gas phase of a `GASWAT` or `CO2STORE` run is considered as hydrocarbon. This affects the calculation of the summary keywords `FPR`, `FPRH` and `FPPO`. By default, the water and gas phases are considered as hydrocarbon.
Switch **152**	If set to 1 a minimum timestep that fails to converge is re-solved so that the solution accepted is the closest rather than just the last Newton iteration. This option will require more Newton iterations but can help a simulation through a very poor period of convergence.
Switch **153**	An improved vapor versus liquid density check is performed. If set to a non-zero value, the pre-2008.1 behavior will be restored.
Switch **154**	If set to a non-zero value the pre-2008.1 behavior of analytic aquifer inflow equations is used. The 2008.1 behavior correctly calculates the flow from the reservoir to the aquifer in the case of connate water/gas. This change affects all analytical aquifers.
Switch **155**	If set to a non-zero value, the pre-2008.1 behavior for tilted aquifer block equilibration is restored. The 2008.1 equilibration behavior uses the aquifer block properties instead of the tilted block geometrical properties for numerical aquifers associated with grid blocks of zero thickness.
Switch **156**	If set to a non-zero value, the pre-2008.1 behavior for tilted block equilibration is restored. The 2008.1 equilibration behavior corrects the weighted sum calculation for tilted grid blocks, defined using corner point geometry, by adjusting the averaged inclination factors to prevent the planes representing the top and bottom faces based on these averaged inclination factors from intersecting within the grid block. This correction will typically apply to distorted grid blocks, for example, those for which one or more corners share the same or similar depth values for both the upper and lower faces.
Switch **157**	If set to 1 an alternative implementation for water density calculation algorithm is activated.
Switch **158**	If set to 1 the Appleyard chop is applied to components when they appear or disappear. This behavior is the default for simulations using the chemical reactions feature but can be disabled by setting to -1.

Table G.61 Altered OPTIONS3 switches

Switch	Description
Switch **159**	If set to a non-zero value the pre-2008.2 behavior for gas accounting is restored according to the value set. If the value is divisible by 2 and if a well is set to reinject a group vapor source or a group is set to reinject a well vapor source (for example `GINJGAS` item 2 set to WV) then no gas will be reinjected. If the value is divisible by 3 then make-up gas is not switched off if a reinjection target is set for a group. If the value is divisible by 5 then the pre-2008.2 calculation of advance gas is used. This changes the calculation of advance gas if the `WEFAC` and/or `GEFAC` keywords are used. If the value is divisible by 7 then the pre-2008.2 calculation of availability limit is used. This changes the calculation of the availability limit if the `WEFAC` and/or `GEFAC` keywords are used. If the value is divisible by 11 then the pre-2008.2 calculation of gas accounting is used. This changes the calculation of excess gas and reinjected gas if the `GEFAC` keyword is used.
Switch **160**	If set to 1, the pre-2008.2 thermal non-linear damping is restored.
Switch **161**	Not used.
Switch **162**	If set to 1, the pre-2008.1 behavior for GOC and datum depth is restored.
Switch **163**	If set to 1, an alternative memory allocation algorithm is enabled. It reduces the number of allocations/de-allocations while increasing the maximum amount of memory used.
Switch **164**	Not used.
Switch **165**	Not used.
Switch **166**	Not used.
Switch **167**	If set to > 0 the pre-2008.2 behavior of the flash can be restored with regard to single-phase splitting attempt for Gibbs free energy minimization in flash.
Switch **168**	If set to 1 the pre-2008.2 calculation of fluid in place is used, correctly accounting for satellite group injection and production fluids.

OPTIMIZE section

The altered `OPTIMIZE` keywords are:

Table G.62 Altered OPTIMIZE keywords

Keyword	Description
`OPTPARS`	An extra item has been added to the `OPTPARS` keyword to allow the specification of the well control mode which should be used with each `SCSA` parameter.

New developments in 2007.1

New facilities in ECLIPSE 100

RUNSPEC section

- The GDIMS keyword has been added to specify the number of parameters required for the instantaneous gradient options.
- The memory management in ECLIPSE 100 has been changed to improve the handling of large models on 64 bit machines. The BIGMODEL keyword is now redundant and will be ignored. The first item of the MEMORY keyword is also redundant and will be ignored.
- The UDQDIMS and UDQPARAM keywords have been added to specify the dimensions and parameters associated with the user defined summary quantity facility.
- The LGRCOPY keyword has been added to allow modification of values within an LGR when the values have only been set for global cells. Used in the RUNSPEC section means that it will apply to all LGRs in the model
- PSTEADY has been modified (data item 11) to add lower limit for time-stepping to ensure calculation does not end too soon.
- The RIVRDIMS keyword has been added to define river dimension data.

Aquifers

- OPTIONS(99) switch generates warning messages from the AQUANCON keyword when cells cannot be connected to the aquifer because there is an active cell on the aquifer side of the current cell.
- Two additional items have been added to the AQUCHWAT keyword to allow the flow to be limited between a minimum and maximum value.

GRID section

- The COPYREG keyword has been added to allow you to copy values from one array to another within a specified region.
- The OPERNUM keyword has been added to define regions for performing operations using OPERATER keyword. Keywords ADDREG, EQUALREG, MULTIREG and COPYREG that previously could reference FLUXNUM and MULTNUM can now also reference OPERNUM regions.
- The OPERATER keyword performs operations on arrays for specific regions
- It is now possible to apply a maximum and minimum limit to a property array using the OPERATE keyword. In addition, a new multiplication operator has also been added.
- The LGRCOPY keyword has been added to allow modification of values within an LGR when the values have only been set for global cells.

EDIT section

- The ADDREG, EQUALREG, MULTIREG and COPYREG keywords have been added to allow the user to modify array values within a specified region.
- The OPERATER keyword performs operations on arrays for specific regions (defined by the OPERNUM keyword in the GRID section).
- It is now possible to apply a maximum and minimum limit to a property array using the OPERATE keyword. In addition, a new multiplication operator has also been added.
- The LGRCOPY keyword has been added to allow modification of values within an LGR when the values have only been set for global cells.

PROPS section

- The ADDREG, EQUALREG, MULTIREG and COPYREG keywords have been added to allow the user to modify array values within a specified region.
- The OPERATER keyword performs operations on arrays for specific regions (defined by the OPERNUM keyword in the GRID section).
- It is now possible to apply a maximum and minimum limit to a property array using the OPERATE keyword. In addition, a new multiplication operator has also been added.
- The RKTRMDIR keyword allows directional transmissibility multipliers to be defined in the ROCKTAB tables.
- The RIVRXSEC and RBEDCONT keywords have been added to define river cross-sectional profile and bed contact table data.
- The DEPTHTAB, MASSFLOW and QHRATING keywords have been added to define both upstream and downstream river boundary conditions.

REGIONS section

- The ADDREG, EQUALREG, MULTIREG and COPYREG keywords have been added to allow the user to modify array values within a specified region.
- The MULTIPLY keyword has been added to allow the user to multiply values from one array by an integer constant within the current box.
- The OPERNUM keyword has been added to define regions for performing operations using OPERATER keyword.
- The OPERATER keyword performs operations on arrays for specific regions.
- It is now possible to apply a maximum and minimum limit to a property array using the OPERATE keyword. In addition, a new multiplication operator has also been added.

SOLUTION section

- The ADDREG, EQUALREG, MULTIREG and COPYREG keywords have been added to allow you to modify array values within a specified region.
- The ADD, MULTIPLY and COPY keywords have been added to allow you to modify grid array values within the current box.
- The OPERATER keyword performs operations on arrays for specific regions (defined by the OPERNUM keyword in the GRID or REGIONS section).
- It is now possible to use the OPERATE keyword in the SOLUTION section

- The RIVERSYS keyword has been added to define each new river

SUMMARY section

- Cumulative field aquifer influx and total field aquifer influx for a gas aquifer (input using AQUCHGAS keyword) can now be output in the summary section using mnemonics FAQRG and FAQTG. This information can also be output using the ALL mnemonic.
- New summary mnemonics have been added to output well and group potentials including wells that are shut or stopped. (When using the pre-existing conventional potential mnemonics such as WOPP, such wells are reported as having zero potential, even if this potential has actually been calculated for purposes such as priority drilling.) These new mnemonics have the same name as the conventional potential mnemonics, except that they are appended with a “2”.

  The list of these new mnemonics is: FOPP2, GOPP2, WOPP2, FWPP2, GWPP2, WWPP2, FGPP2, GGPP2, WGPP2, FOPI2, GOPI2, WOPI2, FWPI2, GWPI2, WWPI2, WWIP2, FGPI2, GGPI2, WGPI2, WGIP2, FGPPS2, GGPPS2, WGPPS2, FGPPF2, GGPPF2 and WGPPF2.
- A new facility has been added to enable you to define summary quantities in terms of constants, existing summary quantities and a number of mathematical functions. The user defined quantities are identified by a "U" as the second character of the name, with connection, field, group, region, segment and well quantities permitted. User defined quantities are created in the SCHEDULE section using the UDQ keyword.
- Grid block interpolation keywords BSCN_X, BCTRA_X for global grid, and LBSCN_X, LBCTRA_X for local grid are included to report interpolated values at specified XYZ coordinates for block brine and tracer concentration. The dimensions for the number of required summary interpolation outputs are defined by the RUNSPEC keyword BPIDIMS. This adds to existing interpolation keywords for pressures, heads and fresh water heads.
- River output keywords SRRQR, SRRQT, SRBQR, SRBQT, SRTQR, SRTQT, SRRFLOW, SRRAREA, SRRDEPTH, SRREXCH and SRRFRODE are added for run time reporting.
- The summary mnemonics WBHWCn, WGFWCn, WOFWCn and WWFWCn have been added to report gradient data with respect to well control variables.

SCHEDULE section

- A new multi-phase injector type has been added. This injects oil and/or water and /or gas depending on which phases have non-zero surface volume proportions specified. These surface volume proportions can be specified in the WCONINJE, WCONINJH or WCONINJP keywords. A multi-phase injector always injects its preferred phase (as specified in the keyword WELSPECS for the well) so that phase must have a non-zero surface volume proportion specified.
- The GWRTWCV keyword defines parameters with which the instantaneous gradients of certain well quantities can be calculated.
- The GUPFREQ keyword can used to specify the frequency at which the instantaneous gradients are updated.
- The WELCNTL keyword allows the control mode and operating target of a well to be changed without having to re-specify any other settings for the well.
- PLYVISC, PLYVISCS and SURFVISC can now all be used in the SCHEDULE section.
- The UDQ keyword has been added to the SCHEDULE section and facilitates the definition of the user defined summary quantities.

- The RIVERSYS keyword has been added to define each new river.
- REACHES, COMPRIV and GRDREACH keywords, part of the new river facility to define data structures for river systems and its connections to grid cells.
- The allowed field, group, well, segment and connections triggers in ACTIONX are all those field, group, well, segment and connections mnemonics listed in the SUMMARY section.
- Multipliers for the dual porosity matrix-fracture transmissibilities can now be specified in the SCHEDULE section. The new keywords MULTSIG, MULTSIGV, MULSGGD, MULSGGDV specify multipliers corresponding to the original specification of these transmissibilities in the GRID section through the keywords SIGMA, SIGMAV, SIGMAGD, SIGMAGDV. Additionally the DIFFMMF multiplier for the matrix-fracture diffusivities can now be specified multiple times in the SCHEDULE section as well as in the GRID section.

Tracers

- A new TRASxxx mnemonic has been added to the keyword RPTRST which controls the output of the product of tracer concentration with the carrier saturation to the restart file.
- TNUM and TRKPF can now be used in conjunction with the ADD, ADDREG, MULTIPLY and MULTIREG operations.

API Tracking

- An improved interpolation to avoid crossing lines in PVTO is available by setting OPTIONS(100). If > 0, it sets up the gap between two consecutive RS values for the interpolation when API Tracking is used, and uses an improved interpolation to avoid crossing lines in PVTO. The gap is a percentage given by RSGAP=0.01*OPTION(100). Note that this gap should not be too large. A minimum value of 0.01 is set.

History matching

- The gradient option is now available with the parallel option.

Rivers

- A new water river facility is available through the RIVERSYS and related keywords. This provides a full coupling of surface river networks with groundwater aquifers. Each river should be introduced by the RIVERSYS keyword. Its structure defined by the COMPRIV, REACHES and GRDREACH keywords. Its profile and boundary conditions are defined by MASSFLOW and DEPTHTAB. RIVRDIMS sets required dimensions of the different tables. RFT/PLT type output for the river option is sent to the RFT file. For further details, see "Rivers",

Output and restarts

- Output of regions defined for performing operations on arrays can be achieved by entering mnemonic OPERNUM under the RPTGRID keyword.

New facilities in ECLIPSE 300

RUNSPEC section

- The GDIMS keyword has been added to specify the number of parameters required for the instantaneous gradient options.
- The CBMOPTS keyword can be used to disallow water flow in and out of coal cells.
- The UDQDIMS and UDQPARAM keywords have been added to specify the dimensions and parameters associated with the user defined summary quantity facility.
- The LGRCOPY keyword has been added to allow modification of values within an LGR when the values have only been set for global cells. Used in the RUNSPEC section means that it will apply to all LGRs in the model
- The ASPHALTE keyword has been added to activate the Asphaltene modeling
- The TABDIMS keyword has been extended with item 17,18,19 and 20 to specify the number of rows in look-up tables used for asphaltene modeling and the number of maximum flocculation allowed.
- The CO2STORE keyword has been added to activate the CO_2 storage option. It supersedes the CO2SEQ keyword.

GRID section

- The OPERNUM keyword has been added to define regions for performing operations using OPERATER keyword. Keywords ADDREG, EQUALREG, MULTIREG and COPYREG that previously could reference FLUXNUM and MULTNUM can now also reference OPERNUM regions.
- The OPERATER keyword performs operations on arrays for specific regions.
- It is now possible to apply a maximum and minimum limit to a property array using the OPERATE keyword. In addition, a new multiplication operator has also been added.
- The COPYBOX keyword has been added to copy values from one box of cells to another.
- The keywords BTOBALFA and BTOBALFV have been added for dual porosity runs to switch on additional transmissibilities between fracture cells and the matrix cells below them in the grid Z-direction, and provide multipliers specifying the contact fraction between these cells.
- The LGRCOPY keyword has been added to allow modification of values within an LGR when the values have only been set for global cells.
- Using switch 117 of keyword OPTIONS3 the rock volumes are scaled by the net to gross, NTG, values, restoring the pre-2007.1 behavior.

EDIT section

- The OPERATER keyword performs operations on arrays for specific regions (defined by the OPERNUM keyword in the GRID section).
- It is now possible to apply a maximum and minimum limit to a property array using the OPERATE keyword. In addition, a new multiplication operator has also been added.
- The LGRCOPY keyword has been added to allow modification of values within an LGR when the values have only been set for global cells.

PROPS section

- The OPERATER keyword performs operations on arrays for specific regions (defined by the OPERNUM keyword in the GRID section).
- It is now possible to apply a maximum and minimum limit to a property array using the OPERATE keyword. In addition, a new multiplication operator has also been added.
- The LBCCOEFR keyword enables the LBC coefficients to be modified on an equation of state region basis.
- The COPYBOX keyword has been added to copy values from one box of cells to another.
- The RKTRMDIR keyword has been added as a placeholder only. It allows five-column directional transmissibility ROCKTAB tables created for ECLIPSE 100 to be read without error. Columns four and five are ignored
- SOLUBILT is a new keyword extending the existing SOLUBILI keyword. It allows you to specify the aqueous phase properties of a water-CO_2 system with temperature dependencies.
- The columns of the SOLUBILI keyword can now be defaulted. The default values for the formation volume factor are calculated using a more accurate method. This keyword can be used with SALINITR and RTEMP to specify the salinity and temperature on a regional basis. There is no limit on the number of records anymore.
- A new item 3 has been added to record 1 of the SOLUBILS keyword to define the reference salt concentration for stock tank water. There is no limit on the number of records anymore.
- The ALPHAD keyword has been added to specify tables of transport coefficients in oil and gas phases when hysteresis is activated. It allows the user to specifically modify the drainage mobility components as functions of the total mole fraction of a specified component.
- The ALPHAI keyword has been added to specify tables of transport coefficients in oil and gas phases when hysteresis is activated. It allows the user to specifically modify the imbibition mobility components as functions of the total mole fraction of a specified component.
- The ODD3P keyword is used to select the ODD3P three-phase relative permeability and capillary pressure hysteresis model option.
- The EPSODD3P keyword is used to specify the end-point scaling, reference surface tensions and threshold interfacial tensions for the ODD3P three-phase relative permeability and capillary pressure hysteresis model option.
- The PCODD3P keyword is used to specify the capillary pressure and saturation calculation control parameters for the ODD3P three-phase relative permeability and capillary pressure hysteresis model option.
- The PCODD3PW keyword is used to specify the relationship between capillary pressure and water saturation as a tabulated function for the ODD3P three-phase relative permeability and capillary pressure hysteresis model option.
- The PCODD3PG keyword is used to specify the relationship between capillary pressure and gas saturation as a tabulated function for the ODD3P three-phase relative permeability and capillary pressure hysteresis model option.
- The PSORG, PSGRO, PSORW, PSWRO, PSGRW and PSWRG keywords are used to specify the gridblock end-point primary saturations for the ODD3P three-phase relative permeability and capillary pressure hysteresis model option.

- The HSORG, HSGRO, HSORW, HSWRO, HSGRW and HSWRG keywords are used to specify the gridblock end-point hysteresis saturations for the ODD3P three-phase relative permeability and capillary pressure hysteresis model option.
- The LANGMULT keyword is used to scale the adsorption capacities for the coal bed methane model.
- The DIFFCWAT keyword is used to specify diffusion coefficients when using the CO2STORE or GASWAT option.
- The ASPREWG keyword has been added to specify asphaltene precipitation data based on percentage weight of specified components, when Asphaltene modeling is activated.
- The ASPFLOC keyword has been added to specify asphaltene precipitating and flocs components, when Asphaltene modeling is activated. This keyword must be used in conjunction with ASPFLRT.
- The ASPFLRT keyword has been added to specify asphaltene flocculation/dissociation rates, when Asphaltene modeling is activated. This keyword must be used in conjunction with ASPFLOC.
- The ASPDEPO keyword has been added to specify asphaltene deposition data, when Asphaltene modeling is activated.
- The ASPKDAM keyword has been added to specify permeability damage due to asphaltene deposition, when Asphaltene modeling is activated.
- The ASPVISO keyword has been added to specify oil viscosity changes due to asphaltene precipitation, when Asphaltene modeling is activated.
- The LGRCOPY keyword has been added to allow modification of values within an LGR when the values have only been set for global cells.
- The RPTPROPS keyword has been extended for the ODD3P tables and saturation end-points with the following mnemonics; PSORG, PSGRO, PSORW, PSWRO, PSGRW, PSWRG, HSORG, HSGRO, HSORW, HSWRO, HSGRW, HSWRG, EPSODD3P, PCODD3P, PCODD3PG and PCODD3PW

REGIONS section

- The OPERNUM keyword has been added to define regions for performing operations using OPERATER keyword.
- The OPERATER keyword performs operations on arrays for specific regions.
- It is now possible to apply a maximum and minimum limit to a property array using the OPERATE keyword. In addition, a new multiplication operator has also been added.
- The COPYBOX keyword has been added to copy values from one box of cells to another.
- The ALPHANUD keyword has been added to specify which set of transport coefficient tables (input using ALPHAD in the PROPS section) should be used to calculate transport coefficients for drainage in a compositional model.
- The ALPHANUI keyword has been added to specify which set of transport coefficient tables (input using ALPHAI in the PROPS section) should be used to calculate transport coefficients for imbibition in a compositional model.
- The PSTNUM, ISTNUM and DSTNUM keywords are used to specify the gridblock primary, increasing and decreasing saturation table numbers for the ODD3P three-phase relative permeability and capillary pressure hysteresis model option.

- The SDROG, SDRGO, SDROW, SDRWO, SDRGW and SDRWG keywords are used to specify the initial gridblock saturation directions for the ODD3P three-phase relative permeability and capillary pressure hysteresis model option.
- The LGRCOPY keyword has been added to allow modification of values within an LGR when the values have only been set for global cells.
- The RPTREGS keyword has had the following mnemonics added for the ODD3P option to output saturation region numbers and direction indicators; PSTNUM, ISTNUM, DSTNUM, SDROW, SDRWO, SDROG, SDRGO, SDRGW and SDRWG.

SOLUTION section

- The OPERATER keyword performs operations on arrays for specific regions (defined by the OPERNUM keyword in the GRID or REGIONS section).
- It is now possible to use the OPERATE keyword in the SOLUTION section
- New RPTSOL mnemonics have been added for the ODD3P three-phase relative permeability and capillary pressure hysteresis model option.
- The list of new mnemonics consists of: SONRM, SGNRM, SWNRM, SDROW, SDRWO, SDROG, SDRGO, SDRGW, SDRWG, SOWNT, SWONT, SOGNT, SGONT, SGWNT, SWGNT, SOWNH, SWONH, SOGNH, SGONH, SGWNH, SWGNH, SOWNE, SWONE, SOGNE, SGONE, SGWNE, SWGNE, SORWB, SWROB, SORGB, SGROB, SGRWB, SWRGB, SORWP, SWROP, SORGP, SGROP, SGRWP, SWRGP, IFTGO, IFTOW, IFTGW, KROWN, KRWON, KROGN, KRGON, KRGWN, KRWGN, KROWH, KRWOH, KROGH, KRGOH, KRGWH, KRWGH, KROWE, KRWOE, KROGE, KRGOE, KRGWE, KRWGE, KROWT, KRWOT, KROGT, KRGOT, KRGWT and KRWGT.
- The DBGODD3P keyword provides debug information for the ODD3P three-phase relative permeability and capillary pressure hysteresis model option

SUMMARY section

- New SUMMARY mnemonics have been added to output well and group potentials including well that are shut/stopped. (When using the pre-existing conventional potential mnemonics such as WOPP, such wells are reported as having zero potential, even if this potential has actually been calculated for purposes such as priority drilling.) These new mnemonics have the same name as the conventional potential mnemonics, except that they are appended by a “2”.
- The list of these new mnemonics is: FOPP2, GOPP2, WOPP2, FWPP2, GWPP2, WWPP2, FGPP2, GGPP2, WGPP2, FWPI2, GWPI2, WWPI2, WWIP2, FGPI2, GGPI2, WGPI2, WGIP2, FCVPP2, GCVPP2, WCVPP2
- New SUMMARY mnemonics have been added to output the component molar totals FCGMT, GCGMT, WCGMT, FCOMT, GCOMT and WCOMT.
- A new facility has been added to enable the user to define summary quantities in terms of constants, existing summary quantities and a number of mathematical functions. The user defined quantities are identified by a "U" as the second character of the name, with connection, field, group, region, segment and well quantities permitted. User defined quantities are created in the SCHEDULE section using the UDQ keyword.
- New SUMMARY gridblock mnemonics have been added for the ODD3P three-phase relative permeability and capillary pressure hysteresis model option.

- The list of new mnemonics consists of: BSONRM, BSGNRM, BSWNRM, BSDROW, BSDRWO, BSDROG, BSDRGO, BSDRGW, BSDRWG, BSOWNT, BSWONT, BSOGNT, BSGONT, BSGWNT, BSWGNT, BSOWNH, BSWONH, BSOGNH, BSGONH, BSGWNH, BSWGNH, BSOWNE, BSWONE, BSOGNE, BSGONE, BSGWNE, BSWGNE, BSORWB, BSWROB, BSORGB, BSGROB, BSGRWB, BSWRGB, BSORWP, BSWROP, BSORGP, BSGROP, BSGRWP, BSWRGP, BIFTGO, BIFTOW, BIFTGW,BKROWN, BKRWON, BKROGN, BKRGON, BKRGWN, BKRWGN, BKROWH, BKRWOH, BKROGH, BKRGOH, BKRGWH, BKRWGH, BKROWE, BKRWOE, BKROGE, BKRGOE, BKRGWE, BKRWGE, BKROWT, BKRWOT, BKROGT, BKRGOT, BKRGWT and BKRWGT.
- The summary mnemonics WBHWCn, WGFWCn, WOFWCn and WWFWCn have been added to report gradient data with respect to well control variables.
- Cumulative field aquifer influx and total field aquifer influx for a gas aquifer (input using AQUCHGAS keyword) can now be output in the summary section using mnemonics FAQRG and FAQTG. This information can also be output using the ALL mnemonic.

SCHEDULE section

- The UDQ keyword has been added to the SCHEDULE section and facilitates the definition of the user defined summary quantities.
- Two new items have been added to the PICOND keyword to allow you to specify the lower and upper bounds for use in constructing the generalized pseudo-pressure integrals. These bounds are specified as dimensionless factors which are applied to the completion cell pressure to generate the lower and upper pressures for the table. They allow the use of smaller quadrature intervals by allowing control over the pressure range of the table.
- New RPTSCHED mnemonics have been added for the ODD3P three-phase relative permeability and capillary pressure hysteresis model option.
- The list of new mnemonics consists of: SONRM, SGNRM, SWNRM, SDROW, SDRWO, SDROG, SDRGO, SDRGW, SDRWG, SOWNT, SWONT, SOGNT, SGONT, SGWNT, SWGNT, SOWNH, SWONH, SOGNH, SGONH, SGWNH, SWGNH, SOWNE, SWONE, SOGNE, SGONE, SGWNE, SWGNE, SORWB, SWROB, SORGB, SGROB, SGRWB, SWRGB, SORWP, SWROP, SORGP, SGROP, SGRWP, SWRGP, IFTGO, IFTOW, IFTGW, KROWN, KRWON, KROGN, KRGON, KRGWN, KRWGN, KROWH, KRWOH, KROGH, KRGOH, KRGWH, KRWGH, KROWE, KRWOE, KROGE, KRGOE, KRGWE, KRWGE, KROWT, KRWOT, KROGT, KRGOT, KRGWT and KRWGT.
- The allowed field, group, well, segment and connections triggers in ACTIONX are all those field, group, well, segment and connections mnemonics listed in the SUMMARY section.
- Almost all SCHEDULE keywords can now be used in an ACTIONX /ENDACTIO block
- Multipliers for the dual porosity matrix-fracture transmissibilities can now be specified in the SCHEDULE section. The new keywords MULTSIG, MULTSIGV, MULSGGD, MULSGGDV specify multipliers corresponding to the original specification of these transmissibilities in the GRID section through the keywords SIGMA, SIGMAV, SIGMAGD, SIGMAGDV. Additionally the DIFFMMF multiplier for the matrix-fracture diffusivities can now be specified multiple times in the SCHEDULE section as well as in the GRID section.
- The DBGODD3P keyword provides debug information for the ODD3P three-phase relative permeability and capillary pressure hysteresis model option
- The PERM_MOD mnemonic have been added to RPTSCHED for output of permeability multipliers when using the rock compaction option.
- The XFW and YFW mnemonics have been added to RPTSCHED for output of liquid and vapor components mass fraction.

Wells

- A new multi-phase injector type has been added. This injects water and gas in specified surface volume proportions. These surface volume proportions can be specified in the WCONINJE, WCONINJH or WCONINJP keywords. A multi-phase injector always injects its preferred phase (as specified in the keyword WELSPECS for the well) so that phase must have a non-zero surface volume proportion specified.
- New group production rate control modes have been added to allow production balancing using either the surface gas, or surface water, volumes. These new modes are requested using the PBGS and PBWS mnemonics in item 2 of GCONPROD respectively. With a balancing fraction of 1, these new control modes allow you to model scenarios where you do not want to produce excess gas or water. The keywords GTADD, GTMULT,GRUPTARG and GCONPRI have also been extended to alter or set these values.
    - By default, gas injected during PBGS production balancing is reported as re-injected gas, and subsequently included in the calculation of Export and Excess gas (for example, in the FEGR and FEXGR summary vectors). OPTIONS3(108) can be used to exclude gas injected during PBGS production balancing from being reported as re-injected gas.
    - If OPTIONS3(104) is set to a value greater than 0, then the PBGS surface volume production balancing group control mode balances the *produced* surface gas, instead of the *excess* surface gas, to the specified fraction (or multiple) of the injected surface water volume.
- The keywords GTADD and GTMULT have been extended to allow alteration of production balancing (PBRL) limits.
- The WLISTARG keyword, previously only available in ECLIPSE 100, is now available in ECLIPSE 300. This is used to reset operating targets or limits for well lists.
- The GWRTWCV keyword defines parameters with which the instantaneous gradients of certain well quantities can be calculated.
- The GUPFREQ keyword can used to specify the frequency at which the instantaneous gradients are updated.
- The WELCNTL keyword allows the control mode and operating target of a well to be changed without having to re-specify any other settings for the well.
- The WREGROUP keyword, previously only available in ECLIPSE 100, is now available in ECLIPSE 300. This is used to transfer wells automatically from one group to another, depending on the value of a chosen quantity such as the well's water cut or oil production rate.
- OPTIONS3 item 121 can be used to specify an upper limit for the blocking factor calculated by the generalized pseudo-pressure special inflow equation. This can be used to prevent convergence problems when blocking factors greater than unity are generated. See "Generalized pseudo-pressure in ECLIPSE 300" for more information.

Aquifers

- The property data for the constant head/pressure gas aquifer can now be specified using the AQUCHGAS keyword in both the SOLUTION and SCHEDULE sections.
- For compositional runs, the AQSTREAM keyword should be used to specify the composition of the injected gas stream for the AQUCHGAS aquifer.
- Connections to the Constant Head Gas (AQUCHGAS) aquifer are read in by the AQUANCON keyword.

- Output of analytic data in the SOLUTION section can be achieved by entering mnemonic AQUCHGAS under the RPTSOL keyword.
- Output of analytic data in the SCHEDULE section can be achieved by entering mnemonic AQUCHGAS under the RPTSCHED keyword.
- If this new aquifer (AQUCHGAS) has been re-specified in the SCHEDULE section, then that keyword, together with its modified data, will be output to the .PRT file by default.
- Aquifer quantities for AQUCHGAS can now be output using mnemonics in the SUMMARY section. This information can also be output via the ALL mnemonic.
- Two additional items have been added to the AQUCHWAT keyword to allow the flow to be limited between a minimum and maximum value.
- OPTIONS3 (122) switch generates warning messages from the AQUANCON keyword when cells cannot be connected to the aquifer because there is an active cell on the aquifer side of the current cell.

Geomechanics

- Two new mnemonics PERMSTAB and BIOTCTAB can be used in RPTRST, RPTSOL and RPTSCHED to output permeability and Biot's constant multipliers based on the stress dependence of those two variables (see keyword PERMSTAB and BIOTCTAB respectively).

Thermal

- The IAPWS 97 analytic steam table can be accessed with the keyword THSTT97 in the PROPS section.
- The Antoine K-value correlation can be specified with keyword KVAN in the PROPS section.
- Improvements to thermal AIM have been made including two new criteria that can be specified with the AIMCON keyword in the SCHEDULE section.
- Stability of the thermal Multi-Segment well model has improved. There are some new adjustable convergence parameters in the keyword CVCRIT but normally these should be defaulted.
- Two new thermal well constraints, "steam trap" which includes either saturation pressure or saturation temperature control, are now available with the thermal Multi-Segment well model. This means that the SATP and SATT modes are available for use in the thermal MSW model in WCONPROD and also items 15 and 16 of the WELLPROD keyword.
- A new linear solver has been introduced for thermal cases. See keyword JALS.

Reservoir Optimization

- A new CLRFILES mnemonic has been added to the RPTOPT keyword. This removes SEPARATE results files from any previous optimization for this model if necessary (that is, if the current optimization is about to overwrite some or all of them).

Coal bed methane and Shale gas

- A new option (data item 2) have been added to the CBMOPTS keyword. This can be used to disallow water flow in and out of coal cells.
- The LANGMULT keyword is used to scale the adsorption capacities on a cell by cell basis.

- The Palmer-Mansoori rock compaction model can be used to model coal swelling and shrinkage. Two models exists. Using the keyword ROCKPAMA or the keyword ROCKPAME. The second model is a modified Palmer-Mansoori model taking into account the composition of the adsorbed gas.
- Coal region numbers can be output to the print file by RPTGRID.

CO2 storage

The option CO2STORE for modeling of geological storage of CO2 in saline aquifers is now compatible with the Adaptive Implicit Method, AIM. Using the option the amount of dissolved CO2 in the aqueous phase is now reported in the fluid in place report (PRT file). Using the Hysteresis option also the amount of CO2 trapped by an immobile gas phase is reported. Salt precipitation and dissolution of the NaCl component is made dependent on the concentration of CaCL2. It is now also possible to output the aqueous pH and ions (H+,H-,OH-,CO3--,HCO3-1,CL-,Na+, Ca++, CaCl+).

- A target solid saturation change can be set by item 21of TSCRIT when using the CO2STORE option.

Output and restarts

- Output of analytic data in the SOLUTION section can be achieved by entering mnemonic AQUCHGAS under the RPTSOL keyword.
- Output of analytic data in the SCHEDULE section can be achieved by entering mnemonic AQUCHGAS under the RPTSCHED keyword.
- If this new aquifer (AQUCHGAS) has been re-specified in the SCHEDULE section, then that keyword, together with its modified data, will be output to the .PRT file by default.
- Output of regions defined for performing operations on arrays can be achieved by entering mnemonic OPERNUM under the RPTGRID keyword.
- Output of coal regions can be achieved by entering mnemonic COALNUM under the RPTGRID keyword.
- The mnemonics LANGMULT, ROCKPAMA and ROCKPAME have been added to the keyword RPTPROPS, outputting the Langmuir multipliers and the rock compaction data.
- The aqueous speciation and pH-level can be output with RPTRST or RPTSCHED when using the CO2STORE option with the mnemonics AQSP and AQPH.
- When using the Hysteresis option the critical gas saturation and the trapped gas saturation (immobile) can be output with RPTRST or RPTSCHED using the mnemonics SGCRH and SGTRH.
- The pore volume and permeability multipliers as used by the rock compaction option can now be output by the ROCKC mnemonic (both) or the PORV_MOD or PERM_MOD mnemonic added to the keyword RPTRST.
- Mass fractions for each component in liquid and vapor is output with RPTRST, RPTSOL or RPTSCHED using the mnemonics XFW and YFW.

Tracers

- A new TRMFxxxx mnemonic has been added to the keyword RPTRST, which outputs the product of tracer concentration with the molar fraction of the carrier to the restart file. The tracer name to be output must be appended to the end of the TRMF part of the mnemonic.

Behavioral changes in ECLIPSE 100

These may cause changes to the results of a simulation, or alter the form or content of the output.

- Various (non-physical) constants within the program have had their precision increased to reduce cross-platform (and cross-version) differences in behavior. Pre 2007.1 behavior can be restored using OPTIONS(97).
- Note that OPTIONS(97) was used in 2006.2 to provide the functionality now provided by switch 100.
- Various improvements have been made regarding property calculations in the BRINE model. Pre 2007.1 behavior can be restored using OPTIONS(102).
- The tracer concentrations used in the scale and gas calorific value models are now only lagged by a single timestep. Prior to 2007.1 the concentrations were lagged by two timesteps. Pre 2007.1 behavior can be restored using OPTIONS(104).
- A bug was fixed which may cause a change in results for models that contain the GRAVDR keyword. Pre-2007.1 behavior can be restored using OPTIONS(106).

RUNSPEC section

- The BIGMODEL keyword is now redundant and will be ignored.
- The first item of the MEMORY keyword is redundant and will be ignored.

SUMMARY section

- FAQRG and FAQTG have been added to the list of summary mnemonics represented by ALL.

SCHEDULE section

- The WLISTARG keyword now accepts default values specified for new operating targets and limits for well lists. These default values have the same behavior as those in the WELTARG keyword (that is, the simulator will use the corresponding flowing value for the well at the end of the previous timestep as the target or limit)
- A problem in the allocation of work-space for well potential calculations has been fixed. This only affects models that use an old-style RUNSPEC section, and this may result in a change in the reported well quantities in such models. The new behavior can be produced with pre-2007.1 code by converting the model to use a new-style RUNSPEC section (for example, using ECLIPSE Office). The pre-2007.1 behavior can be restored in the 2007.1 code by setting OPTIONS item 101 to a multiple of 2.
- Potentials are now always calculated for wells and groups. Previously there was logic to limit the calculation of potentials to cases where they were deemed necessary. The pre-2007.1 behavior can be restored by setting OPTIONS item 101 to a multiple of 3.
- An issue with the preservation of the actual well control mode when well potentials are calculated for the prioritization option (see the PRIORITY keyword) or for prioritized drilling queues (see the WDRILPRI keyword) has been fixed, and this may result in a change in the reported well quantities in such models. The pre-2007.1behavior can be restored by setting OPTIONS item 101 to a multiple of 5.

- Various aspects of the GCONSALE keyword have been modified.
  - When item 5 is set to MAXR (to specify the procedure on exceeding a maximum rate limit) then the re-injection fraction limit is now set to 100 to effectively remove the reinjection fraction limit on the injectors. Before the 2007.1 version, the re-injection fraction limit was set to 1 but that caused problems if the sales target was zero and there was no import or consumption.
  - A non-positive value can now be specified for the lower limit. Before the 2007.1 version, a non-positive value was taken to mean that no lower limit was to be applied.

  The pre-2007.1 behavior can be restored by setting OPTIONS item 103 to a positive value. Note that the value of the MAXR re-injection fraction limit can be modified by setting OPTIONS item 103 to a negative value.
- Various fixes have been made to the code for applying group efficiency factors (specified with the GEFAC keyword) in reservoir coupling runs. These may result in changes to the rates reported for the groups in the master reservoir. The pre-2007.1 behavior can be recovered by setting OPTIONS item 105 to a non-zero value.

Behavioral changes in ECLIPSE 300

- A better handling of negative capillary pressure handling in gas water systems has been implemented. Pre-2006.2 behavior can be restored using OPTIONS3 item 110.
- An improved treatment of high RV values has been implemented which can affect the nature of the single phase (oil or gas) seen by the simulator. Pre-2007.1 behavior can be restored using OPTIONS3 item 111.
- Various improvements have been made regarding equilibration of the BRINE model. Pre 2007.1 behavior can be restored using OPTIONS3 item 116.
- A better handling of dual porosity systems has been implemented. Pre 2006.2 behavior can be restored using OPTIONS3 item 120.
- The rock volumes are no longer scaled by the NTG values. Pre 2007.1 behavior can be restored using OPTIONS3 item 117.
- Dual porosity transmissibilities are no longer scaled by the NTG values. Pre 2007.1 behavior can be restored using OPTIONS3 item 117
- A better handling of track tracers concentration has been implemented. A better behavior can be obtained using OPTIONS3 item 123.
- The phase stability check for the GASWAT option can be modified slightly in order to prevent the H2O rich phase being labeled as gas; this facility can be optioned by using OPTIONS3 item 125.

SUMMARY section

- FAQRG and FAQTG have been added to the list of summary mnemonics represented by ALL.
- The units of both mass rates and mass totals (for example, the WOMR and GWMT summary vectors) have been changed from **long tons** (1 long ton = 2,240 lbs) to **short tons** (1 short ton = 2,000 lb) when working in FIELD units. The pre-2007.1 behavior can be restored by setting OPTIONS3 item 112 to a non-zero value.

SCHEDULE section

- The same results are now obtained if WNETDP is used inside ACTIONX compared to using it directly within the SCHEDULE section. The former behavior within ACTIONX was different to the behavior if it was used just in the main body of the SCHEDULE section.
- The calculations and reporting in the drift flux model have been made more robust in the presence of small flow rates and small hold-ups. The pre-2007.1 behavior can be restored using OPTIONS3 item 107.
- The checking for whether separator stage gas is being used for injection has been made more flexible. (Injection of separator stage gas requires more complex logic in the gas accounting calculations). Previously this was turned on (and never off) if separator stage gas was specified in any of the GINJGAS, GRUPINJE, WINJORD or WINJMIX keywords. From 2007.1, it is only turned on while separator stage gas is actually being used for injection. This may cause differences in the gas accounting results.
- A new algorithm has been added to calculate the availability of items in mixtures specified using WINJORD. Previously WINJORD mixtures used the WINJMIX algorithm and this led to incorrect violabilities. OPTIONS3 item 113 can be used to recover the pre-2007.1 behavior.
- If the injected fluid nature 'GAS' is specified in the GINJGAS, GRUPINJE, WELLINJE, WINJGAS, WINJMIX and WINJORD keywords, it is internally converted to injection fluid nature 'GV' with its source being the field separator (which has better gas accounting and re-injection behavior). OPTIONS3 item 113 can be used to recover the pre-2007.1 behavior.
- In the complex gas accounting that occurs when there is fuel and/or sales gas with intermediate separator stage gas used for re-injection, mixture items were previously excluded from the re-injection calculation. These have been included for 2007.1. OPTIONS3 item 113 can be used to recover the pre-2007.1 behavior.
- Prior to 2007.1 ACTIONR and ACTIONX conditions involving regions were evaluated using the summary quantity values from the previous timestep. From 2007.1 the values from the current timestep are used. The pre-2007.1 behavior can be recovered via OPTIONS3 item 118.
- Prior to 2007.1 the maximum timestep length after a well modification was not respected if the well modification was a result of an ACTION(G,R,W,S,X) being triggered. From 2007.1 the maximum timestep will be honoured. The pre-2007.1 behavior can be recovered via OPTIONS3 item 118.
- Prior to 2007.1 the ACTION(G,R,W,S,X) conditions were processed before outputting the summary data. As a result summary data from the previous timestep may be overwritten when an ACTION(G,R,W,S,X) is triggered. From 2007.1 the summary data will be exported before processing the ACTION(G,R,W,S,X) conditions. The pre-2007.1 behavior can be recovered via OPTIONS3 item 118.
- To accommodate the increased range of summary quantities permitted within an ACTION(G,W,X) condition a new syntax for specify a component number has been introduced. From 2007.1 component numbers should be appended to the quantity with an underscore. Hence the pre-2007.1 syntax for the well liquid mole fraction of component 1 is modified from WXMF1 to WXMF_1.
- The default BHP limit has changed to 592.15 atma when using the CO2STORE option with the keywords WCONINJE and WCONINJP.

- The modified solution update method for multi-segment well previously activated by OPTIONS3 item 83 is now the default. The pre 2007.1 update method can be restored by via OPTIONS3 item 83.

Non-Darcy flows

- The alternative model for Capillary Number effects in gas condensate reservoirs, which is selected by setting item 5 of the VELDEP keyword to a non-zero value, has been modified to be more faithful to the ideas in Whitson, Fevang and Saevareid [Ref. 99]. A description of both the old and new behavior is given in "Non-Darcy Flow". The pre-2007.1 behavior can be restored by setting OPTIONS3 item 99 to a non-zero value.

Thermal

- The improved stability in the Multi-Segment well model and improvements in the nonlinear convergence may produce slight differences in results if the simulation is run for a long time.
- If a user specifies the A, B and C coefficients to be zero in keyword KVCR, then the simulator will automatically set this component to be nonvolatile. Previously, the user had to explicitly set a component to be DEAD with keyword CVTYPE.
- The rock volumes are no longer scaled by the NTG values. Pre 2007.1 behavior can be restored using OPTIONS3 item 117. Note that for a thermal case both the rock volume and the pore volume need to be zero for the cell to become inactive.

Aquifers

- The equilibration tilt correction is now applied to numerical aquifer blocks when item 9 of the EQUIL keyword is positive. Previous behavior can be restored via OPTIONS3(105).
- For the GASWAT or CO2STORE options the water properties are calculated using the equation of state. An EoS region number can entered for the keywords AQUCHWAT, AQUCT and AQUFETP. If the item 119 of OPTIONS3 is set to 1, this reverts to the pre-2007.1 method which uses a PVT table number.

Equilibration

- The equilibration block pressure is now taken as the central pressure as oppose to an average over the sampling points specified by item 9 of the EQUIL keyword. Previous behavior can be restored via OPTIONS3(105).

CO2SOL

- Pre-2007.1 the default values of the water formation volume factor of the solubility table used in the CO2SOL option could produce an incorrect water density. This has been corrected in 2007.1. If OPTIONS3(115) is set =1 then the pre-2007.1 behavior is restored.
- The water density is now calculated more accurately.

Reservoir Optimization

- There has been a change in behavior when an SCSA control parameter is specified (in item 1 of `OPTPARS`) for a segment of a well. In order to remove possible conflicts with other targets and limits during the optimization runs, such wells are now switched to BHP control mode (using the actual BHP for the well from the base run) and all other limits are removed. Prior to 2007.1, pre-existing control targets and limits were not removed, and these often interfered with the manipulation of the segments by the optimizer.

Adjoint gradient screening

- The adjoint gradient screening code has been modified so that it will attempt to calculate the requested adjoint gradients even if the simulation terminates early (for instance, because all the wells are shut, or because an economic limit has been violated).
  - If an adjoint gradient screening run terminates early (for instance, because all the wells are shut, or because an economic limit has been violated) and the `AJGWELLS` and `AJGPARAM` keywords have not been specified by the early exit time, then no adjoint gradients will be generated as none will have been requested by that stage of the simulation. In order to avoid this problem, these keywords should be specified right at the beginning of the `SCHEDULE` section.

Note This advice on the positioning of the `AJGWELLS` keyword is different to the advice given pre-2007.1. Previously it was recommended that this keyword should be placed just before the `END` keyword).

New keywords in ECLIPSE 100

RUNSPEC section

GDIMS	Specify the number of instantaneous gradient parameters
LGRCOPY	Copy global values to local cells before operations within an LGR.
RIVRDIMS	Define river dimension data.
UDQDIMS	Dimensions for the user defined quantity facility.
UDQPARAM	Parameters for the user defined quantity facility.

GRID section

COPYREG	Copy one array to another in the specified region
OPERNUM	Defines regions for performing operations using OPERATER keyword.
OPERATER	Performs operations on arrays for specific regions.
LGRCOPY	Copy global values to local cells before operations within an LGR.

EDIT section

ADDREG	Add a constant to an array in the specified region
EQUALREG	Set an array to a constant value in the specified region
MULTIREG	Multiply an array by a constant in the specified region
COPYREG	Copy one array to another in the specified region
OPERATER	Performs operations on arrays for specific OPERNUM regions.
LGRCOPY	Copy global values to local cells before operations within an LGR.

PROPS section

ADDREG	Add a constant to an array in the specified region
EQUALREG	Set an array to a constant value in the specified region
MULTIREG	Multiply an array by a constant in the specified region
COPYREG	Copy one array to another in the specified region
OPERATER	Performs operations on arrays for specific OPERNUM regions.
RKTRMDIR	Expands ROCKTAB tables to have five columns instead of three, for directional transmissibility multipliers.
RIVRXSEC	River cross-sectional profile table data.
RBEDCONT	River bed contact data
DEPTHTAB	River boundary condition data
MASSFLOW	River boundary condition data
QHRATING	River boundary condition data

REGIONS section

ADDREG	Add a constant to an array in the specified region
EQUALREG	Set an array to a constant value in the specified region
MULTIREG	Multiply an array by a constant in the specified region
COPYREG	Copy one array to another in the specified region
MULTIPLY	Multiply an array by a constant in the current box
OPERNUM	Defines regions for performing operations using OPERATER keyword.
OPERATER	Performs operations on arrays for specific regions.

SOLUTION section

ADDREG	Add a constant to an array in the specified region
EQUALREG	Set an array to a constant value in the specified region
MULTIREG	Multiply an array by a constant in the specified region
COPYREG	Copy one array to another in the specified region
ADD	Add a constant to an array in the current box
MULTIPLY	Multiply an array by a constant in the current box
COPY	Copy one array to another in the current box
OPERATER	Performs operations on arrays for specific OPERNUM regions.
OPERATE	Performs operations on arrays for a specific box.
RIVERSYS	Defines each new river

SUMMARY section

ALL	Extension of ALL mnemonic to include FAQRG and FAQTG.
BSCN_X	Report brine concentration at specified coordinate
BCTRA_X	Report tracer concentration at specified coordinate.
CUxxxxxx	User defined connection quantity.
FAQRG	Field aquifer influx rate for gas aquifer.
FAQTG	Cumulative field aquifer influx rate for gas aquifer.
FGPI2	Field gas potential injection rate (including values calculated on shut/stopped wells)
FGPP2	Field gas potential production rate (including values calculated on shut/stopped wells)
FGPPF2	Field free gas potential production rate (including values calculated on shut/stopped wells)
FGPPS2	Field solution (dissolved) gas potential production rate (including values calculated on shut/stopped wells)
FOPI2	Field oil potential injection rate (including values calculated on shut/stopped wells)
FOPP2	Field oil potential production rate (including values calculated on shut/stopped wells)
FUxxxxxx	User defined field quantity.

FWPI2	Field water potential injection rate (including values calculated on shut/stopped wells)
FWPP2	Field water potential production rate (including values calculated on shut/stopped wells)
GGPI2	Group gas potential injection rate (including values calculated on shut/stopped wells)
GGPP2	Group gas potential production rate (including values calculated on shut/stopped wells)
GGPPF2	Group free gas potential production rate (including values calculated on shut/stopped wells)
GGPPS2	Group solution (dissolved) gas potential production rate (including values calculated on shut/stopped wells)
GOPI2	Group oil potential injection rate (including values calculated on shut/stopped wells)
GOPP2	Group oil potential production rate (including values calculated on shut/stopped wells)
GUxxxxxx	User defined group quantity.
GWPI2	Group water potential injection rate (including values calculated on shut/stopped wells)
GWPP2	Group water potential production rate (including values calculated on shut/stopped wells)
SRRQR	River reach flow.
SRRQT	River reach cumulative flow
SRBQR	River branch flow
SRBQT	River branch cumulative flow
SRTQR	River total flow
SRTQT	River total cumulative flow
SRRFLOW	River reach flux.
SRRAREA	River reach cross sectional area.
SRRDEPTH	River reach depth.
SRREXCH	River reach exchange flux.
SRRFORDE	River reach Froude number.
RUxxx	User defined region quantity.
SUxxxxxx	User defined segment quantity.
WBHWCn	Derivative of well BHP with respect to GWRTWCV parameter n
WGFWCn	Derivative of well gas flow rate with respect to GWRTWCV parameter n
WGIP2	Well gas potential injection rate (including values calculated on shut/stopped wells)
WGPI2	Well gas potential injection rate (including values calculated on shut/stopped wells)
WGPP2	Well gas potential production rate (including values calculated on shut/stopped wells)
WGPPF2	Well free gas potential production rate (including values calculated on shut/stopped wells)
WGPPS2	Well solution (dissolved) gas potential production rate (including values calculated on shut/stopped wells)
WOFWCn	Derivative of well oil flow rate with respect to GWRTWCV parameter n

`WOPI2`	Well oil potential injection rate (including values calculated on shut/stopped wells)
`WOPP2`	Well oil potential production rate (including values calculated on shut/stopped wells)
`WUxxxxxx`	User defined well quantity.
`WWFWCn`	Derivative of well water flow rate with respect to `GWRTWCV` parameter n
`WWIP2`	Well water potential injection rate (including values calculated on shut/stopped wells)
`WWPI2`	Well water potential injection rate (including values calculated on shut/stopped wells)
`WWPP2`	Well water potential production rate (including values calculated on shut/stopped wells)

SCHEDULE section

DIFFMMF	Specify dual porosity matrix-fracture diffusivity multiplier here as well as in GRID section.
GUPFREQ	Specify update frequency of the instantaneous gradient option.
GWRTWCV	Specify gradient parameters as collections of wells for the instantaneous gradient option.
MULSGGD	Specify dual porosity alternative oil-gas matrix-fracture transmissibility multiplier
MULSGGDV	Specify dual porosity alternative oil-gas matrix-fracture transmissibility multipliers
MULTSIG	Specify dual porosity matrix-fracture transmissibility multiplier
MULTSIGV	Specify dual porosity matrix-fracture transmissibility multipliers
PLYVISC	Sets polymer solution viscosity function
PLYVISCS	Sets polymer/salt solution viscosity function
SURFVISC	Surfactant solution viscosity function
UDQ	Specify user defined quantities.
WELCNTL	Resets the control mode and operating target on wells
RIVERSYS	Defines each new river
REACHES,	Defines the reaches within a river
COMPRIV	Specifies grid block connections to a river
GRDREACH	Defines location of grid blocks connecting to a river

New keywords in ECLIPSE 300

RUNSPEC section

ASPHALTE	The ASPHALTE keyword has been added to activate the Asphaltene modeling
CO2STORE	Turns on the CO_2 storage option
GDIMS	Specify the number of instantaneous gradient parameters
JALS	New linear solver for thermal runs
LGRCOPY	Copy global values to local cells before operations within an LGR.
UDQDIMS	Dimensions for the user defined quantity facility.
UDQPARAM	Parameters for the user defined quantity facility.

GRID section

BTOBALFA	Activate new dual porosity transmissibilities from fracture blocks to the matrix block below, and specify a contact area multiplier.
BTOBALFV	Activate new dual porosity transmissibilities from fracture blocks to the matrix block below, and specify contact area multipliers.
COPYBOX	Copy values from one box of cells to another.
OPERNUM	Defines regions for performing operations using OPERATER keyword.

	OPERATER	Performs operations on arrays for specific regions.
	LGRCOPY	Copy global values to local cells before operations within an LGR.

EDIT section

	OPERATER	Performs operations on arrays for specific OPERNUM regions.
	LGRCOPY	Copy global values to local cells before operations within an LGR.

PROPS section

	LGRCOPY	Copy global values to local cells before operations within an LGR.
	COPYBOX	Copy values from one box of cells to another.
Thermal	KVAN	Specifies the use of the Antoine K-value correlation for use with THERMAL live oil component k-values.
	LBCCOEFR	Set non-default LBC coefficients for multiple equation of state regions.
Thermal	THSTT97	Requests the analytic IAPWS 97 steam table for use with THERMAL simulations.
	OPERATER	Performs operations on arrays for specific OPERNUM regions. In addition, it is possible to use arrays from different sections of the dataset, e.g. SWL = Fn(PORO).
	RKTRMDIR	Enables correct interpretation of five-column ROCKTAB tables from ECLIPSE 100. Columns four and five are ignored.
	SOLUBILT	Specify properties of water-CO_2 system with temperature dependencies.
	ALPHAD	Specify tables of transport coefficients for drainage processes.
	ALPHAI	Specify tables of transport coefficients for imbibition processes.
	ODD3P	Select the ODD3P relative permeability and capillary pressure hysteresis model
	EPSODD3P	Specify the end-point scaling, reference surface tensions and threshold interfacial tensions for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
	PCODD3P	Specify the capillary pressure and saturation calculation control parameters for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
	PCODD3PW	Specify the relationship between capillary pressure and water saturation as a tabulated function for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
	PCODD3PG	Specify the relationship between capillary pressure and gas saturation as a tabulated function for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
	PSORG	Specify the primary end-point (residual) gridblock oil saturations for oil-gas system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
	PSGRO	Specify the primary end-point (residual) gridblock gas saturations for gas-oil system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
	PSORW	Specify the primary end-point (residual) gridblock oil saturations for oil-water system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.

	PSWRO	Specify the primary end-point (residual) gridblock water saturations for water-oil system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
	PSGRW	Specify the primary end-point (residual) gridblock gas saturations for gas-water system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
	PSWRG	Specify the primary end-point (residual) gridblock water saturations for water-gas system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
	HSORG	Specify the hysteresis end-point (residual) gridblock oil saturations for oil-gas system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
	HSGRO	Specify the hysteresis end-point (residual) gridblock gas saturations for gas-oil system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
	HSORW	Specify the hysteresis end-point (residual) gridblock oil saturations for oil-water system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
	HSWRO	Specify the hysteresis end-point (residual) gridblock water saturations for water-oil system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
	HSGRW	Specify the hysteresis end-point (residual) gridblock gas saturations for gas-water system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
	HSWRG	Specify the hysteresis end-point (residual) gridblock water saturations for water-gas system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
Coal Bed Methane	LANGMULT	Langmuir multiplier, scales the adsorption capacities on a cell by cell basis.
Coal Bed Methane	ROCKPAMA	Palmer-Mansoori rock compaction model
Coal Bed Methane	ROCKPAME	Palmer-Mansoori alike rock compaction model, dependent on composition.
	DIFFCWAT	Water components diffusion coefficient for the CO2STORE or GASWAT option.
	ASPREWG	The ASPREWG keyword has been added to specify asphaltene precipitation data as molar percentage weight, when Asphaltene modeling is activated
	ASPFLOC	The ASPFLOC keyword has been added to specify asphaltene flocculation data, when Asphaltene modeling is activated
	ASPFLRT	Specifies asphaltene flocculation/dissociation rates, when Asphaltene modeling is activated.
	ASPDEPO	The ASPDEPO keyword has been added to specify asphaltene deposition data, when Asphaltene modeling is activated.
	ASPKDAM	The ASPKDAM keyword has been added to specify permeability damage due to asphaltene deposition, when Asphaltene modeling is activated
	ASPVISO	The ASPVISO keyword has been added to specify oil viscosity changes due to asphaltene precipitation, when Asphaltene modeling is activated

REGIONS section

LGRCOPY	Copy global values to local cells before operations within an LGR.
COPYBOX	Copy values from one box of cells to another.
OPERNUM	Defines regions for performing operations using OPERATER keyword.
OPERATER	Performs operations on arrays for specific regions.
ALPHANUD	Specify which set of transport coefficient tables should be used to calculate transport coefficients for drainage.
ALPHANUI	Specify which set of transport coefficient tables should be used to calculate transport coefficients for imbibition.
PSTNUM	Specify the gridblock primary saturation table numbers for the ODD3P three-phase relative permeability and capillary pressure hysteresis model option.
ISTNUM	Specify the gridblock increasing saturation table numbers for the ODD3P three-phase relative permeability and capillary pressure hysteresis model option.
DSTNUM	Specify the gridblock decreasing saturation table numbers for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
SDROG	Oil phase initial gridblock saturation direction indicator for the gas-oil system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
SDRGO	Gas phase initial gridblock saturation direction indicator for the gas-oil system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
SDROW	Oil phase initial gridblock saturation direction indicator for the oil-water system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
SDRWO	Water phase initial gridblock saturation direction indicator for the oil-water system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
SDRGW	Gas phase initial gridblock saturation direction indicator for the gas-water system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
SDRWG	Water phase initial gridblock saturation direction indicator for the gas-water system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.

SOLUTION section

AQUCHGAS	The property data for the constant head gas aquifer can now be specified using the AQUCHGAS keyword.
AQSTREAM	For compositional runs, this keyword should be used to specify the composition of the injected gas stream for the AQUCHGAS keyword.
OPERATER	Performs operations on arrays for specific OPERNUM regions. In addition, it is possible to use arrays from different sections of the dataset.
OPERATE	Performs operations on arrays for a specific box.
DBGODD3P	Debug information for the ODD3P three-phase relative permeability and capillary pressure hysteresis model option

SUMMARY section

ALL	Extension of ALL mnemonic to include FAQRG and FAQTG.
CUxxxxxx	User defined connection quantity.
FAQRG	Field aquifer influx rate for gas aquifer.
FAQTG	Cumulative field aquifer influx rate for gas aquifer.
FCVPP2	Field calorific value production potential (including values calculated on shut/stopped wells)
FGPI2	Field gas potential injection rate (including values calculated on shut/stopped wells)
FGPP2	Field gas potential production rate (including values calculated on shut/stopped wells)
FOPP2	Field oil potential production rate (including values calculated on shut/stopped wells)
FUxxxxxx	User defined field quantity.
FWPI2	Field water potential injection rate (including values calculated on shut/stopped wells)
FWPP2	Field water potential production rate (including values calculated on shut/stopped wells)
FCOMT	Component molar totals in the oil phase
FCGMT	Component molar totals in the gas phase
GCVPP2	Group calorific value production potential (including values calculated on shut/stopped wells)
GGPI2	Group gas potential injection rate (including values calculated on shut/stopped wells)
GGPP2	Group gas potential production rate (including values calculated on shut/stopped wells)
GOPP2	Group oil potential production rate (including values calculated on shut/stopped wells)
GUxxxxxx	User defined group quantity.
GWPI2	Group water potential injection rate (including values calculated on shut/stopped wells)
GWPP2	Group water potential production rate (including values calculated on shut/stopped wells)
GCOMT	Component molar totals in the oil phase
GCGMT	Component molar totals in the gas phase
RUxxx	User defined region quantity.
SUxxxxxx	User defined segment quantity.
WBHWCn	Derivative of well BHP with respect to GWRTWCV parameter n
WCVPP2	Well calorific value production potential (including values calculated on shut/stopped wells)
WGFWCn	Derivative of well gas flow rate with respect to GWRTWCV parameter n
WGIP2	Well gas potential injection rate (including values calculated on shut/stopped wells)
WGPI2	Well gas potential injection rate (including values calculated on shut/stopped wells)

WGPP2	Well gas potential production rate (including values calculated on shut/stopped wells)
WOFWCn	Derivative of well oil flow rate with respect to GWRTWCV parameter n
WOPP2	Well oil potential production rate (including values calculated on shut/stopped wells)
WUxxxxxx	User defined well quantity.
WWFWCn	Derivative of well water flow rate with respect to GWRTWCV parameter n
WWIP2	Well water potential injection rate (including values calculated on shut/stopped wells)
WWPI2	Well water potential injection rate (including values calculated on shut/stopped wells)
WWPP2	Well water potential production rate (including values calculated on shut/stopped wells)
WCOMT	Component molar totals in the oil phase
WCGMT	Component molar totals in the gas phase
BSDROG	Block oil phase saturation direction indicator for oil-gas system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
BSDRGO	Block gas phase saturation direction indicator for gas-oil system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
BSDROW	Block oil phase saturation direction indicator for oil-water system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
BSDRWO	Block water phase saturation direction indicator for water-oil system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
BSDRGW	Block gas phase saturation direction indicator for gas-water system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
BSDRWG	Block water phase saturation direction indicator for water-gas system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
BSONRM	Block normalized oil saturation for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
BSGNRM	Block normalized gas saturation for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
BSWNRM	Block normalized water saturation for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
BSOGTN	Block normalized oil turning point saturation for oil-gas system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
BSGOTN	Block normalized gas turning point saturation for gas-oil system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
BSOWTN	Block normalized oil turning point saturation for oil-water system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
BSWOTN	Block normalized water turning point saturation for water-oil system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
BSGWTN	Block normalized gas turning point saturation for gas-water system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.

BSWGTN	Block normalized water turning point saturation for water-gas system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
BSOGNH	Block normalized oil hysteresis saturation for oil-gas system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model
BSGONH	Block normalized gas hysteresis saturation for gas-oil system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
BSOWNH	Block normalized oil hysteresis saturation for oil-water system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
BSWONH	Block normalized water hysteresis saturation for water-oil system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
BSGWNH	Block normalized gas hysteresis saturation for gas-water system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
BSWGNH	Block normalized water hysteresis saturation for water-gas system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
BSOGNE	Block normalized oil equivalent opposite direction saturation for oil-gas system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
BSGONE	Block normalized gas equivalent opposite direction saturation for gas-oil system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
BSOWNE	Block normalized oil equivalent opposite direction saturation for oil-water system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
BSWONE	Block normalized water equivalent opposite direction saturation for water-oil system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
BSGWNE	Block normalized gas equivalent opposite direction saturation for gas-water system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
BSWGNE	Block normalized water equivalent opposite direction saturation for water-gas system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
BSORGB	Residual oil saturation used to calculate normalised saturations in oil-gas system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
BSGROB	Residual gas saturation used to calculate normalised saturations in gas-oil system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
BSORWB	Residual oil saturation used to calculate normalised saturations in oil-water system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
BSWROB	Residual water saturation used to calculate normalised saturations in water-oil system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
BSGRWB	Residual gas saturation used to calculate normalised saturations in gas-water system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
BSGRWB	Residual water saturation used to calculate normalised saturations in water-gas system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.

BSORGP	Process dependent residual oil saturation in oil-gas system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
BSGROP	Process dependent residual gas saturation in gas-oil system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
BSORWP	Process dependent residual oil saturation in oil-water system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
BSWROP	Process dependent residual water saturation in water-oil system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
BSGRWP	Process dependent residual gas saturation in gas-water system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
BSWRGP	Process dependent residual water saturation for water-gas system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
BIFTGO	Block gas-oil interfacial tension for the ODD3P three-phase relative permeability and capillary pressure hysteresis model
BIFTWO	Block water-oil interfacial tension for the ODD3P three-phase relative permeability and capillary pressure hysteresis model
BIFTGW	Block gas-water interfacial tension for the ODD3P three-phase relative permeability and capillary pressure hysteresis model
BKROGN	Block representative oil-gas relative permeability for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
BKRGON	Block representative gas-oil relative permeability for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
BKROWN	Block representative oil-water relative permeability for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
BKRWON	Block representative water-oil relative permeability for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
BKRGWN	Block representative gas-water relative permeability for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
BKRWGN	Block representative water-gas relative permeability for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
BKROGH	Block turning point relative permeability to oil for oil-gas system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
BKRGOH	Block turning point relative permeability to gas for gas-oil system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
BKROWH	Block turning point relative permeability to oil for oil-water system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
BKRWOH	Block turning point relative permeability to water for water-oil system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
BKRGWH	Block turning point relative permeability to gas for gas-water system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
BKRWGH	Block turning point relative permeability to water for water-gas system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
BKROGE	Block equivalent relative permeability to oil for oil-gas system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.

BKRGOE	Block equivalent relative permeability to gas for gas-oil system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
BKROWE	Block equivalent relative permeability to oil for oil-water system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
BKRWOE	Block equivalent relative permeability to water for water-oil system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
BKRGWE	Block equivalent relative permeability to gas for gas-water system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
BKRWGE	Block equivalent relative permeability to water for water-gas system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
BKROGT	Block opposite direction turning point relative permeability to oil for oil-gas system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
BKRGOT	Block opposite direction turning point relative permeability to gas for gas-oil system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
BKROWT	Block opposite direction turning point relative permeability to oil for oil-water system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
BKRWOT	Block opposite direction turning point relative permeability to water for water-oil system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
BKRGWT	Block opposite direction turning point relative permeability to gas for gas-water system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.
BKRWGT	Block opposite direction turning point relative permeability to water for water-gas system for the ODD3P three-phase relative permeability and capillary pressure hysteresis model.

SCHEDULE section

AQUCHGAS	The property data for the constant head gas aquifer can now be re-specified using the AQUCHGAS keyword in the SCHEDULE section.
AQSTREAM	For compositional runs, this keyword should be used to specify the composition of the injected gas stream for the AQUCHGAS keyword.
DIFFMMF	Specify dual porosity matrix-fracture diffusivity multiplier here as well as in GRID section.
GUPFREQ	Specify update frequency of the instantaneous gradient option.
GWRTWCV	Specify gradient parameters as collections of wells for the instantaneous gradient option.
MULSGGD	Specify dual porosity alternative oil-gas matrix-fracture transmissibility multiplier
MULSGGDV	Specify dual porosity alternative oil-gas matrix-fracture transmissibility multipliers
MULTSIG	Specify dual porosity matrix-fracture transmissibility multiplier
MULTSIGV	Specify dual porosity matrix-fracture transmissibility multipliers
UDQ	Specify user defined quantities.
WELCNTL	Resets the control mode and operating target on wells
WLISTARG	This keyword, previously only available in ECLIPSE 100, is now available in ECLIPSE 300. This is used to reset operating targets or limits for well lists.
WREGROUP	This keyword, previously only available in ECLIPSE 100, is now available in ECLIPSE 300. This is used to transfer wells automatically from one group to another, depending on the value of a chosen quantity such as the well's water cut or oil production rate.
DBGODD3P	Debug information for the ODD3P three-phase relative permeability and capillary pressure hysteresis model option

Altered keywords in ECLIPSE 100

RUNSPEC section

PSTEADY	Modified (data item 11) to add lower limit for time-stepping to ensure calculation doesn't end too soon.
REGDIMS	The seventh item of this keyword can now be used to set the maximum number of user-defined regions used by the OPERNUM keyword.
PETOPTS	This keyword has been extended to use a new item of data, LGRSIMGD. Setting this mnemonic implies that the .GSG file imported using the PETGRID keyword contains local grid refinements. For the 2007.1 release, the import of .GSG files is not supported in ECLIPSE 100. This is for future use when Petrel is able to export the .GSG format. Please refer to Petrel documentation and release notes.

GRID section

ADDREG	It is now also possible to use the OPERNUM keyword to perform this operation on a specified array.
EQUALREG	It is now also possible to use the OPERNUM keyword to perform this operation on a specified array.

MULTIREG	It is now also possible to use the OPERNUM keyword to perform this operation on a specified array.
OPERATE	It is now possible to apply a maximum and minimum limit to a property array. In addition, a new multiplication operator has been added.
PETGRID	Imports grid from a .GSG file. For the 2007.1 release, the import of .GSG files is not supported in ECLIPSE 100. This is for future use when Petrel is able to export the .GSG format. Please refer to Petrel documentation and release notes.
RPTGRID	Output of regions defined for performing operations on arrays can be achieved by entering mnemonic OPERNUM.

EDIT section

OPERATE	It is now possible to apply a maximum and minimum limit to a property array. In addition, a new multiplication operator has been added.

PROPS section

OPERATE	It is now possible to apply a maximum and minimum limit to a property array. In addition, a new multiplication operator has been added. It is also possible to use arrays from different sections of the dataset, for example SWL = Fn(PORO).
ROCKTAB	If the RKTRMDIR keyword has been specified, each table consists of 5 columns of data, the last 3 being the transmissibility multipliers in the x-, y- and z-directions.

REGIONS section

OPERATE	It is now possible to apply a maximum and minimum limit to a property array. In addition, a new multiplication operator has been added.
TNUM	These arrays can now be used in conjunction with the ADD, ADDREG, MULTIPLY and MULTIREG operations.
TRKPF	These arrays can now be used in conjunction with the ADD, ADDREG, MULTIPLY and MULTIREG operations.

SOLUTION section

AQUCHWAT	Two additional items have been added to the AQUCHWAT keyword to allow the flow to be limited between a minimum and maximum value.
OPERATE	It is now possible to apply a maximum and minimum limit to a property array. In addition, a new multiplication operator has been added. It is also possible to use arrays from different sections of the dataset. This keyword can now be used in the SOLUTION section.
RPTRST	A new TRAS mnemonic has been added which outputs the product of tracer concentration with the carrier saturation to the restart file.

SCHEDULE section

ACTIONX	All field, group, well, segment and connection summary section quantities are now available for use as triggers within an ACTIONX keyword
AQUCHWAT	Two additional items have been added to the AQUCHWAT keyword to allow the flow to be limited between a minimum and maximum value.
WCONINJE	A new 'MULTI' option is available for the injector type (item 2). This specifies that the well is a multi-phase injector and can inject more than one phase simultaneously. New items 12, 13 and 14 are used to specify the surface volume proportions of the oil, water and gas phase respectively.
WCONINJH	A new 'MULTI' option is available for the injector type (item 2). This specifies that the well is a multi-phase injector and can inject more than one phase simultaneously. New items 9, 10 and 11 are used to specify the surface volume proportions of the oil, water and gas phase respectively.

WCONINJP	A new 'MULTI' option is available for the injector type (item 2). This specifies that the well is a multi-phase injector and can inject more than one phase simultaneously. New items 9, 10 and 11 are used to specify the surface volume proportions of the oil, water and gas phase respectively.
WLISTARG	This keyword now accepts default values specified for new operating targets and limits for well lists. These default values have the same behavior as those in the WELARG keyword (that is, the simulator will use the corresponding flowing value for the well at the end of the previous timestep as the target or limit).
RPTRST	A new TRAS mnemonic has been added which outputs the product of tracer concentration with the carrier saturation to the restart file.

Switch 97	If set to a non-zero value, then this option restores the pre-2007.1 behavior of some sensitive constants within the program. At 2007.1, the precision of these program constants was increased to reduce cross-platform (and cross-version) differences in behavior.
Switch 99	Generates warning messages from the `AQUANCON` keyword when cells cannot be connected to the aquifer because there is an active cell on the aquifer side of the current cell.
Switch 100	If > 0, sets up the gap between two consecutive RS values for the interpolation when API Tracking is used, and uses an improved interpolation to avoid crossing lines in PVTO. The gap is a percentage given by RSGAP=0.01*`OPTIONS`(100). Note that this gap should not be too large. A minimum value of 0.01 is set.
Switch 101	If > 0, this restores various aspects of pre-2007.1 behavior related to well potential calculations. If a multiple of 2, this reverses a fix in the allocation of work-space for well potential calculations. This fix only affects models which use an old-style `RUNSPEC` section, and it may result in a change in the reported well quantities in such models. The new behavior can also be produced with pre-2007.1 code by converting the model to use a new-style `RUNSPEC` section (for example, using ECLIPSE Office). If a multiple of 3, this restores the logic for only calculating potentials on wells for which they are required. If a multiple of 5, this reverses a fix in the preservation of the actual well control mode when well potentials are calculated for the prioritization option (see the `PRIORITY` keyword) or for prioritized drilling queues (see the `WDRILPRI` keyword).
Switch 102	If set to a non-zero value, the pre-2007.1 implementation of the `BRINE` model is restored. This affects the PVT calculations in the reservoir and wells.
Switch 103	This controls aspects of the `GCONSALE` keyword. If set to a value greater than zero value, this restores pre-2007.1 behavior. In cases where a net sales, import and consumption target of zero has been specified, this sets the `MAXR` procedure re-injection fraction limit to a value of 1, instead of the new default value of 100. It also restores the handling of the lower limit in which a non-positive lower limit was taken to mean that no lower limit was to be applied. If set to a value less than zero, this sets the `MAXR` procedure re-injection fraction limit to `-OPTIONS(103)` in cases where a net sales, import and consumption target of zero has been specified.
Switch 104	If set to a non-zero value, the pre-2007.1 implementations of the scale and gas calorific value models are restored. Prior to 2007.1 the tracer concentrations used in the scale and gas calorific value calculations were lagged by two timesteps. From 2007.1 the lag has been reduced to a single timestep.
Switch 105	If set to a non-zero value, the pre-2007.1 use of group efficiency factors (specified with the `GEFAC` keyword) in reservoir coupling runs is restored. At 2007.1, various fixes were made to this part of the code.
Switch 106	If set to a non-zero value, the pre-2007.1 gravity drainage behavior is restored. At 2007.1, a bug was fixed which may cause a change in results for models that contain the `GRAVDR` keyword.

DEBUG keyword

Switch 76	Outputs debug from the river model

Altered keywords in ECLIPSE 300

RUNSPEC section

CBMOPTS	A new option (data item 2) can be used to disallow water flow in and out of coal cells.
VELDEP	Item 5 can now be set to 2, which indicates that the alternative model for Capillary Number effects in gas condensate reservoirs should be applied to the calculation of the oil phase relative permeability.
REGDIMS	The seventh item of this keyword can now be used to set the maximum number of user-defined regions used by the OPERNUM keyword.
PETOPTS	This keyword has been extended to use a new item of data, LGRSIMGD. Setting this mnemonic implies that the .GSG file imported using the PETGRID keyword contains local grid refinements. For the 2007.1 release, the import of .GSG files is not supported in ECLIPSE 300. This is for future use when Petrel is able to export the .GSG format. Please refer to Petrel documentation and release notes.
TABDIMS	The TABDIMS keyword has been extended with item 17,18,19 and 20 to specify the number of rows in look-up tables used for asphaltene modeling and the maximum number of flocculation allowed.

GRID section

ADDREG	It is now also possible to use the OPERNUM keyword to perform this operation on a specified array.
COPYREG	It is now also possible to use the OPERNUM keyword to perform this operation on a specified array.
EQUALREG	It is now also possible to use the OPERNUM keyword to perform this operation on a specified array.
MULTIREG	It is now also possible to use the OPERNUM keyword to perform this operation on a specified array.
OPERATE	It is now possible to apply a maximum and minimum limit to a property array. In addition, a new multiplication operator has been added.
PETGRID	Imports grid from a .GSG file. For the 2007.1 release, the import of .GSG files is not supported in ECLIPSE 300. This is for future use when Petrel is able to export the .GSG format. Please refer to Petrel documentation and release notes.
RPTGRID	Output of regions defined for performing operations on arrays can be achieved by entering mnemonic OPERNUM.
RPTGRID	Output of coal regions can be achieved by entering mnemonic COALNUM.

EDIT section

ADDREG	It is now also possible to use the OPERNUM keyword to perform this operation on a specified array.
COPYREG	It is now also possible to use the OPERNUM keyword to perform this operation on a specified array.

`EQUALREG`	It is now also possible to use the `OPERNUM` keyword to perform this operation on a specified array.
`MULTIREG`	It is now also possible to use the `OPERNUM` keyword to perform this operation on a specified array.
`OPERATE`	It is now possible to apply a maximum and minimum limit to a property array. In addition, a new multiplication operator has been added.

PROPS section

ADDREG	It is now also possible to use the OPERNUM keyword to perform this operation on a specified array.
COPYREG	It is now also possible to use the OPERNUM keyword to perform this operation on a specified array.
EQUALREG	It is now also possible to use the OPERNUM keyword to perform this operation on a specified array.
MULTIREG	It is now also possible to use the OPERNUM keyword to perform this operation on a specified array.
OPERATE	It is now possible to apply a maximum and minimum limit to a property array. In addition, a new multiplication operator has been added.
SOLUBILI	The columns of this keyword can now be defaulted. The default values for the formation volume factor are calculated using a more accurate method. This keyword can be used with SALINITR and RTEMP to specify the salinity and temperature on a regional basis. There is no limit on the number of records anymore.
SOLUBILS	A new item 3 has been added to record 1 to define the reference salt concentration for stock tank water. There is no limit on the number of records anymore.
RPTPROPS	The mnemonics LANGMULT, ROCKPAMA and ROCKPAME have been added outputting the Langmuir multipliers and the rock compaction data.
RPTPROPS	The RPTPROPS keyword has been extended for the ODD3P tables and saturation end-points with the following mnemonics; PSORG, PSGRO, PSORW, PSWRO, PSGRW, PSWRG, HSORG, HSGRO, HSORW, HSWRO, HSGRW, HSWRG, EPSODD3P, PCODD3P, PCODD3PG and PCODD3PW

REGIONS section

ADDREG	It is now also possible to use the OPERNUM keyword to perform this operation on a specified array.
COPYREG	It is now also possible to use the OPERNUM keyword to perform this operation on a specified array.
EQUALREG	It is now also possible to use the OPERNUM keyword to perform this operation on a specified array.
MULTIREG	It is now also possible to use the OPERNUM keyword to perform this operation on a specified array.
OPERATE	It is now possible to apply a maximum and minimum limit to a property array. In addition, a new multiplication operator has been added.
RPTREGS	The RPTREGS keyword has had the following mnemonics added for the ODD3P option to output saturation region numbers and direction indicators; PSTNUM, ISTNUM, DSTNUM, SDROW, SDRWO, SDROG, SDRGO, SDRGW and SDRWG.

SOLUTION section

ADDREG	It is now also possible to use the OPERNUM keyword to perform this operation on a specified array.
AQUANCON	Connections to the Constant Head Gas aquifer (AQUCHGAS) are read in using this keyword.

AQUCHWAT	Two additional items have been added to the AQUCHWAT keyword to allow the flow to be limited between a minimum and maximum value. For the GASWAT or CO2STORE options, item 6 refers to an EoS region number.
AQUCT	For the GASWAT or CO2STORE options, item 10 refers to an EoS region number.
AQUFETP	For the GASWAT or CO2STORE options, item 6 refers to an EoS region number.
COPYREG	It is now also possible to use the OPERNUM keyword to perform this operation on a specified array.
EQUALREG	It is now also possible to use the OPERNUM keyword to perform this operation on a specified array.
MULTIREG	It is now also possible to use the OPERNUM keyword to perform this operation on a specified array.
OPERATE	It is now possible to apply a maximum and minimum limit to a property array. In addition, a new multiplication operator has been added. It is also possible to use arrays from different sections of the dataset. This keyword can now be used in the SOLUTION section.
RPTRST	A new TRMFxxxx mnemonic which controls the output of the product of tracer concentration with the molar fraction of the carrier to the restart file has been added. The tracer name to be output must be appended to the end of the TRMF part of the mnemonic.
RPTRST	PERMSTAB and BIOTCTAB to output permeability and Biot's constant multipliers based on the stress dependence of those two variables (see keyword PERMSTAB and BIOTCTAB respectively
RPTRST	When using the Hysteresis option the critical gas saturation and the trapped gas saturation (immobile) can be output using the mnemonics SGCRH and SGTRH.
RPTRST	The aqueous speciation and pH-level can be output when using the CO2STORE option with the mnemonics AQSP and AQPH.
RPTSOL	New RPTSOL mnemonics have been added for the ODD3P three-phase relative permeability and capillary pressure hysteresis model option. The list of new mnemonics consists of: SONRM, SGNRM, SWNRM, SDROW, SDRWO, SDROG, SDRGO, SDRGW, SDRWG, SOWNT, SWONT, SOGNT, SGONT, SGWNT, SWGNT, SOWNH, SWONH, SOGNH, SGONH, SGWNH, SWGNH, SOWNE, SWONE, SOGNE, SGONE, SGWNE, SWGNE, SORWB, SWROB, SORGB, SGROB, SGRWB, SWRGB, SORWP, SWROP, SORGP, SGROP, SGRWP, SWRGP, IFTGO, IFTOW, IFTGW, KROWN, KRWON, KROGN, KRGON, KRGWN, KRWGN, KROWH, KRWOH, KROGH, KRGOH, KRGWH, KRWGH, KROWE, KRWOE, KROGE, KRGOE, KRGWE, KRWGE, KROWT, KRWOT, KROGT, KRGOT, KRGWT and KRWGT.
RPTSOL	PERMSTAB and BIOTCTAB to output permeability and Biot's constant multipliers based on the stress dependence of those two variables (see keyword PERMSTAB and BIOTCTAB respectively
RPTSOL	Output of analytic data in the SOLUTION section can be achieved by entering mnemonic AQUCHGAS
RPTSOL	Output of pore volume and permeability multipliers can be achieved by entering mnemonics PORV_MOD and PERM_MOD.
RPTSOL	Output of liquid and vapor mass fractions for each component can be achieved by entering the mnemonics XFW and YFW.

SCHEDULE section

	ACTIONX	All field, group, well, segment and connection summary section quantities are now available for use as triggers within an ACTIONX keyword
	ACTIONX	Almost all SCHEDULE keywords can now be used in an ACTIONX/ENDACTIO block
Thermal	AIMCON	Two parameters have been added to control the switch from explicit to implicit. They are THERMAL specific.
	AQUCHWAT	Two additional items have been added to the AQUCHWAT keyword to allow the flow to be limited between a minimum and maximum value.
Thermal	CVCRIT	Three parameters have been added to control the converge of the Multi-Segment well model.
	GCONPRI	New items 14 and 15 allow you to specify the surface gas/water volume production balancing fraction upper limit.
	GCONPROD	Two new group production rate control modes, PBGS and PBWS, have been added to item 2. These modes allow you to limit a group's available surface gas volume rate, or surface water production volume rate, to the specified fraction (or multiple) of the group's surface gas/water volume injection rate. New items 18 and 19 allow you to specify the surface gas/water production balancing fraction target or upper limit.
	GINJGAS	When injection fluid nature 'GAS' is specified in item 2, it is now internally converted to 'GV' from the field separator. In this case, items 3 (source of injection) and 5 (separator stage) are ignored and assumed to be FIELD and 0 respectively. This gives better reinjection and gas accounting behavior. OPTIONS3 item 113 can be used to recover the pre-2007.1 behavior.
	GRUPINJE	When injection fluid nature 'GAS' is specified in item 2, it is now internally converted to 'GV' from the field separator if this is a compositional run. In this case, items 3 (source of injection) and 16 (separator stage) are ignored and assumed to be FIELD and 0 respectively. This gives better reinjection and gas accounting behavior. OPTIONS3 item 113 can be used to recover the pre-2007.1 behavior.
	GRUPTARG	PBGS and PBWS have been added to the list of items that can be reset. These reset the surface gas volume production balancing fractions for gas and water respectively.
	GTADD	PBGS and PBWS have been added to the list of items that can be modified. These add to the surface gas volume production balancing fractions for gas and water respectively. PRBL has also been added for production balancing.
	GTMULT	PBGS and PBWS have been added to the list of items that can be modified. These multiply the surface gas volume production balancing fractions for gas and water respectively. PRBL has also been added for production balancing.
	PICOND	Two new items have been added to the PICOND keyword to allow you to specify the lower and upper bounds for use in constructing the generalized pseudo-pressure integrals. These bounds are specified as dimensionless factors which are applied to the completion cell pressure to generate the lower and upper pressures for the table. They allow the use of smaller quadrature intervals by allowing control over the pressure range of the table.
	RPTRST	A new TRMFxxxx mnemonic which outputs the product of tracer concentration with the molar fraction of the carrier to the restart file has been added. The tracer name to be output must be appended to the end of the TRMF part of the mnemonic.

	RPTRST	PERMSTAB and BIOTCTAB to output permeability and Biot's constant multipliers based on the stress dependence of those two variables (see keyword PERMSTAB and BIOTCTAB respectively
	RPTRST	When using the Hysteresis option the critical gas saturation and the trapped gas saturation (immobile) can be output using the mnemonics SGCRH and SGTRH.
	RPTRST	The aqueous speciation and pH-level can be output when using the CO2STORE option with the mnemonics AQSP and AQPH.
	RPTRST	Output of pore volume and permeability multipliers can be achieved by entering the mnemonics PORV_MOD and PERM_MOD or ROCKC (both) when using the rock compaction mode (ROCKCOMP or ROCKPAME)
	RPTRST	Output of liquid and vapor mass fractions for each component can be achieved by entering the mnemonics XFW and YFW.
	RPTSCHED	New RPTSCHED mnemonics have been added for the ODD3P three-phase relative permeability and capillary pressure hysteresis model option. The list of new mnemonics consists of: SONRM, SGNRM, SWNRM, SDROW, SDRWO, SDROG, SDRGO, SDRGW, SDRWG, SOWNT, SWONT, SOGNT, SGONT, SGWNT, SWGNT, SOWNH, SWONH, SOGNH, SGONH, SGWNH, SWGNH, SOWNE, SWONE, SOGNE, SGONE, SGWNE, SWGNE, SORWB, SWROB, SORGB, SGROB, SGRWB, SWRGB, SORWP, SWROP, SORGP, SGROP, SGRWP, SWRGP, IFTGO, IFTOW, IFTGW, KROWN, KRWON, KROGN, KRGON, KRGWN, KRWGN, KROWH, KRWOH, KROGH, KRGOH, KRGWH, KRWGH, KROWE, KRWOE, KROGE, KRGOE, KRGWE, KRWGE, KROWT, KRWOT, KROGT, KRGOT, KRGWT and KRWGT.
	RPTSCHED	PERMSTAB and BIOTCTAB to output permeability and Biot's constant multipliers based on the stress dependence of those two variables (see keyword PERMSTAB and BIOTCTAB respectively)
	RPTSCHED	Output of analytic data in the SCHEDULE section can be achieved by entering mnemonic AQUCHGAS
	RPTSCHED	When using the Hysteresis option the critical gas saturation and the trapped gas saturation (immobile) can be output using the mnemonics SGCRH and SGTRH.
	RPTSCHED	The aqueous speciation and pH-level can be output when using the CO2STORE option with the mnemonics AQSP and AQPH.
	RPTSCHED	Output of pore volume and permeability multipliers can be achieved by entering mnemonics PORV_MOD and PERM_MOD
	RPTSCHED	Output of liquid and vapor mass fractions for each component can be achieved by entering the mnemonics XFW and YFW.
	WCONINJE	A new 'MULTI' option is available for the injector type (item 2). This specifies that the well is a multi-phase injector and can inject more than one phase simultaneously. New items 13 and 14 are used to specify the surface volume proportions of the water and gas phase respectively.
CO2STORE *option*	WCONINJE	The default BHP limit have changed to 592.l5 atma when using the CO2STORE option.
	WCONINJH	A new 'MULTI' option is available for the injector type (item 2). This specifies that the well is a multi-phase injector and can inject more than one phase simultaneously. New items 10 and 11 are used to specify the surface volume proportions of the water and gas phase respectively.

	WCONINJP	A new 'MULTI' option is available for the injector type (item 2). This specifies that the well is a multi-phase injector and can inject more than one phase simultaneously. New items 10 and 11 are used to specify the surface volume proportions of the water and gas phase respectively.
CO2STORE *option*	WCONINJP	The default BHP limit have changed to 592.l5 atma when using the CO2STORE option.
	WELLINJE	When injection fluid nature 'GAS' is specified in item 2, it is now internally converted to 'GV' from the field separator if this is a compositional run. In this case, items 3 (source of injection) and 19 (separator stage) are ignored and assumed to be FIELD and 0 respectively. This gives better reinjection and gas accounting behavior. OPTIONS3 item 113 can be used to recover the pre-2007.1 behavior.
	WINJGAS	When injection fluid nature 'GAS' is specified in item 2, it is now internally converted to 'GV' from the field separator. In this case, items 3 (source of injection) and 5 (separator stage) are ignored and assumed to be FIELD and 0 respectively. This gives better reinjection and gas accounting behavior. OPTIONS3 item 113 can be used to recover the pre-2007.1 behavior.
	WINJMIX	When injection fluid nature 'GAS' is specified in item 4, it is now internally converted to 'GV' from the field separator. In this case, items 5 (source of injection) and 6 (separator stage) are ignored and assumed to be FIELD and 0 respectively. This gives better reinjection and gas accounting behavior. OPTIONS3 item 113 can be used to recover the pre-2007.1 behavior.
	WINJORD	When injection fluid nature 'GAS' is specified in item 3, it is now internally converted to 'GV' from the field separator. In this case, items 4 (source of injection) and 5 (separator stage) are ignored and assumed to be FIELD and 0 respectively. This gives better reinjection and gas accounting behavior. OPTIONS3 item 113 can be used to recover the pre-2007.1 behavior.
CO2STORE *option*	TSCRIT	Item 21 can be used to specify a target solid saturation change.

OPTIONS3 keyword

Switch 83	Post 2007.1 the multi-segment well model uses the modified solution update method by default. The modified method can improve convergence for difficult problems, but may be slower in more straightforward cases. If=-1 the pre-2007.1 multi-segment well update method is restored.
Switch 99	The alternative model for Capillary Number effects in gas condensate reservoirs, which is selected by setting item 5 of the VELDEP keyword to a non-zero value, has been modified to be more faithful to the ideas in Whitson, Fevang and Saevareid [Ref. 99]. A description of both the old and new behavior is given in "Non-Darcy Flow". The pre-2007.1 behavior can be restored by setting this item to a non-zero value.
Switch 104	If this item is set to a non-zero value, then the PBGS surface volume production balancing group control mode balances the *produced* surface gas, instead of the *excess* surface gas, to the specified fraction (or multiple) of the injected surface water volume.
Switch 105	Restores pre-2007.1 behavior regarding tilted aquifer equilibration and the calculation of the block equilibration pressures.

	Switch 106	If this item is set to a value greater than 5000, then the maximum number of internal arrays that can be allocated is increased to the value specified. Previously there was a hard-wired limit of 5000 on the number of such arrays and the run fails if this is exceeded. This situation can arise, for example, if you have a large number of equilibration regions.
	Switch 107	Restores pre-2007.1 behavior of the drift flux model for small flows and hold-ups. If > 0, the 2006.2 behavior is restored; this affects the drift flux calculations in the presence of small values of flow rate and hold-up. If > 1, the pre-2006.2 behavior is restored; this only affects the reporting of segment phase volume flow rates and velocities for small hold-ups.
	Switch 108	By default, injected gas during the surface gas production balancing group control mode is reported as re-injected gas, and the reported excess and export gas is based on this injected gas (and not just gas formally re-injected during a re-injection group control mode). If this item is non-zero, then this production balancing injected gas is **not** treated as re-injected gas. This affects the reporting of re-injected, excess and export gas in the gas accounting tables in the PRT file, and also in the values of these quantities written to the summary files.
	Switch 110	Negative capillary pressure handling in gas water systems. If >0 switch back to pre-2006.2 behavior
	Switch 111	If set to 1, restores pre-2007.1 behavior for high RV (vapor oil ratio). This can affect the nature of the single phase (oil or gas) seen by the simulator.
	Switch 112	This option must be set in the `RUNSPEC` section. If set to a non-zero value, this reverts to the pre-2007.1 default behavior of reporting mass rates and totals in **long tons** (instead of **short tons**).
	Switch 113	If set to various non-zero values, this restores aspects of the pre-2007.1 behavior regarding injection of gas and mixtures. If divisible by 2, use the pre-2007.1 calculation of availability of items in mixtures specified using `WINJORD`. If divisible by 3, use availability fractions when calculating mixture requirements. (These availability fractions are not used from 2007.1, instead a new calculation of the availability is performed.) If divisible by 5, use the pre-2007.1 calculation of mixture potentials when calculating the availability fractions for mixture requirements. (Bugs were fixed in this calculation in 2007.1 to prevent crashes.) If divisible by 7, use the pre-2007.1 treatment of gas specified using the injected fluid nature '`GAS`' in the `GINJGAS`, `GRUPINJE`, `WELLINJE`, `WINJGAS`, `WINJMIX` and `WINJORD` keywords. From 2007.1 these are internally converted to injection fluid nature '`GV`' with its source being the field separator (which has better gas accounting and re-injection behavior). If divisible by 11, use the pre-2007.1 treatment of re-injection for mixture items from a separator stage. (Before 2007.1, these mixture items were effectively ignored in the complex gas accounting that is used when there is fuel and/or sales and separator stage gas is used for re-injection.) If divisible by 13, complex gas accounting (that is, where there is fuel and/or sales gas, with intermediate separator stage gas re-injection) uses the pre-2007.1 treatment of re-injection in calculations of excess and export gas. If divisible by 17, gas from a satellite group is not reported as separator stage 1 gas for the group.
`CO2STORE`- option	Switch 114	If set to a nonzero value, the max NaCl solubility in water is computed using a simple model - ignoring the CaCl concentration.

CO2SOL- *option*	Switch 115	If set to 1, the default values of the water formation volume factor of the solubility table used in the CO2SOL option are calculated using the pre-2007.1 method. This affects the water density calculation.
	Switch 116	If set to a non-zero value the pre-2007.1 implementation of the BRINE model is restored. This affects the equilibration of the initial solution.
	Switch 117	If set to a non-zero value the computed rock volumes will be scaled by the NTG values. This is the pre-2007.1 behavior.
	Switch 118	If set to a non-zero value various aspects of the pre-2007.1 ACTION(G,R,W,S,X) implementation are restored. The default behavior from 2007.1 onwards is that ACTION(R,X) conditions involving regions are evaluated using the summary quantities from the current timestep, the maximum timestep length after a well modification is respected, and the summary data is output before processing the ACTION(G,R,W,S,X) conditions. If set to 1 the pre-2007.1 behavior of the ACTION(G,R,W,S,X) keywords is restored. Specifically, the ACTION(R,X) conditions are tested before the region summary quantities such as ROIP are updated, and before the summary quantities are output. As a result summary data from the previous timestep can be overwritten when an ACTION(G,R,W,S,X) is triggered. In addition the maximum timestep length after a well modification is not imposed, see item 11 of TSCRIT keyword, and item 10, record 1 of the TUNING keyword. If the user defined quantity feature is active then these quantities will also be calculated before the region summary quantities are updated. If set to -1 the pre-2007.1 time stepping sequence can be recovered. Specifically, the 2007.1 order of evaluating the ACTION(G,R,W,S,X) conditions and the output of the summary data is followed, but the maximum timestep length after a well modification is not imposed.
	Switch 119	If set to 1, the water properties of analytical aquifers are calculated using the PVT tables for the GASWAT or CO2STORE options. This was the pre-2007.1 method. By default, the water properties are calculated using the EOS.
	Switch 120	If set to a positive number, the pre-2006.2 handling of dual porosity system is restored. Otherwise, active matrix cells that are connected to inactive fracture cells (because of a MINPV criterion) are inactivated.
	Switch 121	If set to a positive number, that number is taken as an upper limit for the blocking factor calculated by the generalized pseudo-pressure special inflow equation. If a zero (i.e. default), or negative value is specified, then no limit is applied. This can be used to prevent convergence problems when blocking factors greater than unity are generated. See "Generalized pseudo-pressure in ECLIPSE 300" for more information.
	Switch 122	Generates warning messages from the AQUANCON keyword when cells cannot be connected to the aquifer because there is an active cell on the aquifer side of the current cell.
	Switch 123	If a zero (i.e. default), or negative value is specified, then no normalization is applied. If set to 1, then normalization of the track tracers concentration is applied for each well and the calculation of injected track tracers concentration is also performed. This can be used to prevent discrepancy problems between field tracers' characteristics and field characteristics. The injected track tracers' concentration can be monitored in a similar way as ordinary track tracers. The naming of the injected tracers is done automatically in a similar manner (e.g. AI1, BI2). If the 'REGONLY' option is used then the tracer AI1 refers to region A and injected component 1, BI2 to region B and injected component 2, etc. For example, mnemonic FTTLBI2 can be used in the summary section to monitor FTTL for the injected tracer BI2.
GASWAT *option*	Switch 125	If set to a positive number enhance phase stability checks.

OPTIMIZE section

RPTOPT	A new CLRFILES mnemonic has been added. This removes SEPARATE results files from any previous optimization for this model if necessary (that is, if the current optimization is about to overwrite some or all of them).

New Developments in 2006.1 and 2006.2

New facilities in ECLIPSE 100

RUNSPEC section

- The BPIDIMS keyword has been added to define the number of SUMMARY off-centered interpolation outputs.
- The PETOPTS keyword has been added in order to set options for an ECLIPSE 100 run fired from Petrel.

Aquifers

- AQUCHWAT: New item 10 allows the user to turn off capillary pressure effects.

Output and restarts

- If outputting block heads the depth terms can be dropped from the calculation using item 3 of HYDRHEAD.
- Individual phase velocities can be output using the mnemonics VELOIL, VELWAT or VELGAS instead of VELOCITY in the RPTRST keyword.
- Pore volumes can now be output using the RPORV mnemonic in the RPTRST keyword.
- The initial coal gas saturated content (for the coal bed methane option) can be output to the PRT file using the GASSATC mnemonic in the RPTSOL keyword.
- Norms of the residuals can be output with DEBUG switch 79.
- Shear viscosity in the Polymer Flood option (PLYSHEAR) can now be output using the VISC or ALLPROPS mnemonic in the RPTRST keyword.
- Polymer concentrations can now be output in the summary section using mnemonics (F,G,W,C)C(P,I)C
- Data from the end of the first pass of the Gas Field Operations model can be output in summary section using the mnemonics WGPRFP, WTHPFP, WBHPFP, GPRFP, GGPRNBFP.
- The CPU time spent in the SCHEDULE section, which is the total CPU time minus the time spent initializing the problem, can be output using TCPUSCH in the SUMMARY section
- RPTONLY can now be entered in the SCHEDULE section.
- RPTONLYO is a new keyword that turns off the RPTONLY restricted summary output.

SCHEDULE section

- Field contract gas rates can now be specified for the second pass of the Gas Field Operations Model using the new keyword GASFTARG.
- A monthly decrement to the field contract gas rates can now be specified for the second pass of the Gas Field Operations Model using the new keyword GASFDECR.

- WWPAVE is a new keyword which extends the functionality of the WPAVE keyword by enabling the well block pressure average controls to be specified for individual wells.
- WTEMPQ is a new keyword that queries the extended well name and well list template matching algorithm. See "Well name and well list template matching".
- Well connection quantities and well LGR connection quantities can now be used with the ACTIONX keyword.
- The initial coal gas saturated content (for the coal bed methane option) can be output to the PRT file using the GASSATC mnemonic in the RPTSCHED keyword.
- SCDETAB defines the Karst Aquifer scale dissolution table used with Karst Aquifer evolution modeling. This table is assigned to wells by using WSCTAB and its dimension defined by using SCDPDIMS.

SUMMARY section

- Wildcard characters can be included in well names and well lists when reporting well quantities.
- Grid block interpolation keywords BPR_X, BHD_X,BHDF_X for global grid, and LBPR_X, LBHD_X, LBHDF_X for local grid are included to report interpolated values at specified XYZ coordinates for block pressures, heads and freshwater heads. The dimensions for the number of required summary interpolation outputs are defined by new RUNSPEC keyword BPIDIMS.
- Fluid flow rates in the network model can now be recorded using the GOPRNB, GWPRNB, GGPRNB, GWIRNB and GGIRNB summary mnemonics.

Temperature option

- This option is no longer restricted to water injection wells

Performance

- ZIPPY2 has a new ‘aggressive’ option
- NEXTSTEP has a new argument to allow the value to be applied at all subsequent report steps.

Wells

- CECON has been extended to allow workover procedures that shut, or plug back, the well. In addition, a follow-on well can now be specified. This will be opened (if possible) when the connection’s well is shut due to the violation of a connection economic limit.
- WTADD can be used to add (or subtract) from well targets/limits.
- GTADD can be used to add (or subtract) from group targets/limits.
- Multiple wells can now be referenced using the extended well name and well list template matching algorithm. See "Well name and well list template matching" on page 1220.
- Item 3 of the WLIFT keyword has been extended to allow a production rate limit to be specified on the GAS phase when performing automatic retubing, and THP or lift switching.

- OPTIONS item 92 allows the default behavior of WCONINJE item 9 (the VFP table number) to be modified.
  - If OPTIONS item 92 is left at zero (the default value), then the default behavior of WCONINJE item 9 remains as it was pre-2006.1.
  - If OPTIONS item 92 is set to greater than zero, then the default value of WCONINJE item 9 is initially zero, and subsequently there is no change from the previous value. (This mimics the behavior of WCONINJH item 7.)

Tracers

- The SAMG linear solver is now compatible with LGRs.
- The implicit TVD solver is now available through keyword TRACITVD.
- Group production concentrations of tracers can now be re-injected into wells. See keywords WTRACER and WSALT.

Polymer/Brine

- The WPOLYMER keyword has been extended to allow wells to inject a concentration obtained from a group's production.

PROPERTIES section

- SOF32D keyword functionality has been extended to water and gas. The new associated keywords are: SWF32D, PCW32D and SGF32D, PCG32D. Use of this option requires the user to input 2D tables for the relative permeabilities and capillary pressures.

Coal bed methane

- GASSATC is a new keyword that allows the initial saturated gas concentration in the coal to be specified. This is used to scale the Langmuir isotherm (specified using the LANGMUIR keyword) at the initial reservoir pressure.
- The Palmer-Mansoori rock model has been added in the coal bed methane option. This provides an alternative method for calculating the multipliers to the pore volume and transmissibility as the pressure changes during the simulation. This rock model is activated using the new option 'PALM-MAN' in item 1 of the ROCKCOMP keyword. The new keyword ROCKPAMA is used to specify the Palmer-Mansoori rock model parameters.

Parallel and Flux Boundary Options

- The USEFLUX Flux Boundary options and the Parallel Option are now compatible.

New facilities in ECLIPSE 300

RUNSPEC section

- For SOLID and BLACKOIL runs, the COMPSOL keyword should be used to set the number of components.

- The PETOPTS keyword has been added in order to set options for an ECLIPSE 300 run fired from Petrel.
- The NMATRIX keyword activates the multi porosity model. See"Multi porosity".
- The CBMOPTS keyword is used to specify the type of adsorption model to be used with the coal bed methane option and if water flow is allowed in out of coal cells (second data item).

  In release 2006.2, this keyword can be used to disallow water flow in and out of coal cells.
- The CO2SEQ keyword activates the CO2 sequestration option (a special license is required).

GRID section

- The PINCHXY keyword can now be used to generate horizontal pinchout connections.
- The NMATOPTS keyword is used to activate automatic matrix discretization in connection with the multi porosity model.
- The COALNUMR keyword is used to redefine coal region numbers in connection with the multi- porosity option.
- The CRNDENS keyword can be used to set the rock density for each coal region number.

Properties and equilibration

- SALINITR is a new keyword extending the existing SALINITY keyword. It allows the user to specify salinity region by region.
- The CALVALR keyword has been added to allow separate component calorific values to be assigned to different equation of state regions.
- The SOLUBILS is a new keyword extending the existing SOLUBILI keyword. It allows you to specify the aqueous phase properties of a water-CO_2 system with salt dependencies.

SUMMARY section

- Wildcard characters can be included in well names and well lists when reporting well quantities.
- New SUMMARY keywords FCNWR, GCNWR, WCNWR have been added giving the NGL component mass rates.
- Calorific value production rate and gas qualities can now be reported at connection and segment levels using the new SUMMARY keywords CCVPR, CGQ, SCVPR and SGQ.
- The new keywords ROIPL, BOIPL, ROIPG and BOIPG enable the oil in place in the liquid and gas phases to be reported at region and block level.
- FPR, FOIP, FOIPL, FOIPG, FWIP, FGIP, FGIPL,FGIPG and WPI have been added to the list of summary mnemonics represented by ALL.

SCHEDULE section

- The WVFPDP keyword (already available in ECLIPSE 100) has been added to ECLIPSE 300. The WVFPDP keyword enables a better flow rate match to be achieved for a THP controlled well by allowing you to adjust the BHP obtained from the VFP tables.

- A new keyword, WWPAVE, has been added. This extends the functionality of the WPAVE keyword by allowing well block pressure average controls to be specified on a well-by-well basis.
- WTEMPQ is a new keyword that queries the extended well name and well list template matching algorithm. See "Well name and well list template matching".
- A group's excess gas rate, GEXGR, can now be included in the triggering conditions for ACTIONX and ACTIONG keywords.
- NEXTSTEP has a new argument to allow the value to be applied at all subsequent report steps.

Wells

- CECON has been extended to allow a workover procedure that shuts the connection's well. In addition, a follow-on well can now be specified. This will be opened (if possible) when the connection's well is shut due to the violation of a connection economic limit.
- When the gas fuel, sales and re-injection requirements of a group exceed the total available gas for that group, the only remedial action pre-2006.1 was to scale back both the fuel and the gas in the same proportion until the requirements matched the available gas. Two alternative methods have been added at 2006.1.
  - The first honours the priority order specified in WTAKEGAS and scales back the lowest priority out if the fuel and sales gas first.
  - The second does not apply any scaling of the fuel and sales gas, so the gas usage will exceed the available gas.
- The default is the pre-2006.1 behavior. The alternative methods can be selected using item 95 of OPTIONS3.
- GTADD can be used to add (or subtract) from group targets/limits.
- WTADD can be used to add (or subtract) from well targets/limits.
- Multiple wells can now be referenced using the extended well name and well list template matching algorithm. See "Well name and well list template matching".
- An option has been added (see item 96 of the OPTIONS3 keyword) to include the pressure equivalent radius in the calculation of the velocity dependent skin factor. By default, this term is not included.
- New group production rate control modes have been added to allow production balancing using either the surface gas, or surface water, volumes. These new modes are requested using the PBGS and PBWS mnemonics in item 2 of GCONPROD respectively. With a balancing fraction of 1, these new control modes allow you to model scenarios where you do not want to produce excess gas or water.

Aquifers

- The aquifer properties can now be modified at any time during the simulation by re-entering the AQUCT keyword in the Schedule section.
- Initial tracer concentrations for new analytic aquifers, AQUFETP, AQUFLUX and AQUCHWAT can now be set in the Solution section using the AQANTRC keyword.
- The property data for Fetkovich aquifers can now be specified in both the Solution and Schedule sections using the AQUFETP keyword.

- A Constant Flux aquifer can be specified using the AQUFLUX keyword in both the Solution and Schedule sections.
- The property data for constant head aquifers can now be specified using the AQUCHWAT keyword in both the Solution and Schedule sections.
- Connections to the Fetkovich (AQUFETP), Constant Flux (AQUFLUX) and Constant Head (AQUCHWAT) aquifers are read in by the AQUANCON keyword.
- It is now possible to specify the aquifer influx coefficient multiplier, item 10 of the AQUANCON keyword.
- Output of analytic data in the Solution section can be achieved by entering mnemonics AQUFETP, AQUFLUX or AQUCHWAT under the RPTSOL keyword.
- Output of analytic data in the Schedule section can be achieved by entering mnemonics AQUFETP, AQUFLUX or AQUCHWAT under the RPTSCHED keyword.
- If any of the analytic aquifers (AQUCT, AQUFETP, AQUFLUX, AQUCHWAT) have been re-specified in the Schedule section, then that keyword, together with its modified data, will be output to the .PRT file by default.
- Analytic and Numerical aquifer quantities can now be output via mnemonics in the Summary section.

Non-Darcy (Forchheimer)

- An extra “pressure equivalent radius” term has been incorporated into the calculation of the velocity dependent skin factor (see [EQ 45.19]). By default, this term is not included in the calculation. To use this term, set item 96 of the OPTIONS3 keyword to 1 or 3.

Geomechanics

- Cap model: The extended Sandler-Rubin cap model is used to simulate the plastic deformation with shear failure surface together with a cap shaped yield in compression (GEOYCAP keyword).
- Faulted grids can now be simulated using finite element formulation.
- A set of new keywords has been added for a better handling of the boundary conditions: OVERBDI, TECBDI, GMDISBC, GMPSTBC, GMTRABC, as well as their equivalent for local grid refinement, for example OVERBDIL.
- Keyword STRESSBC has been changed to STRESBC.
- The set of boundary conditions can be set in both the SOLUTION and SCHEDULE sections.
- A set of new mnemonics has been added to the RPTSOL keyword: GOSTRES PGOSTRES TCSTRES PTCSTRES THSTRES PTHSTRES.
- GEOMECH keyword has been extended with a new item to allow initial stress calculations for thermal loading.
- The directions of the principal stresses are now output together with the values of the principal stresses (PRSTRESS mnemonic in RPTRST and RPTSCHED).
- New mnemonics DSTRAIN DEFFSTRS DTOTSTRS DPRSTRES for outputting delta strains and delta stresses have been added to the RPTRST and RPTSCHED keywords.

Reservoir Optimization

- When handling constraints, the control parameters are split into free and dependent sets. Previously, this splitting was done by simply taking the first parameters in the list as dependent, which meant that the algorithm was dependent on the order in which the parameters were specified in the `OPTPARS` keyword.

  In this version, an algorithm has been added to choose the dependent parameters to be the ones for which the constraints have the greatest sensitivity. This makes the algorithm insensitive to the order in which the parameters are specified.

  A new item has been added to the `OPTOPTS` keyword to allow you to revert back to the pre-2006A behavior.

- When calculating the adjoint gradients, pre-2006.1 versions of the code [EQ 47.15] used to eliminate the reservoir constraint adjoint vector, $\underline{\Psi}$, from [EQ 47.14]. This formulation is not well conditioned when the active constraints are insensitive to the dependent parameters.

  In this version of the code, an alternative formulation has been added; this uses [EQ 47.14] to eliminate the optimization active constraint adjoint vector, $\underline{\Phi}$, from [EQ 47.15]. This alternative formulation is generally more expensive than the original formulation, but is more robust.

  The default behavior is for the original formulation to be used over all time intervals, with the alternative formulation used for those time intervals over which the active constraints are considered to be insensitive to the dependent parameters.

  A new item has been added to the `OPTOPTS` keyword to allow you to select that the original formulation, or the alternative formulation, are always used. In addition, a new item has been added to the `OPTTUNE` keyword to allow you to specify when active constraints are considered insensitive to the dependent parameters.

- The objective function, parameters and constraints have been extended to recognize the ‘liquid’ fluid phase (in addition to the ‘oil’, ‘water’ and ‘gas’ fluid phases that were recognized in previous versions). The lists of available mnemonics have been extended on the `OPTFUNC`, `OPTPARS`, and `OPTCONS` keywords.

- Injection quantities have been added to the available objective function items. The lists of available mnemonics have been extended in the `OPTFUNC` keyword.

Adjoint gradient screening

- A new “adjoint gradient screening” facility has been added for this release; it is activated using the AJGRADNT keyword. It calculates sensitivities of a variety of dynamic flow-related parameters for particular wells (for example, component rate, flowing bottom-hole pressure). The required dynamic flow-related parameters are specified using the AJGWELLS keyword. The sensitivities are calculated with respect to a large number of static grid parameters relevant to the flow equations, such as porosity or permeability of every active grid block. The static grid parameters are specified using the AJGPARAM keyword.
  - The sensitivity calculation is performed efficiently for such a large number of static grid parameters by the use of the adjoint method. This is described in "Adjoint Gradient Screening".
  - The sensitivities are stored as real arrays in an additional section at the end of each restart file. They can be plotted by loading restart data into FloViz or other visualization packages.

Multi porosity

- The dual porosity option is extended with a multi porosity option in order to study the transient behavior in the matrix. The number of matrix cells for each simulation cell is specified with the NMATRIX keyword.
- Pore volumes and transmissibilities are automatically assigned by using the keyword NMATOPTS. See "Multi porosity".

Coal bed methane and Shale gas

- It is now possible to use an instant desorption model; the model can be used with single porosity, dual porosity and the new multi porosity feature. The instant desorption model is activated by using the CBMOPTS keyword.
- It is possible using the CBMOPTS keyword to specify whether water flow in and out of a coal cell is permitted.
- The time-dependent desorption model can be used with dual porosity and the multi porosity feature.
- Adsorption properties for the matrix subcells can be specified by using COALNUMR and CRNDENS.
- For further information see "Multi porosity", "Shale Gas" and "The Coal Bed Methane model".
- OPTIONS3 switch 98 can be used to suppress warnings when the ROCKTAB keyword is used with non-monotonic pore volume or transmissibility multipliers.

Output and restarts

- If additional sets of fluid-in-place regions are defined by FIPxxx keywords in the REGIONS section, then specifying the INIT keyword will output this information to the .INIT file.
- For Dual Porosity runs, output to the .PRT file will now explicitly define the matrix and fracture cells, for both global and local grids. This applies to solution and property arrays requested using RPTSOL and RPTSCHED keywords.

- Output of analytic data in the Solution section can be achieved by entering mnemonics AQUFETP, AQUFLUX or AQUCHWAT under the RPTSOL keyword.
- Output of analytic data in the Schedule section can be achieved by entering mnemonics AQUFETP, AQUFLUX or AQUCHWAT under the RPTSCHED keyword.
- If any of the analytic aquifers (AQUCT, AQUFETP, AQUFLUX, AQUCHWAT) have been re-specified in the Schedule section, then that keyword, together with its modified data, will be output to the .PRT file by default.
- Analytic and Numerical aquifer quantities can now be output via mnemonics in the Summary section.
- The current Endpoint Scaling output using either EPSDBGS or EPSDEBUG has been extended to take into account Hysteresis.
- RPTONLY can now be entered in the SCHEDULE section.
- RPTONLYO is a new keyword that turns off the RPTONLY restricted summary output.
- Using NNC or ALLNNC mnemonics under the RPTGRID keyword for corner point geometry grids containing faults, would output the NNCs including any fault NNCs to the .PRT file. In previous releases, any fault NNC would be output with type “fault”. For this release, fault NNCs have been separated into those in the X-direction (with type “X-Fault”), and those in the Y-direction (with type “Y-Fault”).
- Using the multi porosity option, NMATRIX, only the primary unknowns like molar densities and pressures are output to the restart file for the matrix subcells.

Behavioral changes

These may cause changes to the results of a simulation, or alter the form or content of the output.

Behavioral changes in ECLIPSE 100

Linear Solver

- An improved numerical formulation has been added in the accumulation of the solution and residual in the linear solver. This formulation is more stable for cells with small pore volumes. A new switch (item 91) has been added to the OPTIONS keyword to restore the pre-2006.1 behavior.
- The RS (dissolved gas ratio) extrapolation in the water zone has been improved. The new extrapolation avoids having physically impossible values for RS in the water zone. A new switch (item 94) has been added to the OPTIONS keyword to restore the pre-2006.1 behavior.
- The Parallel solver creates additional matrix terms. If the model contains cells which are completely isolated from the rest of the reservoir then the additional terms can cause the solution in these isolated cells to change. A new switch (item 95) has been added to the OPTIONS keyword which prevents the solution changing in these isolated cells.

Behavioral changes in ECLIPSE 300

RUNSPEC section

- The COMPSOL keyword is only required in SOLID and BLACKOIL runs. For the pre-2006.1 release, the COMPS keyword was used to set the number of components for this type of run. However, this caused a few problems, especially with the pre-processor software, as an ECLIPSE 300 deck requires either the BLACKOIL keyword or the COMPS keyword, but not both.

Wells

- The timestep following a change in the control mode for a multi-segment well is no longer limited to 5 days. Pre-2006.1 behavior can be restored via item 97 of the OPTIONS3 keyword

SUMMARY section

- FPR, FOIP, FOIPL, FOIPG, FWIP, FGIP, FGIPL,FGIPG and WPI have been added to the list of summary mnemonics represented by ALL.

SCHEDULE section

- The NEXTSTEP keyword can now be used to control the length of the initial time step of the simulation. Previously, the first step could only be limited with the TUNING keyword.

Non-Darcy (Forchheimer)

- There has been a change to the unit-dependent constants in the equation for the calculation of the velocity dependent skin factor ([EQ 45.18]). Both the pre-2006.1 values and new values are specified in Table 45.2. To revert to the pre-2006.1 values, set item 96 of the OPTIONS3 keyword to 2 or 3.

Geomechanics

- Keyword STRESSBC has been changed to STRESBC.
- Calculations of the directions of the principal stresses have been enhanced.
- New mnemonics DSTRAIN DEFFSTRS DTOTSTRS DPRSTRES for outputting delta strains and delta stresses have been added to the RPTRST and RPTSCHED keywords.
- Improved calculation of the directions of the principal stresses (this may give different results when compared to 2006.1).

Thermal

- OPTIONS3(4) is now automatically turned on for thermal runs. It can be turned off, in thermal runs, by setting it to -1.
- OPTIONS3(92), an alternative time-stepping strategy, is now automatically turned on for thermal runs. It can be turned off, in thermal runs, by setting it to 1.
- In CVCRIT, the default values of items 2, 7 and 9 have been refined for thermal runs.
- In TSCRIT, the default value of item 4 has been refined for thermal runs.

Aquifers

- The equilibration tilt correction is now applied to numerical aquifer blocks when item 9 of the `EQUIL` keyword is positive. Previous behavior can be restored by changing the setting of `OPTIONS3`(105).

Equilibration

- The equilibration block pressure is now taken as the central pressure as opposed to an average over the sampling points specified by Item 9 of the `EQUIL` keyword. Previous behavior can be restored by changing the setting of `OPTIONS3`(105).

New keywords

New keywords in ECLIPSE 100

RUNSPEC section

`BPIDIMS`	This keyword specifies dimensions required for summary interpolated block output keywords BPR_X, BHD_X and BHDF_X.
`PETOPTS`	This keyword specifies 4 options for a simulation run fired from Petrel. In this release, it is possible to have:- INITNNC: the non-neighbor connection index data to be placed in the .INIT file, rather than the `.GRID` file (by default, Petrel defined datasets containing the `PETGRID` keyword will not request grid file output); NOFLTNNC: suppress simulator defined fault transmissibility calculations; TINYINIT: only output pore volumes, transmissibilities and non-neighbor connection information to the .INIT file. EDITSUPP: suppress warning messages that could be output due to NNC not being found while reading the `EDITNNC` or `EDITNNCR` data.

PROPS section

`ROCKPAMA`	Specifies the Palmer-Mansoori rock model parameters in the coal bed methane option.
`TRACITVD`	Implements implicit TVD for tracer tracking.
`SWF32D`	Specifies a 2D table for the water relative permeability as a function of oil saturation and gas saturation
`PCW32D`	Specifies a 2D table for the water/oil capillary pressure as a function of oil saturation and gas saturation
`SGF32D`	Specifies a 2D table for the gas relative permeability as a function of oil saturation and water saturation
`PCG32D`	Specifies a 2D table for the gas/oil capillary pressure as a function of oil saturation and water saturation

SOLUTION section

GASSATC	Allows the initial saturated gas concentration in the coal to be specified when using the coal bed methane option. The information in this keyword is used to scale the Langmuir isotherm (specified using the LANGMUIR keyword) at the initial reservoir pressure.
RPTSOL	Initial coal gas saturated content can now be output to the PRT file using GASSATC (for the coal bed methane option)

SUMMARY section

(F,G,W,C)C(P,I)C	Polymer injection and production concentrations
BPR_X	Report interpolated pressure at specified coordinate.
BHD_X	Report interpolated hydraulic head at specified coordinate.
BHDF_X	Report interpolated fresh water hydraulic head at specified coordinate.
GOPRNB	Oil flow rate along the group's or node's outlet branch in the production network
GWPRNB	Water flow rate along the group's or node's outlet branch in the production network
GGPRNB	Gas flow rate along the group's or node's outlet branch in the production network
GWIRNB	Water flow rate along the group's or node's inlet branch in the water injection network
GGIRNB	Gas flow rate along the group's or node's inlet branch in the gas injection network
WGPRFP	Well Gas Production Rate from end of First Pass
WTHPFP	Well Tubing Head Pressure from end of First Pass
WBHPFP	Well Bottom Hole Pressure from end of First Pass
GPRFP	Group or node PRessure in network from end of First Pass
GGPRNBFP	Gas flow rate along Group's or node's outlet branch in network, from end of First Pass
TCPUSCH	The CPU time spent in the SCHEDULE section (the total CPU minus the time spent initializing the problem).
RPTONLYO	Requests summary data is written every time-step. This is the default situation and the keyword need only be used to turn off a previously entered RPTONLY instruction.

SCHEDULE section

GASFTARG	Specify gas rates for second pass of gas field operations model
GASFDECR	Specify a decrement for second pass of gas field operations model
GTADD	Adds a factor to group targets/limits.
RPTONLY	Requests that the summary data is only produced at report times. The default if RPTONLY is not present, is to write the summary information at every time.
RPTONLYO	Requests summary data is written every time-step. This is the default situation and the keyword need only be used to turn off a previously entered RPTONLY instruction.

RPTRST	Oil velocities can be output using VELOIL
RPTRST	Water velocities can be output using VELWAT
RPTRST	Gas velocities can be output using VELGAS
RPTRST	Pore volumes can now be output using RPORV
RPTRST	The shear viscosity (Polymer flood option) can now be output with the VISC mnemonic.
RPTSCHED	Initial coal gas saturated content can now be output to the PRT file using GASSATC (for the coal bed methane option).
SCDETAB	Table for specifying karst aquifer dissolution parameters. This table is assigned to wells by using WSCTAB and its dimension defined by using SCDPDIMS.
WWPAVE	Individual well block average pressure controls
WTADD	Adds a factor to well targets/limits.
WTEMPQ	Queries the well name and well list template matching algorithm

New Keywords in ECLIPSE 300

RUNSPEC section

AJGRADNT	Activates the adjoint gradient screening option
CBMOPTS	Specify the type of adsorption model to be used with the coal bed methane option
CO2SEQ	Activates the CO_2 sequestration option.
COMPSOL	For SOLID, BLACKOIL runs, this keyword sets the number of components.
NMATRIX	Activates multi-porosity/matrix discretization.
PETOPTS	This keyword specifies 3 options for a simulation run fired from Petrel. In this release, it is possible to have:- INITNNC: the non-neighbor connection index data to be placed in the .INIT file, rather than the .GRID file (by default, Petrel-defined datasets containing PETGRID keyword will not request grid file output); NOFLTNNC: suppress simulator defined fault transmissibility calculations; TINYINIT: only output pore volumes, transmissibilities and non-neighbor connection information to the .INIT file.

GRID section

COALNUMR	Redefine coal region numbers in connection with the multi-porosity option.
CRNDENS	Specifies rock density for each coal region number.
NMATOPTS	Controls the discretised matrix Dual Porosity option.
PINCHXY	This keyword can now be used to generate horizontal pinchout connections.

PROPS section

CALVALR	Component calorific values for multiple equation of state regions
GEOYCAP	Defines parameters for a shear failure surface, a yield cap in compression and hardening laws in each geomechanics region.
SALINITR	Extends the existing SALINITY keyword to allows the user to specify salinity region by region
SOLUBILS	Specify properties of water-CO_2 system with salt dependencies

SUMMARY section

(F,A)AQR	Aquifer influx rate (for analytic aquifers)
(F,A)AQT	Cumulative aquifer influx (for analytic aquifers)
AAQP	Aquifer pressure (Fetkovich aquifers only)
(F,A)NQR	Aquifer influx rate (for numerical aquifers)
(F,A)NQT	Cumulative aquifer influx (for numerical aquifers)
ANQP	Aquifer pressure (average pressure weighted by water volume). For numerical aquifers.
ALL	Extension of ALL mnemonic to include FAQR, AAQR, FAQT and AAQT.
(F,G,W)CNWR	Component NGL mass rates for the field, groups and wells
RPTONLYO	Requests summary data is written every time-step. This is the default situation and the keyword need only be used to turn off a previously entered RPTONLY instruction.
(S,C)CVPR	Calorific value production rate
(S,C)GQ	Gas quality
(R,B)OIPG	Oil-in-place in gas phase
(R,B)OIPL	Oil-in-place in liquid phase

SOLUTION section

AQUFETP	The property data for Fetkovich aquifers can now be specified using the AQUFETP keyword.
AQUFLUX	A Constant Flux aquifer can be specified using the AQUFLUX keyword.
AQUCHWAT	The property data for constant head aquifers can now be specified using the AQUCHWAT keyword.
GMDISBC	Specify displacement (Dirichlet) boundary condition.
GMDISBCL	Specify displacement (Dirichlet) boundary condition in an LGR.
GMTRABC	Specify traction (Neumann) boundary condition.
GMTRABCL	Specify traction (Neumann) boundary condition in an LGR.
GMPSTBC	Specify principal Stress boundary conditions.
GMPSTBCL	Specify principal Stress boundary conditions in an LGR.
OVERBDI	Specify overburden stresses for initial gravity induced stress field.
OVERBDIL	Specify overburden stresses for initial gravity induced stress field in an LGR.

RPTSOL	Output of analytic data in the Solution section can be achieved by entering mnemonics AQUFETP, AQUFLUX or AQUCHWAT under the RPTSOL keyword.
STRESBC	Specify a load or a velocity (rigid) boundary condition on all external and internal surfaces of the simulation grid. (Replaces previous STRESSBC keyword.)
STRESBCL	Specify displacement and traction boundary conditions in a local grid refinement. (Replaces previous STRESSBCL keyword.)
TECBDI	Specify tectonic boundary condition for initial tectonic-induced stresses.
TECBDIL	Specify tectonic boundary condition for initial tectonic induced stresses in an LGR.

SCHEDULE section

ACTIONG	Define a triggering condition based on a group's excess gas rate using the GEXGR mnemonic.
ACTIONX	Define a triggering condition based on a group's excess gas rate using the GEXGR mnemonic.
AJGWELLS	Specify objective functions for adjoint gradient screening calculations.
AJGPARAM	Specify static parameters for adjoint gradient screening calculations.
AQUCT	The aquifer properties can now be modified at any time during the simulation by re-entering this keyword in the Schedule section.
AQUFETP	The property data for Fetkovich aquifers can be re-specified using the AQUFETP keyword in the Schedule section.
AQUFLUX	A Constant Flux aquifer can be modified using the AQUFLUX keyword in the Schedule section.
AQUCHWAT	The property data for constant head aquifers can now be re-specified using the AQUCHWAT keyword in the Schedule section.
GMDISBC	Specify displacement (Dirichlet) boundary condition.
GMDISBCL	Specify displacement (Dirichlet) boundary condition in an LGR.
GMTRABC	Specify traction (Neumann) boundary condition.
GMTRABCL	Specify traction (Neumann) boundary condition in an LGR.
GMPSTBC	Specify principal Stress boundary conditions.
GMPSTBCL	Specify principal Stress boundary conditions in an LGR.
GTADD	Adds a factor to group targets/limits.
OVERBDI	Specify overburden stresses for initial gravity-induced stress field.
OVERBDIL	Specify overburden stresses for initial gravity- induced stress field in an LGR
RPTONLY	Requests that the summary data is only produced at report times. The default, if RPTONLY is not present, is to write the summary information at every time.
RPTONLYO	Requests summary data is written every time-step. This is the default situation and the keyword need only be used to turn off a previously entered RPTONLY instruction.
RPTRST	Directions of principal stresses output with mnemonic PRSTRESS
RPTSCHED	Output of analytic data in the Schedule section can be achieved by entering mnemonics AQUFETP, AQUFLUX or AQUCHWAT under the RPTSCHED keyword.
RPTSCHED	Directions of principal stresses output with mnemonic PRSTRESS

STRESBC	Specify a load or a velocity (rigid) boundary condition on all external and internal surfaces of the simulation grid. (Replaces previous STRESSBC keyword.)
STRESBCL	Specify displacement and traction boundary conditions in a local grid refinement. (Replaces previous STRESSBCL keyword.)
TECBDI	Specify tectonic boundary condition for initial tectonic-induced stresses.
TECBDIL	Specify tectonic boundary condition for initial tectonic-induced stresses in an LGR.
WVFPDP	Adjust a well's BHP obtained from the VFP tables.
WWPAVE	Individual well block average pressure controls
WTADD	Adds a factor to well targets/limits.
WTEMPQ	Queries the well name and well list template matching algorithm.

Altered keywords

Altered keywords in ECLIPSE 100

RUNSPEC section

ROCKCOMP	A new option, PALM-MAN, has been added to item 1 of this keyword. This activates the Palmer-Mansoori rock model in the coal bed methane runs.
SCDPDIMS	A new item 7 is added to define the number of Karst Aquifer scale dissolution tables defined in SCDETAB.

PROPS section

HYDRHEAD	The depth terms used in the calculation of the block heads can now be removed from the calculation by using item 3 of this keyword.

SOLUTION section

AQUCHWAT	New item 10 allows the user to turn off capillary pressure effects.

SCHEDULE section

CECON	Two new workover procedure options, WELL and PLUG, have been added to item 9 of this keyword. WELL specifies that the well is shut when a connection economic limit is violated. A new item (item 13) has also been added to this keyword to allow a follow-on well to be specified. PLUG specifies that the well is plugged back when a connection economic limit is violated.
NEXTSTEP	An extra item has been added allowing the given time-step to be applied at every subsequent report step.

WCONINJE	The default behavior of WCONINJE item 9 (VFP table number) can now be modified using OPTIONS item 92. If it is set to zero, which is its default value, then the default behavior of WCONINJE item 9 remains as it was pre-2006a. If OPTIONS item 92 is set to greater than zero, then the default is initially zero, and subsequently no change from the previous value.
WLIFT	A new limiting production phase option, GAS, has been added to item 3 of this keyword.
WPOLYMER	Two new items allow a well to inject a polymer and/or brine concentration obtained from a named group.
WSALT	New item allows for the injected concentration to be obtained from a named group.
WSCTAB	A new item 5 is added to associate this well with a Karst Aquifer scale dissolution table define by SCDETAB.
WTEMP	The restriction on only being able to use this keyword for water injection wells has been removed.
WTRACER	New item allows for the injected concentration to be obtained from a named group.
ZIPPY2	An 'aggressive' option increases time-step sizes faster than using the SIM option by itself

OPTIONS keyword

Switch 91	Restores the pre-2006.1 behavior of the accumulation of the solution and residual in the linear solver. (A new numerical formulation has been added with 2006.1 which is more stable for cells with small pore volumes.)
Switch 92	If set greater than zero, the default behavior of the VFP table specification in the WCONINJE keyword is changed so that a zero or default value indicates that the previously specified VFP table for this well should continue to be used. (The standard default here is that if a zero or default value is specified, then no THP calculations are performed.)
Switch 93	If set greater than zero, the pre-2006.1 algorithm for matching well names and well lists is restored.
Switch 94	If set greater than zero, the pre-2006.1 RS extrapolation in the water zone is restored.
Switch 95	If set greater than zero, the additional matrix fill-in terms in the parallel solver are modified.

DEBUG keyword

79	Outputs norms of the non-linear residuals.

Altered keywords in ECLIPSE 300

RUNSPEC section

- For the 2005A release, the COMPS keyword was used to set the number of components for a SOLID, BLACKOIL run. For 2006.1 and future releases, the COMPSOL keyword should be used instead.

PROPS section

EPSDBGS	Output to the DEBUG file has now been extended for runs containing both Endpoint Scaling and Hysteresis options.
EPSDEBUG	Output to the DEBUG file has now been extended for runs containing both Endpoint Scaling and Hysteresis options.

SOLUTION section

AQANTRC	Initial tracer concentrations for new analytic aquifers, AQUFETP, AQUFLUX and AQUCHWAT, can now be set in the Solution section.
AQUANCON	Connections to the Fetkovich (AQUFETP), Constant Flux (AQUFLUX) and Constant Head (AQUCHWAT) aquifers are read in using this keyword. In addition, it is now possible to specify the aquifer influx coefficient multiplier, item 10 of the AQUANCON keyword.
AQUCT	The aquifer properties can now be modified at any time during the simulation by re-entering the AQUCT keyword in the Schedule section.

SCHEDULE section

CECON	A new workover procedure option WELL has been added to item 9 of this keyword. It allows a well to be shut when a connection economic limit is violated. A new item (item 13) has also been added to this keyword to allow a follow-on well to be specified when a well is shut.
NEXTSTEP	An extra item has been added allowing the given time-step to be applied at every subsequent report step.

OPTIONS3 keyword

	Switch 94	If set greater than zero, the pre-2006.1 algorithm for matching well names and well lists is restored.
	Switch 95	Changes the way that the fuel and sales gas are scaled back when the total gas usage exceeds the amount available.
	Switch 96	This switch controls options related to the calculation of the velocity-dependent skin factor ([EQ 45.18] or [EQ 45.19]).
	Switch 97	This switch restores the pre-2006.1 5-day limit on the timestep following a change of control mode for a multi-segment well.
Coal bed methane option	Switch 98	If set greater than zero, warnings are suppressed for non-monotonic pore volume or transmissibility multiplier values entered with the ROCKTAB keyword.
	Switch 99	If set to non-zero, return to the pre-2006.1 use of velocity dependent relative permeability calculations in the generalized pseudo-pressure calculations.
	Switch 101	If OPTIONS3 item 101 is set to 1, then the test for zero thickness blocks in the X- and Y-direction is removed. Doing this can improve the pore volume calculation for nasty grids, for example, XY in map feet; IJ orientation with I up the page, J across.

Switch 109	Thermal: If >0 set pore volume multipliers to unity in wells for pressure dependent multipliers
Switch 110	Negative capillary pressure handling in gas water systems. If >0 switch back to pre-2006.2 behavior.

OPTIMIZE section

`OPTCONS`	The following constrained quantity mnemonics have been added: `FLPR`, `GLPR`, `WLPR`.
`OPTFUNC`	The following objective function quantity mnemonics have been added: `FLPT`, `GLPT`, `WLPT`, `FWIT`, `GWIT`, `WWIT`, `FGIT`, `GGIT`, `WGIT`.
`OPTOPTS`	An item 2 has been added to allow the pre-2006.1 method for the selection of the dependent parameters (that is, the first ones in the `OPTPARS` list of parameters) to be restored.
`OPTOPTS`	An item 3 has been added for the formulation to be used in the calculation of the adjoint gradients to be specified.
`OPTPARS`	The following control parameter mnemonics have been added: FLPR, GLPR, WLPR.
`OPTTUNE`	An item 11 has been added to allow you to specify the tolerance below which the active constraints are considered to be insensitive to the dependent parameters. This is used to determine if the alternative formulation, or the original formulation, should be used in the calculation of the adjoint gradients. This item is only used if item 3 of the `OPTOPTS` keyword is 1 (the default).

Developments in 2005A and 2005A_1

Developments in ECLIPSE 100

Aquifers

AQUCHWAT	Allows a constant head aquifer to be defined
AQUCWFAC	Changes the properties of all AQUCHWAT type aquifers
RAINFALL	Specifies a monthly flux rate for a constant flux aquifer
AQUFLUX	Salinity can now be input directly for a constant flux aquifer
AQUALIST	Aquifer lists

Output and restarts

- Well names can be defaulted in segment summary section quantities.
- Hydraulic heads can be output with HYDH and HYDHFW mnemonics in the RPTRST keyword.
- HYDRHEAD allows the input of reference data needed for the calculation of hydraulic heads.
- New SUMMARY section keywords PSS(PR,SO,SW,SG,SC) give a measure of the closeness of the solution to a pseudo steady-state. See PSTEADY.
- All messages, warnings, errors etc. output to the print file are now also output to a separate file, extension MSG, in XML format. The feature is intended primarily for simulation post-processors, but can be controlled with the MSGFILE keyword.
- New SUMMARY keywords WHD and WHDF giving the hydraulic head based on the well's bottom hole reference depth or its hydraulic reference depth if set with keyword WHEDREFD. Only available with a Water Services license.
- New SUMMARY keywords BHD and BHDF have been added, giving the hydraulic head at a grid-block. Only available with a Water Services license.
- Rates and totals can be output for named aquifer lists using ALQR and ALQT in the SUMMARY section.

Performance

- ZIPPY2 can now set the next time-step and the maximum time-step.It can also tighten convergence tolerances
- PSTEADY is a new feature to allow the code to run until a pseudo steady state is reached

Wells

- WRFTPLT RFT/PLT and SEG data can now be output every time-step.

- A number of features have been added to the pattern flood injection option (keyword WCONINJP)
  - The voidage contribution from a pattern producer to the overall injection target may now be limited to the voidage only from those production well connections that fall within a defined fluid-in-place region.
  - A multiplier may now be applied to the calculated voidage rate target for the pattern flood injector
  - If pattern flood injectors are opened or closed during the simulation, then the calculated targets for the remaining injectors may be automatically reset to ensure that the voidage from the pattern producers is still replaced.
- For wells in the scale deposition model (see keyword WSCTAB), the amount of scale deposited around the well connections may be reduced using the keyword WSCCLEAN (or WSCCLENL for wells connected in local grids).
- In the WELTARG keyword the value of a target/limit for a well control/constraint may now be set to the corresponding flowing quantity for this well at the end of the timestep immediately prior to this keyword. Thus, for example, the flowing BHP of a well at the end of the history matching phase could be transferred to a BHP limit on the well during the prediction phase.
- WHEDREFD allows the well's hydraulic head reference depth to be set

Multi-segment wells

- A third drift flux model is available, based on the most recent results of Shi *et al* (2004) [Ref. 29]. This model also takes into account the effect of gas disrupting the slip between oil and water. The model is activated by entering SHI-04 in the three items of keyword WSEGDFMD, before the segments are defined with keyword WELSEGS.
- The keyword WSEGSICD models the pressure drop across an inflow control device that is placed around a section of the tubing and diverts the fluid unfailing from the adjacent part of the formation through a sandscreen and then into a spiral constriction before it enters the tubing.

Tracers

- Diffusion and dispersion are now available on LGRs with the standard tracer solver.
- Diffusion and dispersion are now available on LGRs with the TVD solver, keyword TRACTVD.
- Mechanical dispersion can now be input for tracers with the HDISP keyword.
- Aliases for the two options in item 5 of the TRACERS keyword have been added.
- The multigrid linear solver, keyword SAMG, is now available as an alternative for single phase tracers when using the standard solver. Requires a Water Services license.

NNCs

- New items have been added to the NNC keyword to allow the user to input data needed for the calculation of velocities across user defined NNCs. This is **only** needed for the calculation of velocities or Forchheimer effects (non-Darcy).

Gradients

- Gradient parameters have been added for well connection transmissibility factor and well connection skin factor. These are specified using the new keywords HMWELCON and HMMLTWCN.

Grid

- Active cell numbers can be output using IOCN in the RPTGRID and RPTGRIDL keywords. This is intended, primarily, for developer debugging.
- Grid geometry available in Open Petrel Format (OPF) files can be imported using keyword PETGRID.

Properties

- Rock compressibility (keyword ROCK) can now be associated with rock regions specified using ROCKNUM. See keywords ROCKOPTS and TABDIMS for details.

Non-Darcy (Forchheimer effect)

- Non-Darcy, i.e. Forchheimer effects for gas flow can now be simulated using VDFLOW or VDFLOWR keywords in the PROPS section.

Developments in ECLIPSE 300

RUNSPEC

- The MEMSAVE keyword can now be used to turn on the UpFront Memory option. If this keyword, together with ACTNUM keyword in the GRID section, are used in the same dataset, it is possible for the simulator to focus on these active cells only in order to save memory. This is especially important for large models that have difficulty processing the geometric calculations, for instance.
- Item 1 of the GRIDOPTS keyword has been altered in order to allow both transmissibility and diffusivity multipliers to be specified in the negative direction.
- The EQLOPTS keyword has been extended to allow both the THPRES and IRREVERS options to be used (refer to the THPRESFT keyword in the GRID section, or the THPRES keyword in the SOLUTION section).

Coal Bed Methane option

- The COAL keyword enables the Coal Bed Methane option.

Grid

- The MULTX- keyword specifies transmissibility multipliers in the negative X-direction.
- The MULTY- keyword specifies transmissibility multipliers in the negative Y-direction.
- The MULTZ- keyword specifies transmissibility multipliers in the negative Z-direction.
- The MULTR- keyword specifies transmissibility multipliers in the negative radial-direction.
- The MULTTHT- keyword specifies transmissibility multipliers in the negative theta-direction.

DIFFUSE option • The DIFFMX- keyword specifies diffusivity multipliers in the negative X-direction.

DIFFUSE option • The DIFFMY- keyword specifies diffusivity multipliers in the negative Y-direction.

DIFFUSE option • The DIFFMZ- keyword specifies diffusivity multipliers in the negative Z-direction.

DIFFUSE option • The DIFFMR- keyword specifies diffusivity multipliers in the negative radial-direction.

DIFFUSE option • The DIFFMTH- keyword specifies diffusivity multipliers in the negative theta-direction.

- The MINNNCT keyword specifies a cut-off transmissibility below which a non-neighbor connection is deleted.
- It is now possible to specify a negative face when specifying faults using item 8 of the FAULTS keyword.
- The THPRESFT keyword can now be used to set the threshold pressures for a fault, previously defined using the FAULTS keyword. The threshold pressure switch THPRES must be set in keyword EQLOPTS in the RUNSPEC section.

Coal Bed Methane option • The COALNUM keywords is used to specify coal regions or regions without coal when the Coal Bed Methane option is active.

- Grid geometry available in Open Petrel Format (OPF) files can be imported using keyword PETGRID.

Regions

- The FIP keyword has been implemented in ECLIPSE 300. It is now possible to specify an array of region numbers for extra sets of fluid-in-place regions with keyword FIP, in addition to the standard set defined with keyword FIPNUM.

Properties and equilibration

- An alternative model has been implemented for Capillary Number effects in the gas velocity dependent relative permeability model. This model is designed for near well bore gas flows in gas condensate reservoirs. It is simpler than the existing Capillary Number effect model in that it only depends on two parameters, both of which have default values. The model is activated using the VELDEP keyword and its default parameters may be overridden using the VDKRGC keyword.

Coal Bed Methane option • The LANGMEXT keyword is used to enter gas storage capacities for the Coal Bed Methane option.

- Rock compressibility (keyword ROCK) can now be associated with rock regions specified using ROCKNUM or saturation regions specified using SATNUM. See keywords ROCKOPTS and TABDIMS for details.

Solution

- The THPRES keyword can now be used to set the threshold pressures for flow between adjacent equilibration regions. The threshold pressure switch THPRES must be set in keyword EQLOPTS in the RUNSPEC section.

Coal Bed Methane option • Scales the initial adsorbed gas in a coal bed methane run in order to model undersaturated coal using the keyword SORBFRAC.

Coal Bed Methane option • The keyword SORBPRES specifies the sorption pressure used when calculating the initial adsorbed gas in a coal bed methane run.

Schedule

- The MULTX- keyword specifies transmissibility multipliers in the negative X-direction.
- The MULTY- keyword specifies transmissibility multipliers in the negative Y-direction.
- The MULTZ- keyword specifies transmissibility multipliers in the negative Z-direction.
- The MULTR- keyword specifies transmissibility multipliers in the negative radial-direction.
- The MULTTHT- keyword specifies transmissibility multipliers in the negative theta-direction.

Summary

- New SUMMARY keywords, FCNMR, GCNMR, WCNMR, have been added giving the NGL component molar rates.

Wells

- A number of features have been added to the pattern flood injection option (keyword WCONINJP)
  - The voidage contribution from a pattern producer to the overall injection target may now be limited to the voidage only from those production well connections that fall within a defined fluid-in-place region.
  - A multiplier may now be applied to the calculated voidage rate target for the pattern flood injector.
  - If pattern flood injectors are opened or closed during the simulation, then the calculated targets for the remaining injectors may be automatically reset to ensure that the voidage from the pattern producers is still replaced.
- A scale deposition model has been implemented that accounts for the cumulative effects of scale deposited around the well connections and the resulting degradation of its productivity index. The model relies on two tables (assigned to the well using the keyword WSCTAB):
  - Scale deposition table (keyword SCDPTAB): This defines the rate of scale deposition per unit water flow rate as a function of the fraction of sea water present in the water flowing into a connection. The sea water fraction is represented by a passive water tracer configured by the user (see keyword SCDPTRAC).
  - Scale damage table (keyword SCDATAB) : This defines how the current amount of scale deposited per unit length of perforated interval alters the PI of the well connections.

  The amount of scale deposited around well connections may be reduced using the keyword WSCCLEAN (or WSCCLENL for wells connected in local grids). The connection summary mnemonics CDSM, CDSML and CDSF may be used to report the amount of scale deposited, the amount deposited per unit perforated length and the damage due to deposited scale respectively.
- In the WELTARG keyword the value of a target/limit for a well control/constraint may now be set to the corresponding flowing quantity for this well at the end of the timestep immediately prior to this keyword. Thus, for example, the flowing BHP of a well at the end of the history matching phase could be transferred to a BHP limit on the well during the prediction phase.

VFP table numbers, artificial lift quantities and guide rates may now also be set by this keyword.

- A number of features have been added to the well economic limit keyword WECON, to make it more compatible with ECLIPSE 100:
  - a maximum gas-liquid ratio
  - a minimum liquid production rate
  - a switch to control whether minimum limits apply to well rates or potentials
  - a secondary water cut limit which either applies to the last open completion in the well or to a revived well that has just been closed because the primary water cut resulted in a number of workovers that shut all the well connections.
- Wet gas production and injection rates and totals may now be used as triggering quantities in the ACTIONG, ACTIONW and ACTIONX keywords.

Multi-segment wells

- A third drift flux model is available, based on the most recent results of Shi *et al* (2004) [Ref. 29]. This model also takes into account the effect of gas disrupting the slip between oil and water. The model is activated by entering SHI-04 in the three items of keyword WSEGDFMD, before the segments are defined with keyword WELSEGS.

Gas Lift Optimization

- This ECLIPSE 100 feature is now available as a Special Extension to ECLIPSE 300. It determines how much lift gas to allocate to each well in order to meet well, group or field production targets. If production targets cannot be met, it determines how to make the best use of the existing lift gas resources by allocating lift gas preferentially to the wells that can make the best use of it. This feature is not compatible with the network option in ECLIPSE 300.

Geomechanics

- The finite difference stress calculation method is no longer available.
- The moving or fixed grid option is no longer available.
- A new initialization method is now available where you can specify the stress tensor at the interior of the grid, for example at well locations. Boundary stresses are predicted that honor these interior stresses. The boundary stresses are then used for subsequent time stepping. See keyword GEOMECH.
- You can now choose the sign of compressive stresses. Choices are compressive stresses are negative or positive. See keyword GEOMECH.
- The keyword STRESSBC is being phased out in favor of new keywords that allow traction boundary conditions TRACTBC, displacement boundary conditions DISPBC and principal stress boundary conditions PSTRBC.
- Two additional mnemonics have been added to the RPTRST and RPTSCHED keywords to allow the output and plotting of geomechanics quantities:
  - The TOTSTRES mnemonic outputs the total stress tensor.
  - The STRAIN mnemonic outputs the strain tensor.

Conductive Fractures

- ECLIPSE 300 now incorporates a feature for incorporating the main effects of conductive fractures into a single medium model.

Reservoir Optimization

- A new reservoir optimization feature is now available as a special feature in ECLIPSE 300.

Thermal

- For surface density calculations, new aliases, `DREFTS`, `PREFTS` and `TREFTS` can be used instead of keywords `DREFS`, `PREFS` and `TREFS`. These aliases simplify the input of data from PVT*i*.
- New keywords `ZFACT1` and `ZFACT1S` allow gas phase Z-factors to be expressed as a function of temperature - allowing modelling of the Joule-Thomson effect.

Coal bed methane

- Compositional Coal Bed Methane option is now available in ECLIPSE 300.

Water components

- It is now possible to run ECLIPSE 300 with more than one water component.

Output and restarts

- Produced component molar rates in the oil and gas phases for the field, groups and wells may now be output to the summary file using the mnemonics `(F,G,W)C(O,G)MR`. These rates may also be examined at the n'th stage of a separator via the mnemonics `(F,G,W)C(O,G)MRn`.
- The mean molecular weight of a wellstream may now be output to the summary file using the mnemonic `WMMW`.
- The `EXCEL` keyword can now be used to request run summary output in Excel format.
- All forms of messages (warnings, errors etc.) are now output by default to a separate file, extension MSG, in XML format. The feature is intended primarily for simulation post-processors, but can be controlled with the `MSGFILE` keyword.
- Output of threshold pressures (see keywords `EQLOPTS` and `THPRES`) to the print file can now be done using the THPRES mnemonic under the `RPTSOL` keyword.
- When the `INIT` keyword is used, together in a run containing analytic aquifers (`AQUCT`), then this aquifer information is also output to the `INIT` file under the `AQUIFERA` heading.
- When the `INIT` keyword is used, together in a run containing numerical aquifers (`AQUNUM`), then this aquifer information is also output to the `INIT` file under the `AQUIFERN` heading.
- FTPC and GTPC and now available to output the group detailed molar fractions for the `LUMPING` option.

Behavioral changes

These may cause changes to the results of a simulation, or alter the form or content of the output.

Behavioral changes in ECLIPSE 100

Wells

- A connection's voidage rate is now calculated at the reference pressure for the fluid-in-place region of the connected grid block. Prior to this release, this voidage was calculated at the reference pressure for the well (see item 13 of `WELSPECS`) - item 85 of the keyword `OPTIONS` restores this behavior.
- `In the pattern flood injection option (keyword WCONINJP)`, if production rate fractions (see item 2 of record 2 onwards) are defaulted, then these are updated at the beginning of each time step on the basis of the number of injectors assigned to replace the voidage of each producer. A producer's fractions change if one of its associated injectors is shut (or opened), with the current set of open injectors effectively automatically compensating to ensure the producer's voidage is still replaced. Prior to the 2005A release the production rate fractions were not automatically adjusted in this way.
- History matched wells (`WCONHIST`) with production rates of zero and that are banned from crossflowing are now automatically closed. This behavior can be removed using the `OPTIONS`(90) keyword.
- Injection wells with injection rates of zero and that are banned from crossflowing are now automatically closed. This behavior can be removed using the `OPTIONS`(90) keyword.

Output and restarts

- The output to the summary specification file required by post-processing applications for run time monitoring option is now defaulted to On. (see keywords `MONITOR` and `NOMONITO`).

Tracers

- The behavior of Multi-segmented wells with tracers has changed. The code has been simplified with subsequent, significant, efficiency gains. Results should not be qualitatively different, but pre-2005A behavior can be restored with `OPTIONS`(87) if required.

Solvents

- Derivative limiting is now applied to certain derivative calculations in the solvent model.This behavior can be removed using the `OPTIONS`(90) keyword.

Behavioral changes in ECLIPSE 300

Convergence Controls

- Controls on the linear solver convergence criterion, and the maximum number of linear iterations, have been changed to obey the values entered in the `TUNING` and `CVCRIT` keywords more accurately. Specifically the fourth entry of the second record and the third entry of the third record of the `TUNING` keyword, along with the third and fourth entries of the `CVCRIT` keyword are honoured. For data sets which define these linear solver parameters some changes in convergence history and run times are to be expected. Item 91 of the keyword `OPTIONS3` may be used to restore the pre-2005A behavior, but this is not recommended.

Wells

- A number of improvements have been made to the calculation of the wellbore density in injection wells when using the default segregated model. The wellbore density is used to calculate the hydrostatic pressure heads between a well's reference depth and the depths of its connections. These improvement only come into effect when the injector is cross-flowing. Item 88 of the keyword `OPTIONS3` may be used to restore this calculation to its pre-2005A state.
- In the calculation of well THP values, any ALQ that falls outside its range of values in the VFP table now results in the table being extrapolated linearly. Prior to this release, constant value extrapolation was used. (Constant value extrapolation is still used for water fraction and gas fraction values that fall outside their range of values in the table.) This change alters reported THP values and the flows in THP controlled wells if the VFP table is extrapolated in ALQ. Item 5 of the keyword `WVFPEXP` controls the type of extrapolation used for water fraction, gas fraction and ALQ values, and allows the pre-2005A behavior to be restored.
- A connection's voidage rate is now calculated at the reference pressure for the fluid-in-place region of the connected grid block. Prior to this release, this voidage was calculated at the reference pressure for the well (see item 13 of `WELSPECS`) - item 88 of the keyword `OPTIONS3` restores this behavior. Note that this change does not alter the well's voidage rate.
- For wells on voidage control, the derivatives of the well residual are calculated in a different manner. The new method is more accurate for runs using lab units, for example, slim tube simulations; and can avoid convergence problems. The pre-2005A derivatives can be restored using item 88 of the `OPTIONS3` keyword.
- In the pattern flood injection option (keyword `WCONINJP`), if production rate fractions (see item 2 of record 2 onwards) are defaulted, then these are updated at the beginning of each time step on the basis of the number of injectors assigned to replace the voidage of each producer. A producer's fractions change if one of its associated injectors is shut (or opened), with the current set of open injectors effectively automatically compensating to ensure the producer's voidage is still replaced. Prior to the 2005A release the production rate fractions were not automatically adjusted in this way.

Output and restarts

- The output to the summary specification file required by post-processing applications for run time monitoring option is now defaulted to on (see keywords `MONITOR` and `NOMONITO`).

Non-Darcy (Forchheimer)

- The implementation of the Forchheimer correction (inertia effects) has now been improved. These non-Darcy effects may have been underestimated in previous implementations.

Geomechanics

- The finite element stress calculation option is the default stress calculation method.
- By default, the stress calculation will now equilibrate before timestepping begins. This equilibration step ensures that initial pore volumes and pore pressures are retained when boundary conditions are initially applied.
- The keyword previously used for traction and displacement boundary conditions, `STRESSBC`, is being phased out in favour of new keywords that separately set these conditions. The new keywords are `DISPBC` for setting displacement boundary conditions, `TRACTBC` for setting traction boundary conditions and `PSTRBC` for setting principal stress boundary conditions.
- You can now set the sign of compressive and tensile stresses to conform to one of the two conventions. Compressive stresses can be chosen to be either negative or positive. See keyword `GEOMECH`.

New keywords

New keywords in ECLIPSE 100

RUNSPEC section

`PSTEADY`	Runs the simulator to a pseudo steady-state
`MSGFILE`	Sets output options for MSG files
`SAMG`	Uses a different linear solver for single phase tracers using the standard tracer solution method.

GRID section

`PETGRID`	Imports grid geometry from OPF files

PROPS section

`HYDRHEAD`	Reference data for hydraulic head calculation
`HDISP`	Mechanical dispersion tables for tracers

Solution section

`AQUCHWAT`	Allows a constant head aquifer to be defined
`AQUALIST`	Allows definition of aquifer lists
`RAINFALL`	Specifies a monthly flux rate for a constant flux aquifer

HMWELCON	Specifies the well connections for which gradients are required
HMMLTWCN	Specifies multipliers to well connection transmissibility factor and well connection skin

Summary section

PSSPR	Log of the pressure change per unit time, see PSTEADY
PSSSO	Log of the oil saturation change per unit time, see PSTEADY
PSSSW	Log of the water saturation change per unit time, see PSTEADY
PSSSG	Log of the gas saturation change per unit time, see PSTEADY
PSSSC	Log of the brine concentration change per unit time, see PSTEADY
BHD	Hydraulic head in grid block. Requires Water Services license.
BHDF	Hydraulic head in grid block calculated at freshwater conditions. Requires Water Services license.
WHD	Hydraulic head in well based on the reference depth given in HYDRHEAD and the well's reference depth. Requires Water Services license.
WHDF	Hydraulic head in well based on the reference depth given in HYDRHEAD and the well's reference depth calculated at freshwater conditions. Requires Water Services license.
AQLR	Aquifer influx rate for named aquifer list.
AQLT	Cumulative aquifer influx for named aquifer list.

SCHEDULE section

AQUCHWAT	Allows a constant head aquifer to be re-defined
AQUALIST	Allows the properties of all AQUCHWAT aquifers to be changed
RAINFALL	Specifies a monthly flux rate for a constant flux aquifer
RPTRST	Hydraulic head can be output with the HYDH mnemonic and the freshwater hydraulic head can be output with the HYDHFW mnemonic.(Note: reference data must be input with the HYDRHEAD keyword.)
WSCCLEAN	Reduces the amount of scale deposited around well connections
WSCCLENL	Reduces the amount of scale deposited around local grid well connections
WSEGSICD	Defines segments to represent a 'spiral' inflow control device.
WHEDREFD	Defines a well's reference depth for the calculation of hydraulic heads.

New keywords in ECLIPSE 300

RUNSPEC section

Coal Bed Methane option	COAL	Enables the Coal Bed Methane option.
Water component option	COMPW	Enables the Water Component option and specifies the number of water components
	MEMSAVE	This keyword requests the use of the UpFront Memory option.
	MSGFILE	Sets output options for MSG files.

	RESOPT	Specify that this is a Reservoir Optimization simulation.
	SCFDIMS	Set dimensions for conductive fractures.

GRID section

	MULTX-	Specifies transmissibility multipliers in the negative X-direction.
	MULTY-	Specifies transmissibility multipliers in the negative Y-direction.
	MULTZ-	Specifies transmissibility multipliers in the negative Z-direction.
	MULTR-	Specifies transmissibility multipliers in the negative radial-direction
	MULTTHT-	Specifies transmissibility multipliers in the negative theta-direction
DIFFUSE option	DIFFMX-	Specifies diffusivity multipliers in the negative X-direction
DIFFUSE option	DIFFMY-	Specifies diffusivity multipliers in the negative Y-direction
DIFFUSE option	DIFFMZ-	Specifies diffusivity multipliers in the negative Z-direction
DIFFUSE option	DIFFMR-	Specifies diffusivity multipliers in the negative radial-direction
DIFFUSE option	DIFFMTH-	Specifies diffusivity multipliers in the negative theta-direction
	MINNNCT	Specifies a cut-off transmissibility below which a non-neighbor connection is deleted.
	CONDFRAC	Defines a conductive fracture
	THPRESFT	Sets up a threshold pressure for a fault. The threshold pressure switch THPRES must be set in keyword EQLOPTS in the RUNSPEC section
Coal Bed Methane option	COALNUM	Specifies coal regions for the Coal Bed Methane option
	PETGRID	Imports grid geometry from OPF files

REGIONS section

	FIP	Specifies an array of region numbers for extra sets of Fluid-In-Place regions.

PROPS section

Water Component option	CREFW	Compressibility of water components at the reference pressure
Coal Bed Methane option	DIFFCBM	Diffusion coefficients for the Coal Bed Methane option.
Thermal option	DREFTS	Alias for surface reference density DREFS to simplify the input of data from PVTi.
Water Component option	DREFW	Density of water components at the reference pressure
Coal Bed Methane option	LANGMEXT	Specifies the coal gas storage capacities by Extended Langmuir isotherm.
Water Component option	MWW	Molecular weight of water components
Thermal option	PREFTS	Alias for surface reference pressure PREFS to simplify the input of data from PVT*i*.
Water component option	PREFW	Reference pressure for water component density and viscosity calculations
Thermal option	TREFTS	Alias for surface reference temperature TREFS to simplify the input of data from PVT*i*.
	VDKRGC	Sets data for a capillary number model of velocity dependent relative permeability for gas flow in a gas condensate reservoir.
Water Component option	VREFW	Viscosity and viscosibility for water components at the reference pressure

Water Component option	WNAMES	Water component names
Thermal option	ZFACT1	Coefficient for temperature dependent Z-factor
Thermal option	ZFACT1S	Coefficient for temperature dependent Z-factor. (Surface conditions.)
Coal Bed Methane option	RESORB	Re-adsorption factor for coal bed methane.

SUMMARY section

	(F,G,W)C(O,G)MR	Produced component molar rates in the oil and gas phases for the field, groups and wells.
	(F,G,W)C(O,G)MRn	Produced component molar rates in the oil and gas phases for the field, groups and wells at the n'th stage of a separator.
	WMMW	The mean molecular weight of a wellstream.
	EXCEL	This keyword requests run summary output in Excel format.
	CDSM	Current mass of scale deposited
	CDSML	Current mass of scale deposited per unit perforation length
	CDSF	PI multiplicative factor due to scale damage
	(F,G,W)NMR	Component NGL molar rates for the field, groups and wells

SOLUTION section

	THPRES	This keyword sets the threshold pressures for flow between adjacent equilibration regions. The threshold pressure switch THPRES must be set in keyword EQLOPTS in the RUNSPEC section.
Water Component option	WI	Specifies overall initial water composition
Water Component option	WMF	Specifies initial water composition explicitly
Water Component option	WMFVD	Specifies initial water composition with respect to depth
Coal Bed Methane option	SORBFRAC	Specifies a scaling factor for the initial adsorbed coal gas.
Coal Bed Methane option	SORBPRES	Specifies pressure values for the initial adsorbed coal gas.

SCHEDULE section

Geomechanics option	DISPBC	Sets displacement (Dirichlet) boundary conditions for the geomechanics coupled 3D stress calculation
Geomechanics option	DISPBCL	Sets displacement (Dirichlet) boundary conditions for the geomechanics coupled 3D stress calculation in a Local Grid Refinement
	GLIFTOPT	Sets group lift gas supply limits for the Gas Lift Optimization facility
	LIFTOPT	Activates the Gas Lift Optimization facility
	WLIFTOPT	Controls which wells should have their lift gas injection rates calculated by the optimization facility, and sets their maximum and minimum lift gas injection rates and weighting factors
	MULTX-	Specifies transmissibility multipliers in the negative X-direction.
	MULTY-	Specifies transmissibility multipliers in the negative Y-direction.
	MULTZ-	Specifies transmissibility multipliers in the negative Z-direction.
	MULTR-	Specifies transmissibility multipliers in the negative radial-direction

	MULTTHT-	Specifies transmissibility multipliers in the negative theta-direction
Geomechanics option	PSTRBC	Allows principal stress vectors to be used as boundary conditions for the geomechanics coupled 3D stress calculation
	SCDPTAB	Defines scale deposition tables
	SCDATAB	Defines scale damage tables
	SCDPTRAC	Set a tracer to be associated with the sea water fraction for scale deposition
Geomechanics option	TRACTBC	Sets traction (Neumann) boundary conditions for the geomechanics coupled 3D stress calculation
Geomechanics option	TRACTBCL	Sets traction (Neumann) boundary conditions for the geomechanics coupled 3D stress calculation in a Local Grid Refinement
	WSCCLEAN	Reduces the amount of scale deposited around well connections
	WSCCLENL	Reduces the amount of scale deposited around local grid well connections
	WELLCF	Gives details of well behavior with conductive fractures
Water Component option	WELLSTRW	Sets composition of the injection water stream
Water Component option	WINJW	Specifies the nature of the injection water

OPTIMIZE section

OPTCONS	Specify the constraints for the reservoir optimization problem.
OPTDIMS	Specify the dimensions of the reservoir optimization problem.
OPTFUNC	Specify the objective function for reservoir optimization.
OPTLOAD	Load in a reservoir optimization case from a given iteration.
OPTOPTS	Specify options controlling the reservoir optimization.
OPTPARS	Specify parameters for the reservoir optimization problem.
OPTTUNE	Specify convergence criteria for reservoir optimization.
RPTOPT	Control on output from the reservoir optimization facility.

Altered keywords

Altered keywords in ECLIPSE 100

RUNSPEC section

AQUDIMS	Items 7&8 determine the number and size of the aquifer lists.
TABDIMS	New item 13 is used to specify the number of tables and regions when ROCK is used together with ROCKNUM.
TRACERS	In item 5 ‘STANDARD’ is now allowed as an alias for ‘NODIFF’ and ‘SPECIAL’ is allowed as an alias for ‘DIFF’.
HMDIMS	New item 9 is used to specify the maximum number of well connection parameters for which gradients are required.

GRID section

NNC	New item allows the input of 1/(area*porosity) for use in calculating the velocity. Only needed if the velocities need to be calculated.)
RPTGRID	New item IOCN allows the output of active cell numbers. (Primarily for developer debugging.)
RPTGRIDL	New item IOCN allows the output of active cell numbers. (Primarily for developer debugging.)

PROPS section

ROCKOPTS	String 'ROCKNUM' is now allowed for item 3 in the presence of ROCK keyword.

SOLUTION section

AQUFLUX	New item allows the direct input of salinity

SCHEDULE section

AQUFLUX	New item allows the direct input of salinity
ZIPPY2	Three new mnemonics for setting the maximum time-step (MAXSTEP), the next time-step (NEXTSTEP) and tightening convergence tolerances (TIGHTEN)
WRFTPLT	A new 'TIMESTEP' option allows the output of data at every time-step
WCONINJP	New item 8 of record 1 specifies a multiplier for the calculated voidage rate target for the pattern flood injector.
	New item 3 of records 2 onwards specifies a fluid-in-place region that restricts the voidage contribution from a producer to the injection target.
WELTARG	Item 3 of this keyword may now be defaulted. This instructs ECLIPSE to set the value of a target/limit for a well control/constraint to the corresponding flowing quantity for this well at the timestep prior to this keyword.
WSEGDFMD	New option SHI-04 in items 1 and 2.
	New item 3 activates the disruption of oil-water slip by free gas.

OPTIONS keyword

Switch 85	Restores the pre-2005A calculation of connection voidage rates at the reference pressure for the well. The default behavior in 2005A is that a connection's voidage rate is calculated at the reference pressure for the fluid-in-place region of the connected grid block.
Switch 87	Restores the pre-2005A calculation of tracers with multi-segmented wells.
Switch 88	Oil and water are not considered as an emulsion in the mixture viscosity used in the pressure drop calculation for a spiral inflow control device (see keyword WSEGSICD).

Coal Bed Methane option	Switch 89	Relax the monotonic restriction of the ROCKTAB keyword, allowing non-monotonic pore volume multipliers for coal bed methane runs.
	Switch 90	Removes tests for closing history matched wells and injection wells with zero rates that are also banned from crossflowing. Also removes derivative limiting in the solvent option.

DEBUG keyword

	Switch 77	Debug from steady state calculation.

Altered keywords in ECLIPSE 300

RUNSPEC section

Geomechanics option	GEOMECH	The fully or IMPES coupled option is no longer available. The finite difference stress calculation method is no longer available. New options are available for initialization and for choosing the sign of compressive stresses.
	GRIDOPTS	Item 1 of this keyword has been altered in order to allow both transmissibility and diffusivity multipliers to be specified in the negative direction.
	EQLOPTS	This keyword has been extended to allow both the THPRES and IRREVERS options to be used (refer to the THPRESFT keyword in the GRID section, or the THPRES keyword in the SOLUTION section).
	SCDPDIMS	Items 1 to 4 now dimension scale deposition and damage tables. Item 5 and 6 dimension the proppant flowback scale damage table (ECLIPSE 300 only).
Coal Bed Methane option	REGDIMS	Item 6 specifies the maximum number of coal regions as set by COALNUM.
	TABDIMS	Item 13 is now also used to specify the number of tables and regions when ROCK is used together with ROCKNUM.
	VELDEP	New item 5 controls the activation of the alternative capillary number model of velocity dependent relative permeability for gas flow in a gas condensate reservoir.

GRID section

	FAULTS	It is now possible to specify a negative face when specifying faults using item 8 of this keyword.
	INIT	If the run contains any number of analytic aquifers (specified using the AQUCT keyword), then this information is output to the INIT file under the AQUIFERA heading. If the run contains any number of numerical aquifers (specified using the AQUNUM keyword), then this information is output to the INIT file under the AQUIFERN heading.

PROPS section

	ROCKOPTS	Strings ‘SATNUM’ and ‘ROCKNUM’ are now allowed for item 3 in the presence of ROCK keyword.
	RPTPROPS	New items LANGMEXT and DIFFCBM allows the output of Langmuir isotherms data and the coal bed methane diffusion coefficients.

SOLUTION section

	RPTSOL	Output of threshold pressures (see keywords EQLOPTS and THPRES) to the print file can now be done using the THPRES mnemonic.

SCHEDULE section

	ACTIONG	Wet gas production and injection rates and totals may now be used as triggering quantities.
	ACTIONW	Wet gas production and injection rates and totals may now be used as triggering quantities.
	ACTIONX	Wet gas production and injection rates and totals may now be used as triggering quantities.
Geomechanics option	RPTRST	New mnemonics COHESION, GENPLSTN, GENPSTRS, PLASDISP, STRAIN and TOTSTRES added for the Geomechanics option.
Geomechanics option	RPTSCHED	New mnemonics STRAIN and TOTSTRES added for the Geomechanics option.
	WCONINJP	New item 8 of record 1 specifies a multiplier for the calculated voidage rate target for the pattern flood injector.
		New item 3 of records 2 onwards specifies a fluid-in-place region that restricts the voidage contribution from a producer to the injection target.
	WVFPEXP	New item 5 that controls the type of extrapolation used for water fraction, gas fraction and ALQ values that fall outside the range of VFP table values.
	WELTARG	Item 3 of this keyword may now be defaulted. This instructs ECLIPSE to set the value of a target/limit for a well control/constraint to the corresponding flowing quantity for this well at the timestep prior to this keyword. VFP table numbers, artificial lift quantities and guide rates may now also be set by this keyword.
	WECON	Items 10 to 14 are now enabled for ECLIPSE 300.
	WSCTAB	Now assigns well scale deposition and damage tables. A new item 4 has been added in ECLIPSE 300, to define the scale damage table for proppant flowback.
	WSEGDFMD	New option SHI-04 in items 1 and 2.
		New item 3 activates the disruption of oil-water slip by free gas.

OPTIONS3 keyword

	Switch 88	Overrides some corrections made to the well calculation for the 2005A release, restoring compatibility with earlier releases to allow you to assess the importance of the changes.
Coal Bed Methane option	Switch 93	For adsorption it is possible to alter the flow so it is not scaled by the gas saturation (gas in contact with the coal).

Developments in 2004A

New facilities for 2004A

ECLIPSE 100

RUNSPEC

NRSOUT	Sets the maximum number of quantities that can be output to a restart file at each report step.
ACTPARAM	Allows some control over how much an ACTION condition may be violated. The keyword also allows the default fractional equality tolerance to be changed. This tolerance is used in deciding whether quantities are equal in an ACTIONX condition.

GRID

- It is now possible to modify the power coefficients and to select the permeability direction in the J-function expression. See the JFUNC keyword.
- It is now possible to delete non-neighbor connections that have transmissibility below a threshold. See the MINNNCT keyword.

Properties and equilibration

- EPSDBGS Allows scaled relative permeability curves to be written for multiple sets of grid blocks when the End Point Scaling option is active

Aquifers

- AQUCHGAS Allows a constant head gas aquifer to be defined
- AQUCT Carter-Tracy aquifers can now be modified in the SCHEDULE section
- AQUFETP Fetkovich aquifers can now be modified in the SCHEDULE section.

Well model

- DELAYACT Allows actions to be triggered a given time after other actions have been triggered.
- VFP tables that contain Tubing Head Temperature values may now be read in and used to report temperature values for production wells via the summary mnemonic WTHT or an Open ECLIPSE query. The keyword WHTEMP assigns temperature VFP tables to wells.

- The ACTION family of keywords has been extended to include an ACTIONX keyword. The keyword marks the start of a set of SCHEDULE section keywords that are to be stored for later processing when a set of conditions specified in this keyword are met. This keyword is a generalization of the existing ACTION (G,R,W,S) type keywords and offers the following new features:
  - Multiple triggering conditions may be specified that are combined using Boolean AND and OR operators.
  - Conditions on quantities associated with wells, groups and regions may be combined in a single ACTIONX keyword.
  - Conditions may be set on the day, month or year of the simulation.
  - A condition may compare one variable quantity against another. (For example, a condition might be that a well's GOR is greater than the current value of its controlling group's GOR.)
- A switch has been added to the GUIDERAT keyword for calculating guide rates, that allows production potential rates of free gas (at well-block conditions) to be used in the gas-oil potential ratio of the guide rate formula. By default, total gas production potential rates are used for this ratio.

Output and restarts

- The state of a well may now be reported to the summary file using the mnemonic WSTAT.
- It is now possible to output the velocities to the restart files for all runs (not just a POLYMER or FOAM run) using the mnemonic VELOCITY under the RPTRST keyword.
- Using the mnemonic VELFLO, output of $1/(\text{area} \times \text{porosity})$ will be output to the print file using the RPTGRID keyword. This can only be done if velocity or flow output is also requested via use of summary mnemonics or restart output.
- All messages, warnings, errors etc. output to the print file are now also output to a separate file, extension MSG, in XML format. The feature is intended primarily as an aid for simulation post-processors, but can be controlled by the user with the MSGFILE keyword.
- New SUMMARY section block mnemonics have been added in order to output inter-block flows and velocities.
- New SUMMARY section mnemonics have been added in order to output solution (dissolved) gas and free gas production potentials for groups and wells.
- New SUMMARY section mnemonics have been added to output:
  - the ratio of produced water to injected water
  - the related recovery efficiency.

Performance

- A TSTEP keyword now has a maximum of 1000 entries.

Tracers

- The ACTION, ACTIONG and ACTIONW keywords can now be triggered with tracer production rates/concentrations.
- New keywords CECONT, WECONT and GECONT allow economic controls on connections/wells/groups based on tracer rates and concentrations.
- WCUTBACT and GCUTBACT have been added to allow cutback and cutback reversal of rate targets, based on tracer rates and concentrations.

- Diffusion can now be applied to all tracers. See the TRDIF keyword.
- Diffusion can now be used in the same run as the TRACTVD keyword. (In previous releases the solution technique would have reverted to the fully implicit method.)
- Adsorption and decay can now be used in the same run as the TRACTVD keyword.
- There are new summary section quantities (F,G,W,C)S(P,I)C for brine production and injection concentrations.
- A form of dispersion has now been added to the tracer calculation. This can be turned on, and dimensioned, with the DISPDIMS keyword. The required data is input with the TRDIS and DISPERSE keywords. For use with the Brine option there is also the DSPDEINT keyword which causes the dispersion tables to be interpolated on water density as opposed to tracer concentration. An alternate functional form for dispersion can be enabled using the DIAGDISP keyword. When the dispersion is only being used in order to apply constant but non-homogeneous diffusion, substantial CPU savings can be made by 'warning' the code and using the TRNHD keyword.
- The TRACTVD option can now be used in runs with LGRs. The only exceptions are runs with diffusion or dispersion.
- The user can increase the number of mini time-steps taken with the TRACTVD option by setting flag 84 in the OPTIONS keyword.
- Multi-segmented wells can now be used when the diffusion control flag is on (item 5 of the TRACERS keyword). Note that a standard well model is still used within the tracer equations and hence the multi-segmented well tracer summary section keywords should not be used.The wells in the actual reservoir simulation itself will be solved as multi-segmented wells.

Multi-segment Wells

- The user can choose between two sets of default parameter values for the Drift Flux slip model - the original set and an optimized set proposed by Shi *et al*. [Ref. 28]. Use keyword WSEGDFMD to select the new parameter set. There is also an option to account for the effect of pipe inclination via an input table of scaling factor vs. deviation angle (see keyword WSEGDFIN), instead of using the formula of Hasan and Kabir [Ref. 32]; the built-in default table reflects the results of Shi *et al*.

Reservoir Coupling

- MPI may now be used as the message passing system for Reservoir Coupling, allowing the user to simulate slave reservoirs on multiple processors, in parallel. Accordingly, the SLAVES keyword has been modified to allow the user to specify the number of processors on which a slave reservoir is to be run.

ECLIPSE 300

RUNSPEC

- The MEMORY keyword is now ignored by ECLIPSE 300; dynamic memory is used instead. This means that it is not necessary to estimate the memory usage in advance of the run. This makes it easier to run in parallel with best possible memory usage on each processor.
- The ACTPARAM keyword allows the default fractional equality tolerance to be changed. This tolerance is used in deciding whether quantities are equal in an ACTIONX condition.
- The MSGFILE keyword turns on or off the output of messages to the MSG file.

GRID

- It is now possible to modify the power coefficients and to select the permeability direction in the J-function expression. See the `JFUNC` keyword.

Properties and equilibration

- The diffusion of components driven by a temperature gradient can now be simulated in compositional simulations with keywords `DIFFTGAS` and `DIFFTOIL`.
- A modified form of the Lorentz-Bray-Clark viscosity formula can now be used. Extra parameters for the new formula can be entered with the `VCOMPACT` keyword.
- `EPSDBGS` Allows one to write out the scaled relative permeability curves for multiple sets of grid blocks when the End Point Scaling option is active
- `SOMGAS` Allows one to specify the minimum oil saturation used in the STONE I 3-phase Relative Permeability Model, as a function of the gas saturation.
- `SOMWAT` Allows one to specify the minimum oil saturation used in the STONE I 3-phase Relative Permeability Model, as a function of the water saturation.
- The non-darcy flow option (See `VDFLOW` keyword) now applies to non-neighbor connection flows. If the non-neighbor connection is entered explicitly using the `NNC` keyword, an area and linking permeability needs to be supplied.

Geomechanics option

- Scale damage tables can be set up with the keyword `SCDATABG` when using the Geomechanics option, keyword `GEOMECH`.

Geomechanics option

- Extensive changes were made to `GEOYLDF` for parameters used in the Geomechanics finite element plasticity calculation.

REGIONS

- Transport coefficient table regions (specified using keyword `ALPHA`) may now be assigned to individual grid cells using the new `ALPHANUM` keyword. This replaces the previous behavior of following the `PVTNUM` definition.

Initialization

- The `TEMPI` keyword can now be used with explicit initialization in non-thermal runs to set a different temperature in each cell.

SCHEDULE

Well model

- The ACTION family of keywords has been extended to include an ACTIONX keyword. The keyword marks the start of a set of SCHEDULE section keywords that are to be stored for later processing when a set of conditions specified in this keyword are met. This keyword is a generalization of the existing ACTION(G, R, W, S) type keywords and offers the following new features:
    - Multiple triggering conditions may be specified that are combined using Boolean AND and OR operators.
    - Conditions may be set on well connection quantities
    - Conditions may be set on the day, month or year of the simulation.
    - Conditions on quantities associated with wells, groups and regions may be combined in a single ACTIONX keyword.
    - A condition may compare one variable quantity against another. (For example, a condition might be that a well's GOR is greater than the current value of its controlling group's GOR.)
    - Conditions may also be set on the combined properties of a static (see WLIST keyword) or dynamic (see WLISTDYN keyword) list of wells. Grouping wells together using lists for the purposes of defining controlling conditions in this keyword provides greater flexibility than conventional well groups.
- The WLISTDYN keyword may be used set up a dynamic well list. A well list may be entered wherever a well name root can be entered in the SCHEDULE section well keywords. Well lists may also be used in the ACTIONX keyword to define a temporary grouping of wells for control purposes. A dynamic list differs from a static, user-defined well list (see WLIST keyword) in that the membership of the list is automatically reset at the end of each timestep. The membership is determined by a condition on a well-type quantity. For example, the list might be made up of all the wells in a particular group that have a water cut greater than 0.7.
- Combined rates, totals and ratios for all the wells in a well list may be written to the summary file using an appropriate group summary mnemonic followed by the well list name (see the table "Well list quantities (ECLIPSE 300)" in the SUMMARY section of the ECLIPSE Reference Manual Data File Overview chapter).
- The keyword WCOMPRES may be used to reduce the memory requirements of a simulation that contains a large number of wells, some of which have many connections, but most have relatively few connections. Internal well arrays will be dimensioned to the actual number of connections in the simulation rather than the maximum possible number defined via the WELLDIMS keyword. The number of connections in the simulation may be specified with this keyword or estimated from a pre-sweep of the SCHEDULE section data.
- VFP tables that contain Tubing Head Temperature values may now be read in and used to report temperature values for production wells via the summary mnemonic WTHT or an Open ECLIPSE query. The keyword WHTEMP assigns temperature VFP tables to wells.
- The generalized pseudo-pressure (GPP) option for well inflow calculations may now be enabled for individual wells by entering GPP in item 8 of the WELSPECS keyword. The keyword PSEUPRES will continue to exist and may be used to enable this option for all wells.
- Forchheimer effects (part of the velocity dependent relative permeability (VDRP) model) may now be included in the generalized pseudo-pressure (GPP) calculation that accounts for condensate blocking in well inflow. The GPP option is enabled via item 8 of the WELSPECS keyword and VDRP effects may be included in this calculation by entering YES in item 4 of the PICOND keyword. The VDRP model is specified via the VELDEP, VDKRO and VDKRG keywords.

- The production balancing method of group control has been implemented, whereby a group's reservoir fluid volume production rate target is set to a specified fraction of its reservoir fluid volume injection rate (see keywords GCONPROD and GCONPRI). Production balancing requires ECLIPSE to solve all the injectors before the producers at each time step, instead of the other way round. This places restrictions on some other control modes that are allowed to operate simultaneously, as described in GCONPROD and GCONPRI.
- The keyword GEFAC may now be used to account for the 'downtime' of all the wells in a group that are synchronized such that the group either flows with its full rate or nor at all.
- The keyword WORKTHP may now be used to instruct ECLIPSE to perform a workover if a well is unable to produce at its THP limit, to reduce its water cut. The workover is performed at the end of the time step in which the well dies, and the well is revived at the beginning of the next time step to see if it can produce at the reduced water cut
- Economic limits may now be set on the maximum mole fraction of a component in the wellstream of a production well using the keyword WECONCMF.
- A PVT region number can now be set for each well using item 11 of keyword WELSPECS or item12 of keyword WELSPECL.
- A new well model for injectors is activated by item 16 of the keyword WELSPECS and item 17 of the keyword WELSPECL. The model can be used to overcome convergence problems that arise when connecting grid blocks have high mobilities. Typically this might be the case for thermal runs, injecting steam at high pressure.

Thermal option

- Keyword WSEGEXSS can now be used to inject steam in a multi-segment well at a specified quality if either water import at a specified external pressure or water import at a specified rate is chosen.

Geomechanics option

- A well sanding prediction is available as a Schlumberger service. WELLSMA compares predicted drawdowns at each completion of a well against critical drawdowns that are calculated by the Sand Management Advisor (SMA) onset of sanding or failure criterion. If the completion drawdown exceeds the critical drawdown, either the completion can be assumed to be shut in or actions with the keyword ACTIONX can be specified to control well rates to prevent the onset of sanding. In the latter case, for example, well rates at a producer can be lowered if the actual completion drawdown exceeds a percentage, for example 50%, of the SMA predicted critical drawdown for sanding at any completion. A license to run this calculation is only available internally to Schlumberger staff.

Geomechanics option

- Keyword WSCTAB can be used to link a scale damage table set with keyword SCDATABG with a well.

Output and restarts

- The state of a well may now be reported to the summary file using the mnemonic WSTAT.
- When the INIT keyword is used, and there are non-neighbor connections in the model, the NNC transmissibility information will also be output to the INIT file (under mnemonic TRANNNC).

Diffuse option

- When the INIT keyword is used, and there are non-neighbor connections in the model, the NNC diffusivity information will also be output to the INIT file (under mnemonic DIFFNNC).

Thermal option

- When the INIT keyword is used, and there are non-neighbor connections in the model, the NNC thermal transmissibility information will also be output to the INIT file (under mnemonic HEATNNC).
- The EXCEL keyword can now be used to request run summary output in Excel format.

- All messages, warnings, errors etc. output to the print file are now also output to a separate file, extension MSG, in XML format. The feature is intended primarily as an aid for simulation post-processors, but can be controlled using the `MSGFILE` keyword.

Performance

- `ZIPPY2` time-step selection has been added to E300.
- `ZIPP2OFF` has been added to E300 to turn off `ZIPPY2`.
- There is now a maximum of a 1000 entries in a `TSTEP` keyword.

Geomechanics option

- A finite element stress calculation can be specified. This option allows second-order rock displacement accuracy with corner-point geometry when the gridding is skewed and decidedly non-orthogonal. It is available partly coupled in time, has more extensive boundary condition options than the finite difference stress calculation available with the `STRESSBC` keyword, and will allow inactive cells. LGR and faults are not implemented for this release. In addition, a plastic calculation featuring Mohr-Coulomb, Drucker-Praeger with hardening/softening, but no cap model, can be specified with the `GEOYLDF` keyword. The third item of the `GEOMECH` keyword activates this. This option is compatible with the well sanding, keyword `WELLSMA`, options. The multi-grid solver, keyword `SAMG`, must be specified.
- A second porosity-stress relation is available which allows the rock matrix to be compressible. See the fourth item of the `GEOMECH` keyword for more information.
- A well sanding prediction is available as a Schlumberger service. `WELLSMA` will compare predicted drawdowns at each completion of a well against critical drawdowns that are calculated by the Sand Management Advisor (SMA) onset of sanding or failure criterion. If the completion drawdown exceeds the critical drawdown, either the completion can be assumed to be shut in or actions with the keyword `ACTIONX` can be specified to control well rates to prevent the onset of sanding. In the latter case, for example, well rates at a producer can be lowered if the actual completion drawdown exceeds a percentage, for example 50%, of the SMA predicted critical drawdown for sanding at any completion. A license to run this calculation is only available internally to Schlumberger staff.
- The fixed grid option is no longer available.

Multi-segment Wells

- You choose between two sets of default parameter values for the Drift Flux slip model - the original set and an optimized set proposed by Shi *et al*. [Ref. 28]. Use keyword `WSEGDFMD` to select the new parameter set. There is also an option to account for the effect of pipe inclination via an input table of scaling factor vs. deviation angle, instead of using the formula of Hasan and Kabir; the built-in default table reflects the results of Shi *et al*.

Thermal option

- Keyword `WSEGEXSS` can now be used to inject steam in a multi-segment well at a specified quality if either water import at a specified external pressure or water import at a specified rate is chosen.

Reactions

- Chemical reactions can now be used with blackoil (for example, with solid components) as well as compositional models.
- Reaction rates that depend on deviation from equilibrium can now be used with blackoil, EoS and K-value, as well as Thermal models.

- Reaction rates can now be multiplied by terms dependent on tracer concentrations. An extra item is required in keyword REACTION to define the number of tracer dependent terms, which are then specified with keyword REACTRAC. The tracer multipliers can be output using the mnemonic TREACM in keywords RPTRST or RPTSCHED.

Solid option

The solid option can now be used in blackoil, EoS and compositional K-value runs.

- Keyword CVTYPE can now be used to specify solid components in blackoil, EoS and K-value runs, as well as thermal simulations. Keyword CVTYPES can now be used to specify solid components at surface conditions in thermal, blackoil, EoS and K-value runs.
- The keywords that describe the solid density, SCREF, SDREF, SPREF, STREF and STHERMX1 can now be used in blackoil, EoS and K-value runs.
- Initial values for solid components can now be defined with keywords, SSOLID and SMF.
- A solid transport model has been added: solids suspended in the oil phase will be transported with the oil, while solids adsorbed to the rocks will remain immobile. Keywords SOLIDADS and ROCKDEN can be used to determine the ratio of disobeyed to suspended solids. The suspended solid fraction in a grid block can be output to the summary file with mnemonic BSSFRAC, or to the restart and print files by using mnemonic SSFRAC with the RPTRST, RPTSCHED and RPTSOL keywords. Produced solids can be output to the summary file by volume (F,G,W)SV(R,T) or by mass (F,G,W)SM(R,T).
- Plugging of the pores by solids can now be modeled by using a mobility multiplier. There are two ways of specifying the multiplier, either as a function of the adsorbed solid concentration (SOLIDMMC) or as a function of the adsorbed solid saturation (SOLIDMMS). The mobility multiplier can be output to the summary file with mnemonic BSMMULT, or to the restart and print files by using mnemonic SMMULT with the RPTRST, RPTSCHED and RPTSOL keywords.
- Keyword REACSORD can be used with chemical reactions to modify the way the reaction rate depends on the solid saturation.

Temperature option

- The ECLIPSE 100 blackoil Temperature option can now also be used in ECLIPSE 300. Keyword TEMP is used to initiate the option; keyword THCONR can be used to specify the thermal conductivity of the reservoir; keywords SPECHEAT and SPECROCK can be used to specify specific heats of the fluids and the rock; the temperature dependencies of the oil and water viscosities can be set with keywords OILVISCT and WATVISCT; and the initial temperatures can be set with either RTEMPA or RTEMPVD.

Thermal option

- Thermal simulations can now be run with IMPES and AIM. New items have been added to the AIMCON keyword to allow the AIM state selection to depend on temperature change. The ratio of the actual temperature change to target RATT can be output using the RPTRST keyword.
- In foamy oil simulations an oil (or solid) phase component is used to represent trapped gas. The trapped gas saturation can now be stored as a property SFOIL (or SFSOL for ‘foamy solid’) which can be output with the RPTRST, RPTSCHED and RPTSOL keywords or with the summary mnemonic BSFOIL or BSFSOL.
- Keyword ZFACTORS can now be used to define component Z-factors at the surface.

- Keyword WCONINJE can now be used to inject steam-gas. The temperature data is set by using the keyword WINJTEMP.

Multi-segment wells

- Keyword WSEGEXSS can now be used to inject steam in a multi-segment well at a specified quality if either water import at a specified external pressure or water import at a specified rate is chosen.
- A new well model for injectors is activated by item 16 of the keyword WELSPECS and item 17 of the keyword WELSPECL. The model can be used to overcome convergence problems that arise when connecting grid blocks have high mobilities. Typically this might be the case for thermal runs, injecting steam at high pressure.
- A modified method can be used to update thermal multi-segment well solutions by using switch 83 of the OPTIONS3 keyword. This modified method can improve convergence for difficult problems, but may be slower in more straightforward cases.
- Three new aliases DREFT, PREFT and TREFT (for DREF, PREF and TREF) have been created to allow ECLIPSE to read in thermal properties output by PVTi.
- Temperature-dependent heater rates can now be specified with the HEATER and HEATERL keywords.
- Thermal simulations can now be run with black oil properties. Keywords BLACKOIL and THERMAL should both appear in the RUNSPEC section. Fluid properties should be specified using Temperature option keywords.

Behavioral changes for 2004A

These may cause changes to the results of a simulation, or alter the form or content of the output.

ECLIPSE 100

Output and restarts

- If OFM output is requested and the MAPAXES keyword is present, the X,Y locations of the wells are output relative to the map origin.
- Messages output to the print file during the course of a run (warnings, errors...) are now also output to a separate file, extension MSG, in XML format. The feature can be controlled using the MSGFILE keyword.

Tracers

- When the TRACTVD and TRDIF keywords were used in pre-2004A releases the TRACTVD keyword would have effectively been ignored for any tracer with diffusion; the solution method reverting to the fully implicit method. This is no longer the case; the flux limiting scheme will be used for the convective terms, whilst the diffusive terms are calculated with a standard central difference formula.
- The brine tracer option now no longer automatically uses the cascade solver. It will now use the solver method actually chosen by the user with item 5 of the TRACERS keyword.
- Pre-2004A releases automatically turned on dispersion control (Item 5 in the TRACERS keyword) if LGRs were present in the run. This no longer happens; if dispersion control is required it must be explicitly set.

ECLIPSE 300

Output and restarts

- Messages output to the print file during the course of a run (warnings, errors...) are now also output to a separate file, extension MSG, in XML format. The feature can be controlled using the `MSGFILE` keyword.

Properties

- The `NNC` keyword has been modified to be more compatible with ECLIPSE 100 usage.
- The non-darcy flow (see `VDFLOW`) now applies to non-neighbor connections.
- Where both `JFUNC` and `PCW` or `PCG` are used to specify the maximum scaled capillary pressures, `JFUNC` now takes precedence. Switch 87 in `OPTIONS3` makes the simulation revert to pre-2004A behavior.
- The records of the `ROCK` keyword now correspond to the region numbers set in `PVTNUM`, as stated in the `ROCK` keyword definition. Previously these were incorrectly based on the rock compaction regions defined by `ROCKNUM`.

Well model

- In runs with more than one PVT region, a PVT region number is set for each well, according to the PVT region number of the completion cells. Item 78 of keyword `OPTIONS3` allows the user to override this PVT region number, and set a new PVT region number for all wells. Prior to 2004A the properties of fluids in the wells were calculated using a PVT region number of 1. Therefore setting this option flag to 1 will give the pre-2004A behavior. Alternatively, the PVT region number can now be set individually for each well using item 11 of keyword `WELSPECS` or item12 of keyword `WELSPECL`.
- If a group is set to be not free to respond to a higher level rate target (see item 8 in `GCONPROD` or `GCONINJE`), this restriction will now remain in force even when the group itself no longer controls the production or injection and becomes passive. Prior to 2004A, when the group became passive, wells and groups at a lower level became subject to rate control from groups at a higher level. To revert to the pre-2004A state, set item 79 of the keyword `OPTIONS3` to 1.
- The method used to determine the top-up phase injection targets for a group has been simplified and made more robust. To revert to the pre-2004A state, set item 79 of the keyword `OPTIONS3` to 2.
- Well efficiency factors are now taken into account when calculating a group's potential rates. To revert to the pre-2004A state, set item 79 of the keyword `OPTIONS3` to 3.
- Both Capillary Number and Forchheimer (non-Darcy flow) velocity-dependent relative permeability effects are now incorporated in the Generalized Pseudo-Pressure calculation when item 4 of `PICOND` is set to `YES`. Prior to the 2004A release, only Capillary Number effects were included in this calculation. To revert to the pre-2004A state, set item 82 of the keyword `OPTIONS3` to 2.
- If a well has switched from rate control to pressure control and died in the same timestep, the timestep will now be chopped. This will result in less fluid being removed from the connected grid blocks of a production well and thus the well may move on to pressure control rather than being immediately shut in.The number of times that a timestep may be consecutively chopped in this way is controlled by item 84 of the `OPTIONS3` keyword.
- For the keyword `COMPMOBI` the default value for the total voidage mobility, item 6, has changed from 10 cP^{-1} to 1 cP^{-1}.

Multi-phase flash

- Keyword `MULTIPHA` must now be used if the multi-phase flash is used for asphaltene prediction.

Potential calculations

- The average phase potentials for a region or field are now weighted by the relevant phase saturation, as documented in the chapter “Potential Calculations” of the ECLIPSE Technical Description, rather than by the hydrocarbon volume. To revert to the pre-2004A state, set switch 86 of the keyword `OPTIONS3` to 1.

Thermal option

- The Appleyard chop is used to improve non-linear convergence by chopping non-linear steps that produce a significant change in saturations or temperatures. In 2004A the saturation-based Appleyard chop was enhanced for thermal simulations. This enhancement can be switched off by using item 18 of the `OPTIONS3` keyword.

Geomechanics option

- The sign of user specified boundary stresses must be determined from whether the stress is compressive (-ve) or tensile (+ve). This differs from the previous releases where direction of the boundary was used. Consequently, on '−' boundaries, the sign of stresses which are meant to apply a force in the coordinate direction will need to be negative (compressive). Stresses on '+' boundaries remain the same.
- A fixed grid option is no longer available.

Non-Darcy (Forchheimer)

- The implementation of the Forchheimer correction (inertia effects) has now been improved. These non-Darcy effects may have been underestimated in previous implementations.

New keywords for 2004A

ECLIPSE 100

RUNSPEC section

NRSOUT	Sets the maximum number of quantities that can be output to a restart file at each report step.
DISPDIMS	Turns on and dimensions tracer dispersion option.
ACTPARAM	Sets parameters for ACTION type keywords.
MSGFILE	Turns on/off output of messages to the MSG file.

GRID section

MINNNCT	Specifies a cutoff transmissibility below which non-neighbor connections are deleted.

PROPS section

TRDIS	Inputs dispersion table numbers for tracers
DISPERSE	Dispersion tables for tracers
DSPDEINT	Dispersion tables for brine interpolated on water density instead of tracer concentration
DIAGDISP	Enables alternate functional form for dispersion.
TRNHD	Informs the code that for this tracer dispersion is actually constant coefficient and diagonal
EPSDBGS	Writes out the scaled relative permeability curves for multiple sets of grid blocks.

Solution section

AQUCHGAS	Constant head gas aquifers

SUMMARY section

(F,G,W,C)S(P,I)C	Brine (production, injection) concentrations
WTHT	Tubing Head Temperature for production wells (see WHTEMP keyword)
WSTAT	Indicates well state (Including PSHUT/PSTOP)
(F,G,W)GPP(S,F)	Solution and free gas production potential rates
(B)FLO(O,W,G)(I,J,K)	Output of oil/water/gas inter-block flows in positive I/J/K direction
(B)VEL(O,W,G)(I,J,K)	Output of oil/water/gas velocities in positive I/J/K direction
(F,G,W)WPIR	Output ratio of produced water to injected water as a percentage

SCHEDULE section

ACTION, ACTIONG, ACTIONW	(F,G,W)(C,S,T)P(R,C) have been added to these keywords to enable actions to be triggered on polymer/salt/tracer production rates and concentrations.
ACTIONX	A more flexible, multi-conditioned version of the ACTION(G,R,W,S) type keywords.
CECONT	Economic control of connections based on maximum tracer rates and concentrations.
WECONT	Economic control of wells based on maximum tracer rates and concentrations.
GECONT	Economic control of groups based on maximum tracer rates and concentrations.
WCUTBACT	Allows well cutbacks based on tracer rates and concentrations.
GCUTBACT	Allows group cutbacks based on tracer rates and concentrations.
DELAYACT	Allows actions to be triggered a given time after other actions have been triggered
WHTEMP	Specifies a VFP table for Tubing Head Temperature calculations or defines a constant tubing head temperature value for production wells.
WSEGDFMD	Defines the default values for the set of parameters for the Drift Flux slip model for Multi-segment Wells.
WSEGDFIN	Inputs a table of inclination scaling factors vs. deviation angle for Multi-segment Well segments.
AQUCHGAS	Constant head gas aquifers

ECLIPSE 300

RUNSPEC section

MULTIPHA	This keyword is used to indicate that the multi-phase flash will be used for asphaltene prediction.
TEMP	This keyword is used to initiate the blackoil Temperature option.
WCOMPRES	This keyword reduces the memory requirements of a simulation that contains a large number of wells, some of which have many connections, but most have relatively few connections.
ACTPARAM	This keyword sets parameters for ACTION type keywords.
Geomechanics option SCDPDIMS	This keyword can be used to specify the dimensions of scale damage tables, keyword SCDATABG, that can be set up together with the Geomechanics, keyword GEOMECH, and proppant flowback, keyword PROPFLOW, options.
MSGFILE	Turns on/off output of messages to the MSG file.

GRID section

Geomechanics option	PROPFLOW	A proppant flowback model is available that will calculate both the closure and drag forces on a proppant pack. The keyword PROPFLOW will activate this and allow the user to both predict the onset of proppant flowback as well as set up a scale damage table (keywords SCDATABG and WSCTAB) for a specified well if desired.

PROPS section

	DIFFTGAS	This keyword can be used to model the gas phase diffusion driven by a temperature gradient.
	DIFFTOIL	This keyword can be used to model the oil phase diffusion driven by a temperature gradient.
Thermal option	DREFT	This keyword is an alias for DREF which allows ECLIPSE to read in thermal properties output by PVTi.
Thermal option	PREFT	This keyword is an alias for PREF which allows ECLIPSE to read in thermal properties output by PVTi.
Solid option	REACSORD	This keyword can be used with chemical reactions to modify the way the reaction rate depends on the solid saturation.
	REACTRAC	This keyword can be used to define tracer dependent reaction rates.
Solid option	SOLIDADS	This keyword can be used to specify the ratio of adsorbed solids to suspended solids.
Solid option	SOLIDMMC	This keyword can be used to specify a mobility multiplier as a function of adsorbed solid concentration.
Solid option	SOLIDMMS	This keyword can be used to specify a mobility multiplier as a function of adsorbed solid saturation.
Temperature option	SPECHEAT	This keyword can be used to specify specific heats of the fluids.
Temperature option	SPECROCK	This keyword can be used to specify specific heats of the rock.
Thermal option	TREFT	This keyword is an alias for TREF which allows ECLIPSE to read in thermal properties output by PVTi.
	VCOMPACT	This keyword specifies the compact or close-packed volumes which are used in a modified form of the Lorentz-Bray-Clark viscosity formula.
Thermal option	ZFACTORS	This keyword can be used to define component Z-factors at the surface.
	EPSDBGS	This keyword writes out the scaled relative permeability curves for multiple sets of grid blocks.
	SOMGAS	This keyword allows the user to specify the minimum oil saturation used in the STONE I 3-phase Relative Permeability Model, as a function of the gas saturation.
	SOMWAT	This keyword allows the user to specify the minimum oil saturation used in the STONE I 3-phase Relative Permeability Model, as a function of the water saturation.
Geomechanics option	SCDATABG	This keyword can be used together with the Geomechanics option, keyword GEOMECH, and the proppant flowback option, keyword PROPFLOW, to account for well scale damage caused by proppant flowback. Dimensions of this table must be set with the SCDPDIMS keyword.

Regions section

	ALPHANUM	This keyword sets the region numbers for the transport coefficient tables.

SOLUTION section

Temperature option	RTEMPA	This keyword sets a constant initial temperature.
Temperature option	RTEMPVD	This keyword sets the initial temperatures as a function of depth.
Solid option	SSOLID	This keyword can be used to define initial solid saturations.
Solid option	SMF	This keyword can be used to define initial mole fractions of solid components

SUMMARY section

Thermal option	BSFOIL	saturation of foamy oil
Thermal option	BSFSOL	saturation of foamy solids
Solid option	BSMMULT	solid mobility multiplier
Solid option	BSSFRAC	suspended solids fraction
Solid option	(F,G,W)SM(R,T)	Field, Group, Well Solid Mass production Rate, Total
Solid option	(F,G,W)SV(R,T)	Field, Group, Well Solid Volume production Rate, Total
	WTHT	Tubing Head Temperature for production wells (see WHTEMP keyword)
	WSTAT	Indicates well state (including PSTOP/PSHUT).
	GMWL	Count of well grouped together in a well list.
	EXCEL	This keyword requests run summary output in Excel format.

SCHEDULE section

	ACTIONX	A more flexible, multi-conditioned version of the ACTION(G, R, W, S) type keywords.
	GEFAC	Group efficiency factors.
	WECONCMF	Sets economic limits on a component mole fraction in the wellstream of a production well.
	WHTEMP	Specifies a VFP table for Tubing Head Temperature calculations or defines a constant tubing head temperature value for production wells.
Thermal option	WINJTEMP	Specify temperature data for injection wells.
	WLISTDYN	Dynamic well lists
	WORKTHP	Perform a workover if a well is unable to produce at its THP limit, to reduce its water cut
	WSEGDFIN	Inputs a table of inclination scaling factors vs. deviation angle for Multi-segment Well segments.
	WSEGDFMD	Defines the default values for the set of parameters for the Drift Flux slip model for Multi-segment Wells.
	ZIPPY2	Time-step selection has been added to ECLIPSE 300.
	ZIPP2OFF	Has been added to ECLIPSE 300 to turn off ZIPPY2.

Geomechanics option	WELLSMA	A well sanding option is available internally to Schlumberger staff that allows prediction of sanding at an open-hole perforation or completion. The Geomechanics option, keyword GEOMECH, must be specified. Keyword WELLSMA will compare predicted drawdowns at each completion of a well against critical drawdowns that are calculated by the Sand Management Advisor (SMA) onset of sanding or failure criterion. If the completion drawdown exceeds the critical drawdown, either the completion can be assumed to be shut in or actions with the keyword ACTIONX can be specified to control well rates to prevent the onset of sanding. A license to run this calculation is only available internally to Schlumberger staff.
Geomechanics option	WSCTAB	Together with the Geomechanics option, keyword GEOMECH, and the proppant flowback option, keyword PROPFLOW, this keyword assigns a well to a particular scale damage table which is entered with the keyword SCDATABG.

Altered keywords for 2004A_1

ECLIPSE 100

RUNSPEC section

ACTDIMS	New item (4) to set the maximum number of conditions in an ACTIONX keyword.

GRID section

JFUNC	New data items to modify the J-function expression
RPTGRID	This keyword has been extended to output 1/(area*porosity), with the VELFLO mnemonic. This can only be done if velocity or flow output is also requested via use of summary mnemonics or restart output.

PROPS section

TRDIF	Can now be used with all tracers

SUMMARY section

OFM	Now references the X,Y location to the map origin if the MAPAXES keyword is present.

SOLUTION section

RPTRST	The VELOCITY mnemonic can now be used for any run, not just a POLYMER or FOAM run. The relevant oil/water/gas velocities will be output to the restart file for each direction (I/J/K)

SCHEDULE section

	VFPPROD	This keyword has been extended to include Tubing Head Temperature (THT) value in the VFP table. If the VFP table carries the tubing head temperature value, Item 9 of record 1 should read 'TEMP'.
	OPTIONS	New item 84 for increasing the number of mini time-steps taken by the TRACTVD option
	RPTRST	The VELOCITY mnemonic can now be used for any run, not just a POLYMER or FOAM run. The relevant oil/water/gas velocities will be output to the restart file for each direction (I/J/K)
Multi-segment wells	WSEGDFPA	New item 11 defining the exponent in the formula for oil-water drift velocity in the Drift Flux model for Multi-segment Wells.
	TSTEP	Up to 1000 entries are now allowed.
Reservoir Coupling	SLAVES	This keyword has been extended to allow the user to specify the number of processors to use for each slave reservoir (if they are to be run in parallel).
	GUIDERAT	New item 11 to switch to using free gas potentials in the gas-oil potential ratios.
	ACTION	Can now be triggered on tracer rates and concentrations.
	ACTIONG	Can now be triggered on tracer rates and concentrations.
	ACTIONW	Can now be triggered on tracer rates and concentrations.
	AQUCT	Carter-Tracy aquifers can now be modified in the SCHEDULE section.
	AQUFETP	Fetkovich aquifers can now be modified in the SCHEDULE section.

ECLIPSE 300

RUNSPEC section

ACTDIMS	New item (4) to set the maximum number of conditions in an ACTIONX keyword.
REACTION	An extra item can be used to define the number of tracer-dependent terms, which are then specified with keyword REACTRAC. This keyword can now be used in blackoil simulations, and items 2 and 3 controlling equilibrium deviation reactions can now be used in blackoil, EoS and K-value models.
WELLDIMS	New item (11) to set the maximum number of well lists that a well may belong to.
WELLDIMS	New item (12) to set the maximum number of dynamic well lists in a simulation.
MEMORY	The keyword is now ignored by ECLIPSE 300; dynamic memory is used instead. This makes it easier to run in parallel with best possible memory usage on each processor.
AIM	This keyword can now be used with the Thermal option.
IMPES	This keyword can now be used with the Thermal option.
SAMG	This keyword has two new items to define a tolerance and a strategy for reusing coarse grid and smoothing operators.
GEOMECH	A finite element stress calculation can be activated instead of the finite difference calculation. In addition, a new porosity-stress relation is available that accounts for rock/matrix compressibility.
BLACKOIL	This keyword can now be used with the Thermal option.

GRID section

INIT	If there are non-neighbor connections in the model when using this keyword, the NNC transmissibilities, NNC diffusivities and NNC thermal transmissibilities will also be output to the INIT file (under the mnemonics TRANNNC, DIFFNNC and HEATNNC respectively).
JFUNC	New data items to modify the J-function expression.
NNC	It is now possible to enter data for the area and linking permeability associated with the NNC for use in the non-darcy flow model. (Data items 16 and 17).
NNC	It is now possible to enter NNC diffusivity and thermal conduction transmissibility in data items 14 and 15. This is more compatible with ECLIPSE 100 behavior.
Temperature option THCONR	This keyword can now be used with the Temperature option (as well as the Thermal option) to specify the thermal conductivity of the reservoir.

PROPS section

	CVTYPE	This keyword can now be used to specify solid components in blackoil, EOS and K-value runs.
	CVTYPES	This keyword can now be used to specify solid components at surface conditions in Thermal, blackoil, EoS and K-value runs.
Reactions	EQLDKVCR	This keyword can now be used with blackoil, EoS and K-values
Reactions	EQLDREAC	This keyword can now be used with blackoil, EoS and K-values
Geomechanics finite element option	GEOYLDF	This keyword has substantially changed to allow more parameters for the finite element plasticity option.
Temperature option	OILVISCT	This keyword can now be used with the Temperature option (as well as the Thermal option) to specify the temperature dependencies of the oil viscosity.
Reactions	REACACT	This keyword can now be used with blackoil
Reactions	REACCORD	This keyword can now be used with blackoil
Reactions	REACPHA	This keyword can now be used with blackoil
Reactions	REACPORD	This keyword can now be used with blackoil
Reactions	REACRATE	This keyword can now be used with blackoil
	ROCK	The records of the ROCK keyword now correspond to the region numbers set in PVTNUM, as stated in the ROCK keyword definition. Previously these were incorrectly based on the rock compaction regions defined by ROCKNUM.
Reactions	ROCKDEN	This keyword can now be used determine the ratio of disobeyed solids to suspended solids.
Solid option	SCREF	This keyword can now be used to determine the solid compressibility in blackoil, EoS and K-value runs.
Solid option	SDREF	This keyword can now be used to determine the solid density in blackoil, EoS and K-value runs.
Solid option	SPREF	This keyword can now be used to determine the reference pressure for the solid density calculation in blackoil, EoS and K-value runs.
Reactions	STOPROD	This keyword can now be used with blackoil
Reactions	STOREAC	This keyword can now be used with blackoil
Solid option	STREF	This keyword can now be used to determine the reference temperature for the solid density calculation in blackoil, EoS and K-value runs.
Solid option	STHERMX1	This keyword can now be used to determine the solid thermal expansion coefficient in blackoil, EoS and K-value runs.
	VDFLOW	This keyword now applies to non-neighbor flows as well as standard flows.
Temperature option	WATVISCT	This keyword can now be used with the Temperature option (as well as the Thermal option) to specify the temperature dependencies of the water viscosity.

SOLUTION section

RPTSOL has the following new arguments:

Thermal option	SFOIL	foamy oil saturation

Thermal option	SFSOL	foamy solid saturation
Solid option	SMMULT	solid mobility multiplier
Solid option	SSFRAC	suspended solids fraction
	TEMPI	can now be used with explicit initialization in non-thermal runs

SUMMARY section

	(F,R)PP(O,G)	Now weighted by phase saturation rather than hydrocarbon volume. To revert to the pre-2004A state, set item 86 of the keyword OPTIONS3 to 1.

SCHEDULE section

Thermal option	AIMCON	Two new items have been added to allow the AIM state to depend on temperature change.
Thermal option	AIMCON	A new way of handling wells has been added, controlled by Item 1.
Thermal option	AIMCON	New default values have been set for Thermal runs.
	CHANDIMS	Two new items appended to reflect the changes to WELLDIMS.
	PICOND	Forchheimer effects may now be included in the generalized pseudo-pressure calculation by entering YES in Item 4.
Thermal option	WCONINJE	New item (11) to specify the volume of gas per volume of cold water. It should only be applied in the case of a steam-gas injector.
	VFPPROD	This keyword has been extended to include Tubing Head Temperature value in the VFP table. If the VFP table carries the tubing head temperature value, Item 9 of record 1 should read 'TEMP'.
Multi-segment wells	WSEGDFPA	New item 11 defining the exponent in the formula for oil-water drift velocity in the Drift Flux model for Multi-segment Wells.
	TSTEP	Up to 1000 entries are now allowed.
	WELSPECL	A PVT region number can be set for a well using item 12 of this keyword.
	WELSPECL	New item 17 to specify well model: Standard or high mobility injection wells.
	WELSPECS	The generalized pseudo-pressure (GPP) option for well inflow calculations may now be enabled for individual wells by entering GPP in item 8.
	WELSPECS	A PVT region number can be set for a well using item 11 of this keyword.
	WELSPECS	New item 16 to specify well model: Standard or high mobility injection wells.
Multi-segment wells and Thermal	WSEGEXSS	Keyword WSEGEXSS can now be used to inject steam in a multi-segment well at a specified quality if either water import at a specified external pressure or water import at a specified rate is chosen.
Geomechanics option	STRESSBC	The keyword has been expanded to allow components of shear or normal stress and components of rock velocity to be set as boundary conditions for use with the finite element option. For more information see the 3rd argument of keyword GEOMECH.
Thermal option	HEATER	New item (7) allows a temperature-dependent heater rate to be specified.
Thermal option	HEATERL	New item (8) allows a temperature-dependent heater rate to be specified.

OPTIONS3 keyword

Thermal option	Switch 18	If set to -1 the pre-2004A saturation-based Appleyard chop will be used with the Thermal option
	Switch 78	If set > 0 defines the PVT region number for well property calculations. A value of 1 will give the pre-2004A behavior.
	Switch 79	Overrides some corrections made to the well calculation for the 2004A release, restoring compatibility with earlier releases to allow you to assess the importance of the changes.
	Switch 82	Changes the way in which Forchheimer effects are incorporated in the Generalized Pseudo-Pressure calculation for a well.
Thermal multi-segment wells	Switch 83	If set=1, a modified method is used to update thermal multi-segment well solutions. This modified method can improve convergence for difficult problems, but may be slower in more straightforward cases.
	Switch 84	Sets the number of times a single timestep can be consecutively chopped because a well has switched from rate control to pressure control and died in the same timestep.
	Switch 86	Reverts to pre-2004A calculation of average phase potentials.
	Switch 87	Reverts to pre-2004A calculation of scaled capillary pressures if both `JFUNC` and `PCW` or `PCG` keywords are present in the dataset.
	Switch 89	If set to 1, enables the Finite Difference stress calculation to run with corner-point geometry.

RPTRST keyword

Thermal option	`RATT`	ratio of actual temperature change to target temperature change
Thermal option	`SFOIL`	foamy oil saturation
Thermal option	`SFSOL`	foamy solid saturation
Solid option	`SMMULT`	solid mobility multiplier
Solid option	`SSFRAC`	suspended solids fraction
	`TREACM`	tracer dependent reaction rate multipliers

RPTSCHED keyword

Thermal option	`SFOIL`	foamy oil saturation
Thermal option	`SFSOL`	foamy solid saturation
Solid option	`SMMULT`	solid mobility multiplier
Solid option	`SSFRAC`	suspended solids fraction
	`TREACM`	tracer dependent reaction rate multipliers

Developments in 2003A

New facilities for 2003A

ECLIPSE 100

GRID

- The MULTIREG, ADDREG and EQUALREG keywords have been extended to operate on MULTNUM regions.
- The mnemonic ROCKVOL has been added to keyword RPTGRID to allow output of bulk net and gross rock volumes to the print file.

EDIT

- The functionality of the keywords MULTREGT, MULTREGD and MULTREGH has been extended so that they can also be used in the EDIT section.
- NNC connections may be re-specified using the EDITNNCR keyword. This extends the capabilities of the EDITNNC keyword which only allows the user to specify multipliers onto existing transmissibilities.

Well model

- A pattern flood option has been implemented that allows the voidage injection rate of a well to be specified in terms of the production voidage rates of a surrounding pattern of producers. This option is controlled by the WCONINJP keyword.
- Connection D-factors for a flow dependent skin factor for gas may now be calculated from an expression defined by the keyword WDFACCOR. This expression is based on a correlation for the coefficient of inertial resistance that depends on the permeability and porosity of connected grid blocks.

Output

- The CPU option of the RPTSCHED keyword can be put equal to 4 to obtain actual CPU and elapsed times in seconds.
- It is now possible to output the SUMMARY data in a form that can easily be loaded into the OFM program.

Performance

- The ZIPPY2 keyword ‘SIM’ option now also applies to the iterations used in a non-linear tracer calculation.
- Two additional tuning parameters have been added to the ZIPPY2 keyword enabling the minimum time-step and the minimum choppable time-step to be reset within the keyword.
- There are two new keywords, TIGHTEN and TIGHTENP, to make tightening of the convergence parameters in ECLIPSE BlackOil easier.

Tracers

- New arguments have been added to the TRACERS keyword to help with some difficult cases.

Petro-elastic Model

A petro-elastic model has been added to allow the calculation of synthetic petro-elastic properties such as acoustic impedance to pressure and shear waves and also Poisson's ratio.

The model is defined using the new PEDIMS, PENUM, PECOEFS, PEKTABx and PEGTABx keywords, and output is requested using new mnemonics on the RPTRST and GRADRESV keywords.

For more information on the petro-elastic model, see the "Petro-Elastic Model" chapter in the ECLIPSE Technical Description.

ECLIPSE 300

Global and multiple section

- The MINVALUE and MAXVALUE keywords are now available in ECLIPSE 300.

RUNSPEC

- The SAMG keyword can be used in ECLIPSE 300 to invoke an algebraic multigrid solver for the underlying linear systems.
- A sub domain simulation of the reservoir can be accomplished by use of the SECTOR keyword.
- The SOLVDIMS keyword is used to set dimensions for nested factorization solver for Pebi grids. This can only be used for global grids.

GRID

- The PINCHOUT keyword can now be used in ECLIPSE 300; though the use of the PINCH keyword is still recommended.
- The NNCGEN keyword allows NNCs to be generated between any two cells in the simulation.
- The MULTIREG, ADDREG, EQUALREG and COPYREG keywords have been extended to operate on MULTNUM regions.
- A CONDFLTS keyword has been added to allow the definition of conductive faults. Output from which is generated by new SUMMARY section keywords
- The mnemonic ROCKVOL has been added to keyword RPTGRID to allow output of bulk net and gross rock volumes to the print file.

Local grid refinement

- When the local grid refinement option is used, the solver directions can be set independently for each local grid by including the SOLVDIRS keyword between the CARFIN/RADFIN/RADFIN4/REFINE and ENDFIN pair. By default, the solver directions will be determined by ECLIPSE for each local grid unless set explicitly.
- The SOLVDIMS keyword is used to set dimensions for nested factorization solver for Pebi grids. This can only be used for local grids.
- The SOLVNUM keyword should be used to set a map from user to solver order for Pebi grids. This can be used for both global grids and local grids.
- Negative values can now be entered in the permeability tensor, keyword PERMXY etc.

- The IMPFILE keyword can be used to import data from the INIT file, such as porosities and permeabilities.

EDIT

- The functionality of the MULTREGT keyword has been extended so that it can also be used in the EDIT section.
- NNC connections may be re-specified using the EDITNNCR keyword. This extends the capabilities of the EDITNNC keyword which only allows the user to specify multipliers onto existing transmissibilities.

Properties and equilibration

- The End-Point scaling option can now be used with Hysteresis. Use the ENDSCALE keyword, and input the End points on a cell by cell basis with the drainage/imbibition saturation and relative permeability End-points keywords (see the "Saturation Table Scaling" chapter in the ECLIPSE Technical Description).
- It is now possible to input the non-Darcy flow coefficient on a (saturation) region by region basis. See the VDFLOWR keyword.

Well model

- Wells and Groups may now be controlled by their Calorific production rate using new control modes and targets/limits in the WCONPROD and GCONPROD keywords. Molar calorific values must be specified with the CALVAL keyword before this new control mode may be used.
- New triggering conditions, based on vapor, liquid and total hydrocarbon component mole fractions, have been added to the ACTIONW keyword.
- A pattern flood option has been implemented that allows the voidage injection rate of a well to be specified in terms of the production voidage rates of a surrounding pattern of producers. This option is controlled by the WCONINJP keyword.
- The WINJMULT keyword may now be used to provide multipliers to the injectivity index of injection wells that vary with the well's BHP or the pressure in the wellbore adjacent to the individual connections.
- The WVFPEXP keyword may now be used to overcome certain problems with wells under THP control. It can instruct a well to use explicit water and gas fractions for VFP table look-up, or when using stabilized VFP tables, to be shut-in at the end of the timestep when it is operating in a part of the table that corresponds to the unstable region. The keyword can also control the reporting of actions taken to stop a well from hunting between control modes when operating in the unstable region of a VFP table.
- The time taken to workover a well may now be set using the WORKLIM keyword. If item 3 of the WDRILTIM keyword is set to 'YES', the well will be temporarily closed during a workover.
- The rate at which workovers are performed and new wells are drilled may now be limited according to the availability of workover and drilling rigs. These rigs are assigned to groups using the keyword GRUPRIG, with each group having up to 5 workover rigs and up to 5 drilling rigs.
- A new item has been added to the COMPKRI (and COMPKRIL) keyword that allows the user to restrict the injection mobility calculation to use only the relative permeability of the injected phase; the permeabilities of the remaining phases being set to zero in this calculation.

- Connection D-factors for a flow dependent skin factor for gas may now be calculated from an expression defined by the keyword WDFACCOR. This expression is based on a correlation for the coefficient of inertial resistance that depends on the permeability and porosity of connected grid blocks.
- Wells may now be drilled from the drilling queue when a group's production rate potential falls below a value specified by the GDRILPOT keyword.

Output and restarts

- The output request 'BASIC=6' may now be used as an argument of RPTRST in ECLIPSE 300.
- New SUMMARY section keywords have been added for increased ECLIPSE 100/300 compatibility.
- It is now possible to output the CO2 aqueous concentration to the print and restart files by using the RSW mnemonic in the RPTSOL, RPTSCHED and RPTRST keywords.
- The set of WPI(1,4,5,9) summary mnemonics are now available. These calculate the productivity/injectivity index of a well based on the flow of its preferred phase, the well BHP and the block pressure average quantities WBP(1,4,5,9).
- The wet gas production and injection rates and totals for component flows at field, group and well levels may now be reported using the summary mnemonics (F,G,W)CWG(P,I)(R,T).
- Output of NNC data to the print file using mnemonic 'ALLNNC' with keyword RPTGRID has been extended to work in parallel runs.
- NNC flow data will automatically be output when flow data is output with the RPTSCHED keyword. For example, the mnemonic 'FLOWAT' will now give both the neighbor water flows and the NNC water flows. (This is equivalent to the ECLIPSE 100 request 'FLOWAT=3').

Local grid refinement

- By default, all solution data requested under the RPTSOL keyword will now be output for any local grid refinements present. This local grid output can be switched off by setting item 75 of OPTIONS3 to 1.
- It is now possible to output information to the restart file providing details of cells which are causing convergence problems. At every non-linear iteration, and for the volume balance residual equation as well as the pressure update, the worst cells will have their counter increased by one. This can be done using the mnemonic 'CONV=integer' with keyword RPTRST. By default, this 'integer' value is set to 10, so that the worst 10 cells in the run will be focused on. Modifying this 'integer' value will alter the number of worst cells.

Performance

- The CFLLIMIT keyword has now been extended to work with the black oil model.

Geomechanics option

- The Geomechanics option now has the ability to handle thermal expansion. The keyword GLTHEX allows you to set a linear coefficient, which modifies the elastic stresses. This keyword is only valid if the THERMAL keyword is specified in the RUNSPEC section.
- An alternative solver may be used. If partly coupled, then specifying the keyword SAMG in the RUNSPEC section allows an algebraic multigrid solver to be used for solving the partly coupled stress equations.

Gradient option

- Gradients of well quantities with respect to multipliers on permeabilities and transmissibilities may now be requested. These are also compatible with the Parallel option. The keywords associated with this are the history matching multiplier keywords HMMULTxx (for example, use HMMULTX to specify history matching transmissibility multipliers in the x-direction), and the history matching region keywords HMxxxxxx (for example use HMTRANX to specify a region where gradients with respect to the x-direction transmissibility are to be calculated.). See "The Gradient Option" chapter in the ECLIPSE Technical Description for details of the usage of this option.

Solid option

- Keyword SOLID can be used to indicate a solid phase in the run. This can either be used with an equation of state or with the Thermal option.

  The SOLID keyword together with the HYDRO keyword can be used with an equation of state to predict asphaltene precipitation. Summary vectors BBSOL, BDENS (or BSDEN) and BSSOLID (or BSSAT) can be used to output block molar densities, densities and solid saturations. Mnemonics BSOL, DENS and SSOLID can be used with keywords RPTRST, RPTSCHED and RPTSOL to output solid molar densities, densities and saturations.

Thermal option

- The SOLID keyword can also be used with the Thermal option - for use with chemical reactions. Components can be defined as a solid by using keyword CVTYPE. Solid phase densities can be specified using keywords SCREF, SDREF, SPREF, STREF and STHERMX1. Solid phase component specific heats can be modeled with the SPECHS and SPECHT keywords. Summary vector BHSOL can be used to output block enthalpies. Mnemonic HSOL can be used with keywords RPTRST, RPTSCHED and RPTSOL to output solid enthalpies.

Thermal option

- The water formation volume factor defined by the PVTW keyword can now be used in thermal simulations. The data is used when the water density temperature dependence is defined by the new keyword WATDENT.
- The water viscosity, as a function of temperature, can now be defined by the WATVISCT keyword.
- The water "viscosibility" defined by the PVTW keyword can now be used in thermal simulations. The data is used when the VISCREF keyword is used.
- The oil phase component specific heat can now be modeled with temperature dependent term given by the SPECHB keyword.
- The gas phase component specific heat can now be modeled with temperature dependent term given by the SPECHH keyword.
- The oil viscosity can now be defined as a non-linear function of the composition using the keyword OILVINDX.
- New summary keywords FHTR, FHTT, WHTR and WHTT can be used for field or well heater rates and totals.
- A maximum cell temperature can be set for well heaters, HEATER, HEATERL.

Tracer option

- An alternative method for calculating the production from above and below the GOC in the TRACK option has been implemented. The method is based on directly splitting a well's volumetric flow rate according to the tracer concentrations above and below the contact. See "Alternative production reporting" in the Tracer Tracking chapter in the ECLIPSE Technical Description for more details.

Behavioral changes for 2003A

These may cause changes to the results of a simulation, or alter the form or content of the output.

ECLIPSE 100

Well model

- In the WFRICSEG keyword, the length of each calculated segment is set to the grid cell thickness, DZ, in the case when a range of connected grid cells is specified in the z direction. Prior to this release the segment lengths were set to DZ*NTG in this case.
- The keyword WRFTPLT will now override the action of any previously read WRFT keywords.

Properties

- The default behavior of the 8th data item in the EHYSTR keyword has been changed to OIL, treating the oil in the oil-to-gas relative permeability as the wetting phase.

Output and restarts

- The EPSDEBUG output has been extended to output the SOM (residual oil) values in cases where the STONE1 three phase rel-perm model is used in conjunction with either SOMGAS or SOMWAT.

ECLIPSE 300

Properties

- If neither the BLACKOIL or the COMPS keyword are present ECLIPSE 300 now assumes the run to be a BLACKOIL run. A warning is issued.
- Default values for keyword DREF have been made consistent for all unit conventions. The default values for keywords TREF and PREF have been set to the values in STCOND. These changes only affect thermal runs, or those using the Zudkevitch Joffe equation of state.
- For MISCIBLE runs, the average surface tension is calculated using the Macleod-Sugden correlation. For single phase runs, the value calculated is zero. In order to re-specify the surface tension data explicitly, use the MISCSTR keyword in the PROPS section.
- The treatment of combined hysteresis and miscible flooding for calculating relative permeabilities has been improved. The critical saturation for the immiscible part is now based on a scanning curve if relevant, rather than the drainage curve. Also the critical point of the miscible curve is not assumed to be zero if the MISCNUM keyword is present.
- The default behavior of the 8th data item in the EHYSTR keyword has been changed to OIL, treating the oil in the oil-to-gas relative permeability as the wetting phase.

Well model

- An export rate for oil (and, if appropriate, NGL) will now be calculated and reported if gas from an intermediate separator stage is both used as a feed for a subsequent separator stage and also acts as a source of re-injection gas. The re-injection from such an intermediate separator stage will reduce the feed to the subsequent stage and the separator oil rate will thus be reduced. This reduced oil rate is reported as the export rate and should be distinguished from the produced oil rate, reported elsewhere in the print file, which is the 'formation' oil rate prior to any re-injection work.

  These export rates and totals are reported in the print file (see section "Export tables" in the Well Model chapter in the ECLIPSE Technical Description for more details) and may also be written to the summary file using the summary section mnemonics (F,G)E(O,G,N)(R,T).

- Item 3 in WDRILTIM, if set YES, will now result in wells being effectively shut during the time taken to drill the well. The well is effectively shut by setting its efficiency factor to 1.0e-9 until the timestep during which the workover over will be finished. Prior to this release the well was fully shut rather than effectively shut during these timesteps.

Input/Output

- TOTCOMP data, output using either RPTSCHED or RPTRST has been modified (by a factor of -1).

CO2SOL option

- The equilibration of CO2SOL cases has been modified to use the water properties defined in the SOLUBILI table (which may be defaulted) rather than the PVTW data.

Thermal option

- The KVCR keyword has been simplified. Up to five coefficients can be specified for each component (instead of nine).
- The water "viscosibility" defined by the PVTW keyword was ignored in thermal simulations. However, it will now be used if the VISCREF keyword is also used.
- Tables of K-values, densities, enthalpies and viscosities at different temperatures are now output to the print file.
- For reservoir heat loss, the number of connections between the reservoir and the overburden / underburden is now calculated by the simulator, and does not need to be specified by the ROCKDIMS keyword. The connection data is now output to the flexible restart files in a modified order. These changes have been introduced to improve the compatibility between serial and parallel runs.
- The temperature based Appleyard chop is now switched on by default. It can be switched off by setting item 27 of OPTIONS3 to 1.

New keywords for 2003A

ECLIPSE 100

RUNSPEC section

PEDIMS	Used to set dimensions for a petro-elastic model.

EDIT section

EDITNNCR	Specifies NNC connections to replace those already calculated.

PROPS section

PECOEFS	Specifies petro-elastic fluid and rock properties, and other petro-elastic model options.
PEKTABx	Specifies petro-elastic frame bulk modulus functions.
PEGTABx	Specifies petro-elastic frame shear modulus functions.

Regions section

PENUM	Specifies the petro-elastic region number for each grid cell in the simulation model.

SUMMARY section

CDFAC	Connection D-factors for gas flow dependent skin factor.
OFM	Requests output of SUMMARY data in OFM format.

SCHEDULE section

TIGHTEN	Enables convergence parameters to be tightened
TIGHTENP	Enables convergence parameters to be tightened
WCONINJP	Sets the rate of an injection well in terms of the voidage of the production wells that surround it.
WDFACCOR	Calculates D-factors for the flow dependent skin factor for gas.

ECLIPSE 300

GLOBAL and multiple section keywords

MAXVALUE	Applies a maximum value to specified arrays within the current box.
MINVALUE	Applies a minimum value to specified arrays within the current box.

RUNSPEC section

SAMG	Specifies that linear systems should be solved using algebraic multigrid.
SECTBC	Specify boundary conditions for a sector model.
SECTOR	Specify a sector of the full field model.

	SOLID	Indicates a solid phase in the run
	SOLVDIMS	For a global unstructured grid, this keyword instructs ECLIPSE that the solver operates on a given cartesian box.

GRID section

Gradient option	HMMULTxx	History matching multipliers on transmissibility (HMMULTX, HMMULTY, HMMULTXY, HMMULTZ) and on permeability (HMMLTPX, HMMLTPY, HMMLTPXY, HMMLTPZ) added.
	PINCHOUT	Added to ECLIPSE 300 for compatibility with ECLIPSE 100. The PINCH keyword is still to be preferred.
	NNCGEN	Generalized input of NNC.
	MINVALUE/MAXVALUE	New array operators (ECLIPSE 100 compatibility).
	CONDFLTS	Conductive faults.
	IMPFILE	Import data from INIT file.
Local grid refinement	SOLVDIRS	This keyword overrides the default directions for the linear solver, for each local grid refinement.
Local grid refinement	SOLVDIMS	This keyword should be specified between the CARFIN or EXTFIN and ENDFIN keywords which define the local grid. It then instructs ECLIPSE that the solver operates on a given cartesian box.
	SOLVNUM	This keyword is used to set a map from user to solver order for Pebi grids. The SOLVDIMS keyword should also be specified to inform ECLIPSE of the solver dimensions used when mapping to a cartesian structured grid for the linear solver.

EDIT section

Gradient option	HMMULTxx	History matching multipliers on transmissibility (HMMULTX, HMMULTY, HMMULTXY, HMMULTZ) and on permeability (HMMLTPX, HMMLTPY, HMMLTPXY, HMMLTPZ) added.
	MINVALUE/MAXVALUE	New array operators (ECLIPSE 100 compatibility).
	MULTREGT	Used to set a transmissibility multiplier between regions.
	EDITNNCR	Specifies NNC connections to replace those already calculated.

PROPS section

	HYDRO	Defines hydrocarbon type, for use with the SOLID option.
	MINVALUE/MAXVALUE	New array operators (ECLIPSE 100 compatibility).
Thermal option	OILVINDX	Allows the oil viscosity to be defined as a non-linear function of the composition.
Thermal option	SCREF	Component solid phase compressibility.
Thermal option	SDREF	Component solid phase reference density.
Thermal option	SPREF	Component solid phase reference pressure.

Thermal option	STREF	Component solid phase reference temperature.
Thermal option	STHERMX1	Component solid phase thermal expansion coefficient.
Thermal option	SPECHB	Component oil specific heat, 2nd coefficient.
Thermal option	SPECHH	Component gas specific heat, 2nd coefficient.
Thermal option	SPECHS	Component solid specific heat, 1st coefficient.
Thermal option	SPECHT	Component solid specific heat, 2nd coefficient.
Thermal option	WATDENT	This keyword can be used in thermal simulations to define the water density temperature dependence.
Thermal option	WATVISCT	This ECLIPSE 100 keyword can now be used in ECLIPSE 300.
	VDFLOWR	Used to input the non-Darcy flow coefficient on a (saturation) region-by-region basis.

REGIONS section

Gradient option	HMMULTxx	History matching region identifiers, HMTRANX, HMTRANY, HMTRANXY and HMTRANZ added for transmissibility gradient calculations. HMPERMX, HMPERMY, HMPERMXY and HMPERMZ added for permeability gradient calculations.

SUMMARY section

(F,G)E(O,G,N)(R,T)

Field and group export oil, gas and NGL rates and totals.

(F,G,W)CVP(R,T)

Field, group and well calorific production rates and totals.

(F,G,W)CVPP

Field, group and well calorific production potential rates.

(F,G,W)(O,G)P(R,T)(1,2)

Alternative field, group and well, oil and gas production rate and totals from fluids above and below the GOC in the TRACK option. See “Alternative production reporting” in the “Tracer Tracking” chapter in the ECLIPSE Technical Description for more details.

(F,G,W)CWG(P,I)(R,T)

Component wet gas production and injection rates and totals.

Thermal option (F,W)HT(R,T)

Field and well heater rates and totals.

WPI(1,4,5,9)	Productivity indices based on block pressure averages.
CDFAC	Connection D-factors for gas flow dependent skin factor.

CF(O,W,G)SAT

Oil, water and gas saturations in a conductive fault

CFPRES	Pressure in a conductive fault
CFOWC	Oil-water contact depth in a conductive fault
CFGOC	Gas-oil contact depth in a conductive fault

	`BBSOL`	Solid molar densities
	`BDENS` (or `BSDEN`)	Solid densities
Thermal option	`BHSOL`	Solid enthalpies.
	`BSSOLID` (or `BSSAT`)	Solid saturations

SCHEDULE section

GDRILPOT	Causes wells to be drilled from the drilling queue when a group's production rate potential falls below a specified value.
GRUPRIG	Assigns workover and drilling rigs to groups.
WCONINJP	Sets the rate of an injection well in terms of the voidage of the production wells that surround it.
WDFACCOR	Calculates D-factors for the flow dependent skin factor for gas.
WINJMULT	Specifies pressure-dependent injectivity multipliers.
WORKLIM	Time taken for each automatic workover.
WVFPEXP	Provides options to overcome some problems with THP controlled wells.

Altered keywords for 2003A

ECLIPSE 100

RUNSPEC section

TRACERS	Five arguments have been added to control convergence of the simulation when problems have been detected in the tracer solution.

GRID section

MULTIREG	This keyword has been extended to operate on MULTNUM regions.
ADDREG	This keyword has been extended to operate on MULTNUM regions.
EQUALREG	This keyword has been extended to operate on MULTNUM regions.
RPTGRID	This keyword has been extended to output rock volumes, with the ROCKVOL mnemonic.

PROPS section

EHYSTR	The default behavior of item 8 has been changed to OIL.

SOLUTION section

RPTRST	New mnemonics (ACIP, ACIS and POIS) have been added to request output from the petro-elastic model
GRADRESV	New mnemonics (ACIP, ACIS and POIS) have been added to request output from the petro-elastic model

SCHEDULE section

ZIPPY2	Two new options MINSTEP and MINCHOP have been added to set the minimum time-step and the minimum choppable time-step.
RPTSCHED	The CPU mnemonic can be set to 4 to obtain cpu and elapsed times in seconds.
RPTRST	New mnemonics (ACIP, ACIS and POIS) have been added to request output from the petro-elastic model

ECLIPSE 300

RUNSPEC section

Thermal option	ROCKDIMS	Item 2 is no longer used and can be defaulted.

GRID section

	MULTIREG	This keyword has been extended to operate on MULTNUM regions.
	COPYREG	This keyword has been extended to operate on MULTNUM regions.
	ADDREG	This keyword has been extended to operate on MULTNUM regions.
	RPTGRID	This keyword has been extended to output rock volumes with the ROCKVOL mnemonic.
	EQUALREG	This keyword has been extended to operate on MULTNUM regions.
	GDFILE	Can now read an EGRID file. By default the keyword looks for the EGRID file before the GRID file.
	PERMXY	Negative values can now be entered in the permeability tensor, using this and related keywords.
	FAULTS	Can now be specified to be compatible with the Geomechanics option. This allows a balance of normal forces and shear limiting behavior.

EDIT section

	MULTREGT	Now also used to set a transmissibility multiplier between regions.
Molecular Diffusion option	MULTREGD	Now also used to set a transmissibility multiplier between regions.
Temperature option	MULTREGH	Now also used to set a transmissibility multiplier between regions.

PROPS section

Thermal option	CVTYPE	This keyword can now be used to define solid phase components, for use with chemical reactions.
	DREF	Default values have been made consistent for all unit conventions.
	EHYSTR	The default behavior of item 8 has been changed to OIL.
Thermal option	KVCR	This keyword has been simplified. Up to five coefficients can be specified for each component.
Thermal option	PREF	The default values have been set to the standard pressure in STCOND. (Previously these default values were only used if the keyword was not used).
	SOLUBILI	The water properties from SOLUBILI (or the default table) are used in the initial equilibrium for CO2SOL runs.
	TREF	The default values have been set to the standard temperature in STCOND. (Previously these default values were only used if the keyword was not used).

SOLUTION section

Thermal option	TEMPI	This keyword, which is used to specify the initial temperature values for each cell, can now be used with the EQUALS keyword.
	RPTSOL	The RSW mnemonic is now available to output aqueous CO_2 concentration.

Local grid refinement	RPTSOL	By default, all solution data requested under this keyword will now be output for any local grid refinements present.
	RPTSOL	Mnemonics BSOL, DENS and SSOLID can be used to output solid molar densities, densities and saturations.
Thermal option	RPTSOL	Mnemonic HSOL can be used to output solid enthalpies.
	RPTRST	The RSW mnemonic is now available to output aqueous CO_2 concentration.
	RPTRST	Mnemonics BSOL, SSOLID, DENS can be used to output solid molar densities, solid saturations, density.
Thermal option	RPTRST	Mnemonic HSOL can be used to output solid enthalpies.
	RPTRST	TOTCOMP data has been modified (by a factor of -1).
	RPTRST	The mnemonic BASIC=6 may be used to request restart file generation at every timestep.

SCHEDULE section

	ACTIONW	New mole fraction based triggers have been implemented.
	COMPKRI	New item to effectively zero the relative permeabilities of the non-injection phases in the injection mobility calculation. Negative relative permeability values will now leave a previously defined value unaltered.
	COMPKRIL	New item to effectively zero the relative permeabilities of the non-injection phases in the injection mobility calculation. Negative relative permeability values will now leave a previously defined value unaltered.
	CVCRIT	New arguments added to control behavior of algebraic multigrid linear solver in combination with the SAMG keyword.
	GCONPROD	New calorific rate control mode and target/limit.
	GRUPTARG	A group calorific rate target may now be set.
	GTMULT	A group calorific rate target may now be multiplied.
	WCONPROD	New calorific rate control mode and target/limit.
	WELTARG	A well calorific rate target may now be set.
	WGRUPCON	A well calorific guide rate may now be set.
	WTMULT	A well calorific rate target may now be multiplied.
Thermal option	HEATER	A maximum cell temperature can be set for well heaters.
Thermal option	HEATERL	A maximum cell temperature, for cells in LGRs, can be set for well heaters.
	RPTSCHED	The RSW mnemonic is now available to output aqueous CO_2 concentration.
	RPTSCHED	Mnemonics BSOL, DENS and SSOLID can be used to output solid molar densities, densities and saturations.
Thermal option	RPTSCHED	Mnemonic HSOL can be used to output solid enthalpies.
	RPTSCHED	TOTCOMP data has been modified (by a factor of -1).
	RPTRST	The RSW mnemonic is now available to output aqueous CO_2 concentration.
	RPTRST	Mnemonics BSOL, DENS and SSOLID can be used to output solid molar densities, densities and saturations.
Thermal option	RPTRST	Mnemonic HSOL can be used to output solid enthalpies.
	RPTRST	TOTCOMP data has been modified (by a factor of -1).

	RPTRST	The mnemonic BASIC=6 may be used to request restart file generation at every timestep.
	RPTRST	The mnemonic ‘CONV’ can be used to output details of cells which are causing convergence problems.

OPTIONS3 keyword

Thermal option	Switch 27	The temperature-based Appleyard chop is now switched on by default. It is switched off by setting this item > 0.
	Switch 74	If set > 0, a more rigorous saturation pressure is performed for output of PSAT values. This calculation can be significantly slower than the default calculation, but gives more accurate answers for fluids near the critical point.
Local grid refinement	Switch 75	If set > 0, the local grid initial solution output, requested under the RPTSOL keyword, will be switched off. This output will apply to all local grids defined in a run.
	Switch 76	If set >0, reverts to the 2002A treatment of combined hysteresis and miscible flooding.

Developments in 2002A

New facilities for 2002A

ECLIPSE 100

Global

- To improve the portability of data files between the simulators, a family of keywords has been added that instruct ECLIPSE to skip over particular sequences of keywords. The instructions may be conditional on which simulator is being used. These keywords are SKIP, SKIP100, SKIP300 and ENDSKIP.

Properties and equilibration

- The TZONE option to set critical saturation is now available in runs where the Mobile Fluid Correction option in not active (see keywords ENDSCALE and EQLOPTS in the RUNSPEC section).
- A new option has been added to the water induced compaction model to allow partial reversibility of the compaction. Compaction can be reversed up to the pore volume equivalent to the minimum pressure and initial water saturation. See the ROCKCOMP keyword in the RUNSPEC section.
- It is now possible to define oil as wetting to gas in the three-phase case, in a water wet system. See the eighth option of the EHYSTR keyword in the PROPS section.

Well model

- Pressure-dependent injectivity multipliers may be specified separately for individual well connections. There is also an option to make these irreversible, so that the injectivity increase is not reversed if the well pressure subsequently declines. See keyword WINJMULT.
- A simple command is now available to reopen “priority-closed” wells, which are temporarily closed under Prioritization control. Setting item 2 of keyword WELOPEN (or item 3 of keyword WELOPENL) to POPN opens the well only if it is priority-closed, and leaves it closed if it has been closed for any other reason. It could be useful to do this when changing the group control method from Prioritization to Guide Rate control.
- You may choose how the connections are ordered in each well (see keyword COMPORD): by depth, by order of input, or by position along the well track as determined by ECLIPSE. Previously this choice had to be made for all wells together, by setting OPTIONS switch 53. Also, the closure of additional connections in +CON workovers is now governed by their position along the well track, although it is possible to revert to the former treatment (purely by depth) by setting OPTIONS switch 53.
- During the history matching process, injection wells may now be treated in the same manner as production wells by defining them as history matching injectors with keyword WCONINJH. The measured injection rate, BHP and/or THP are supplied, and the measured pressures can be output to the Summary file to compare with the calculated pressures.
- In the ACTIONW keyword a trigger on the well bottom hole gas liquid ratio (WBGLR) has been added.

- The NOSIM keyword may now be specified within the SCHEDULE section to turn off time step simulation after a point in time within the simulation.
- The SIMULATE keyword may be specified within the SCHEDULE section to counteract a NOSIM keyword in this section and turn on time step simulation. Note that the SIMULATE keyword cannot counteract a RUNSPEC section NOSIM or a NOSIM in item 3 of the LOAD keyword during a restart.

Output

- The reservoir fluid volume (RESV) rates at the well connections, and their cumulative totals, are now included in the Well Reports that are written to the Print file under control of the WELLS mnemonic in keyword RPTSCHED. Note that the sum of the connection RESV rates does not exactly equal the well's RESV rate if the connections have differing GORs or produce fluid from different PVT regions. RESV rates are now also included in the Layer Totals.
- The reservoir fluid volume rates at the well connections, and their cumulative production and injection totals, may be written to the Summary file with the mnemonics CVFR, CVPT and CVIT. The corresponding quantities for lumped completions may be written with mnemonics CVFRL, CVPTL, CVITL.
- The saturation values in the output files may now be restricted to lie between 0.0 and 1.0 by setting switch 83 of the OPTIONS keyword.
- It is now possible to output the sinks associated with drainage regions to the Restart file; see RPTRST keyword.
- It is now possible to output the transmissibility multipliers applied by the rock compaction option to the Summary file, with mnemonics FRTM, RRTM and BRTM in the SUMMARY section.
- Unified and multiple summary files may be requested with the UNIFOUTS and MULTOUTS keywords. These keywords can be used in conjunction with the UNIFOUT / MULTOUT keywords to give better control over the output.
- New SUMMARY section keywords have been added for increased ECLIPSE 100/300 compatibility.

Performance

- An experimental feature has been added, keyword ZIPPY2, which can dramatically reduce run times in some cases, by attempting to choose the time step more intelligently.

Flux boundary conditions

- It is now possible to advance a USEFLUX run beyond the end time of the original DUMPFLUX case, with the flux or solution extrapolated from the last timestep of the DUMPFLUX case. See the USEFLUX keyword.

Gradient option

- It is now possible to obtain permeability gradients for a radial run.
- For SimOpt compatibility, NOHMO is an alias for NOHMD keyword.

Multi-segment wells

- A pullthrough pump device model is available for controlling the water offtake flow in a down-hole water separator. It can control the pressure differential to achieve optimum separation, so that the flow along the water outlet branch matches the water inflow into the separator, while keeping the amount of oil carry-over to acceptable levels. The working power of the pump can be output to the Summary file using the SUMMARY section mnemonic SPPOW. See keyword WSEGPULL.
- More parameters of the Drift Flux slip model are available for you to set. Separate profile parameters and drift velocity multipliers can be entered for gas-liquid slip and oil-water slip. Keyword WSEGDFPA alters the default values of the parameters for all well segments, while some of these parameters can still be modified for individual segments with WSEGFMOD. The profile parameter and drift velocity for gas-liquid slip (SGLPP, SGLVD) and oil-water slip (SOWPP, SOWVD) can be written to the Summary file.
- The mixture density in a segment can be output to the Summary file using the SUMMARY section mnemonic SDENM.

ECLIPSE 300

Global

- To improve the portability of data files between the simulators, a family of keywords has been added that instruct ECLIPSE to skip over particular sequences of keywords. The instructions may be conditional on which simulator is being used. These keywords are SKIP, SKIP100, SKIP300 and ENDSKIP.

Properties and equilibration

- Chemical reactions can now be modeled in compositional (EoS and K-value) and thermal simulations; see the Chemical Reactions chapter of the ECLIPSE Technical Description. The following keywords have been added to describe the reactions: REACTION, STOPROD, STOREAC, REACRATE, REACACT, REACPORD, REACCORD and REACPHA. In Thermal runs the REACENTH keyword can also be used.
- The viscosibility (sixth column of the PVCO keyword) is now recognized by ECLIPSE 300. It is possible to ignore the viscosibility (the pre-2002A behavior) by setting switch 24 in OPTIONS3 to -1.
- The TZONE option to set critical saturation is now available in runs where the Mobile Fluid Correction option in not active (see keywords ENDSCALE and EQLOPTS in the RUNSPEC section).
- An alternative interpolation scheme has been implemented for black oil PVT tables. This new interpolation is closer to the method used in ECLIPSE 100. See switch 56 in the OPTIONS3 keyword.
- A new option, NOHCSCAL has been added to the FORMOPTS keyword. This is because HCSCAL has now become the default setting.
- It is now possible to define oil as wetting to gas in the three-phase case, in a water wet system. See the eighth option of the EHYSTR keyword in the PROPS section.
- The Integrated capillary pressure option can be activated in dual porosity runs, using the INTPC keyword.

Well model

- You may choose how the connections are ordered in each well (see keyword COMPORD): by depth, by order of input, or by position along the well track as determined by ECLIPSE. Also, the closure of additional connections in +CON workovers is now governed by their position along the well track, although it is possible to revert to the former treatment (purely by depth) with this keyword.
- The SIMULATE keyword may be specified within the SCHEDULE section to counteract a NOSIM keyword in this section and turn on time step simulation.
- The SKIPREST keyword may be used to instructs ECLIPSE to skip keywords in the SCHEDULE section of a restart run until the restart time has been reached. At this point, the timestepping simulation will begin. This saves you from having to delete the keywords yourself.
- A simple command is now available to reopen "priority-closed" wells, which are temporarily closed under Prioritization control (see "Prioritization - an alternative to guide rate control" in the ECLIPSE Technical Description). Setting item 2 of keyword WELOPEN (or item 3 of keyword WELOPENL) to POPN opens the well only if it is priority-closed, and leaves it closed if it has been closed for any other reason. It could be useful to do this when changing the group control method from Prioritization to Guide Rate control.
- During the history matching process, injection wells may now be treated in the same manner as production wells by defining them as history matching injectors with keyword WCONINJH. The measured injection rate, BHP and/or THP are supplied, and the measured pressures can be output to the Summary file to compare with the calculated pressures.

Output and restarts

- When the multi-point flux option MPFA is used, cell reverse transmissibilities and k-orthogonality errors can be output using the RPTGRID keyword with mnemonics MPFTREV and MPFORTH.
- It is now possible to output the processed end-point values from the SWATINIT and TZONE options to both the Print and Restart files. See the RPTSOL, OUTSOL and RPTRST keywords.
- The Rs and Rv mnemonics in the RPTRST/ OUTSOL keyword are now accepted in compositional runs.
- Fluid in place for each component can be output with the RPTRST/ OUTSOL keyword using the mnemonic FIP.
- BASIC and FREQ mnemonics have been added to the RPTRST/ OUTSOL keyword.
- The reaction rates for chemical reactions can be output for each grid cell using the mnemonic REAC in the RPTSCHED and RPTRST/ OUTSOL keywords or as Summary vectors using the BREAC, FREAC or FREACT mnemonics.
- The FILLEPS keyword has been added for output of end point data to the INIT file (for compatibility with ECLIPSE 100).
- The WRFTPLT keyword is now available in ECLIPSE 300, allowing output of well data to the RFT file. This file contains data for one or more wells describing fluid conditions in the wellbore or the connecting grid blocks at selected times in the run. RFT, PLT and, for multi-segment wells, segmentation data may be written to this file at the current simulation time, at report times and in the case of RFT data when a well is first opened.

- A table has been added near the end of the Print file that shows the number of wells in the model, the maximum number of connections in any one well, etc. for this run. Also shown are the corresponding WELLDIMS values. By running a model in NOSIM mode, this table should help you to realistically dimension the well model without having to work out these dimensions manually.
- The reservoir fluid volume (RESV) rates at the well connections are now included within the 'All Connections' table in the Print file. Note that the sum of the connection RESV rates will not exactly equal the well's RESV rate if the connections have differing GORs or produce fluid from different PVT regions.
- The reservoir fluid volume rates at the well connections, and their cumulative production and injection totals, may be written to the Summary file with the mnemonics CVFR, CVPT and CVIT. The corresponding quantities for lumped completions may be written with mnemonics CVFRL, CVPTL, CVITL.
- For connection type SUMMARY section mnemonics (starting with a 'C', or 'LC' for a well completed in a LGR), the three grid block coordinates that define a connection may now be defaulted for a well, which causes the appropriate quantity to be output for **all** connections in the well.
- Unified and multiple summary files may be requested with the UNIFOUTS and MULTOUTS keywords. These keywords can be used in conjunction with the UNIFOUT / MULTOUT keywords to give better control over the output.
- New SUMMARY section keywords have been added for increased ECLIPSE 100/300 compatibility.

Performance

- More options have been added to the time truncation error timestep control. See switch 58 in the OPTIONS3 keyword.
- The SKIPSTAB keyword can be used in compositional EoS to speed up flash calculations.
- When using IMPES or AIM it is now possible to select time step choice from stability requirements with the CFLLIMIT keyword.
- An alternative Black Oil formulation has been implemented with the aim of improving the performance in fully implicit cases. The new formulation is similar to that used in ECLIPSE 100 for the reservoir equations with the well model using the standard ECLIPSE 300 approach. The option is initiated by using the BOILMOD switch in the FORMOPTS keyword. At present the option can only be used in fully implicit cases and is not available for gas condensate cases (PVTG keyword).

Flux boundary conditions

- It is now possible to advance a USEFLUX run beyond the end time of the original DUMPFLUX case, with the flux or solution extrapolated from the last timestep of the DUMPFLUX case. See the USEFLUX keyword.

Gas field operations model

This ECLIPSE 100 feature is now available as a Special Extension to ECLIPSE 300. It contains a set of facilities designed to model gas field production constraints and operations. See the Gas Field Operations Model chapter of the ECLIPSE Technical Description for further details.

Geomechanics option

Rock stress equations can be solved fully or partly coupled with the other fluid flow equations (see the "Geomechanics" chapter of the ECLIPSE Technical Description). This option is chosen with the keyword GEOMECH in the RUNSPEC section. Arguments of this keyword specify whether the simulation is fully or partly coupled, and whether the grid is fixed or moving.

Rock mechanical properties are entered in the GRID section. Keywords YOUNGMOD, POISSONR, ROCKDEN and BIOTC set the Young's modulus, Poisson ratio, rock density and Biot constant for each grid block. Permeability and/or Biot constant may be set to be a tabular function of maximum and minimum principal stress. These are entered with the keywords PERMSTAB and BIOTCTAB in the PROPS section. Separate geomechanical regions may be set with the keyword YLDNUM in the REGIONS section. This allows, for example, different stress dependent permeability or Biot constant tables in different regions of the grid. Stress boundary conditions for all external and internal boundaries may be set with the keyword STRESSBC in the SCHEDULE section.

Gradient option

This feature is now available as a Special Extension to ECLIPSE 300 (see "The Gradient Option" chapter of the ECLIPSE Technical Description). It contains a set of keywords designed to compute gradients of the solution (for example, pressure in a well) with respect to a single property parameters defined by the user.

The SimOpt program uses the gradients computed by the simulator to achieve an efficient regression during a history match, varying property parameters using modifiers.

For this release, the property parameter modifier is the pore volume multiplier input through the keyword HMMULTPV applied to region HMPORVM.

Gradients of observable well quantities (for example, WWCT, WOPR) are written to a file (extension .HMD) at each timestep and these can be used by SimOpt to match against observed data using a gradient based optimization approach.

The principal keywords in this option are:

- HMDIMS specifies the dimensions for the Gradient Option; for instance, the maximum number of gradients required, and the maximum number of gradient regions.
- FMTHMD indicates that the .HMD file is to be formatted.
- UNCODHMD indicates that the .HMD file produced by the Gradient option is to be unencoded. By default, the sensitivities contained in the .HMD file are encoded and can only be interpreted by SimOpt. If unencoded sensitivities are required for use in an external program, an unencoded gradients license feature is also required.
- HMMULTPV sets cumulative pore volume multipliers for Gradient runs. This keyword is analogous to MULTPV except that HMMULTPV acts as a multiplier on any value of MULTPV, which had been previously specified in either the GRID or EDIT section. It is cumulative.
- HMPORVM specifies region data for the Gradient Option. This keyword should be followed by an integer for every grid block in the current input box to indicate the gradient sub-region associated with the keyword to which the cell belongs. A zero indicates that the cell does not belong to any subregion, while a positive number defines the sub-region to which the cell belongs. For this gradient region keyword, ECLIPSE 300 will then compute gradients of the solution with respect to the gradient parameters within each user- defined sub-region possessing active cells.

- GRADWELL can be used to specify which well quantities are written to the .HMD file. For this release, possible output is WOPR (surface oil rate), WWPR (surface water rate), WGPR (surface gas rate), WWCT (well water cut), WGOR (well gas-oil ratio), and WBHP (well bottom hole pressure).

Local grid refinement

- DGRID will now work with an LGR.
- For compatibility, the RPTGRIDL keyword has been added to the GRID section. This keyword is used to output GRID section data for local refined grids.
- Keywords COMPAGHL, COMPKRIL and COMPMBIL can now be used to set gravity head, relative permeability and mobility in local refinement completion cells. These keywords are similar to COMPAGH, COMPKRI and COMPMOBI, which act on global cells.
- It is now possible to restart with an LGR added around a well. The keyword WELMOVEL is added for compatibility with ECLIPSE 100.

Multi-segment wells

- More parameters of the Drift Flux slip model are available for you to set. Separate profile parameters and drift velocity multipliers can be entered for gas-liquid slip and oil-water slip. Keyword WSEGDFPA alters the default values of the parameters for all well segments, while some of these parameters can still be modified for individual segments with WSEGFMOD. The profile parameter and drift velocity for gas-liquid slip (SGLPP, SGLVD) and oil-water slip (SOWPP, SOWVD) can be written to the Summary file.
- An extra item has been added to the CVCRIT keyword to control convergence of drift-flux segments in multi-segment wells.
- The mixture density in a segment can be output to the Summary file using the SUMMARY section mnemonic SDENM.

Parallel option

- The PSPLITX and PSPLITY keywords allow the user to specify all optimized partitions in the X and Y directions respectively. The data for this keyword is output from the PART mnemonic in the RPTSOL keyword. With these new keywords, after an initial run, a good load-balancing can be achieved with minimal editing of the data.

Thermal option

- The keyword CVTYPE can be used to specify non-volatile or non-condensable components in live oil runs. The keyword CVTYPES can be used to specify non-volatile or non-condensable components at surface conditions in live oil runs.
- The keyword CCTYPE can be used to over ride the default formula used to calculate the component liquid phase compressibility. The keyword CCTYPES can be used to define the compressibility type at surface conditions.
- The keyword KVCRS can be used to specify K-values for surface calculations.
- For chemical reactions the REACENTH keyword can be used to define the reaction enthalpy. Keyword EQLDREAC can be used to define reaction rates that depend on the deviation from an equilibrium value specified by EQLDKVCR.
- The YMF keyword can now be used to enter water vapor fractions for explicit initialization.
- The EQUIL keyword can bow be used to equilibrate with a gas cap in thermal simulations (item 10 equals 2 or 3).

- For thermal simulations using the new reaction keywords, the keyword REACCRIT may be used to improve stability.

Tracer option

- A new keyword, DETAILMF, has been added to the delumping option which enables the detailed initial conditions to be specified as mole fractions with respect to depth.
- A new keyword, FASTTRAC, has been added to the RUNSPEC section which instructs the tracer calculation to take some shortcuts. For simulations involving many tracers this can lead to substantial speed gains.

Behavioral changes for 2002A

These may cause changes to the results of a simulation, or alter the form or content of the output.

ECLIPSE 100

Properties

- The SWATINIT data is now honored in a gas zone. In previous versions the capillary pressure scaling was based only on the water-oil pressure difference. In a gas zone the capillary scaling is now based on the water-gas pressure difference.
- The calculations in the choices 5 and 7 (oil wet system) in the second option of the EHYSTR keyword have been slightly modified. In particular, if you select choice 7, it requires a connate oil saturation value to be entered, that is normally in that case a non-zero value (see the “Saturation Functions” chapter of the ECLIPSE Technical Description).

Well model

- The closure of additional connections “below” the offending one in +CON workovers in standard wells is now governed by the position of the connections along the wellbore (see keyword COMPORD). It is possible to revert to the pre-2002A treatment (which simply compares their true depth values) by setting OPTIONS keyword switch 53 > 0. Note also that the previous usage of this OPTIONS switch has been replaced by the COMPORD keyword.

Output and restarts

- The default maximum number of messages controlled by switch 23 of the OPTIONS keyword has been set to 15.
- Unrecognized keywords in the SUMMARY section are now reported as warnings rather than errors.
- The mnemonics used to output formation volume factors to the Restart file have been changed from 1/FVFOIL to 1OVERBO etc. For back compatibility the old names will be output if the ALLPROPS switch is set to 2 in the RPTRST keyword.
- The FIP array name is output in the fluid in place reports.
- The multi-segment well data written to the RFT file (activated by item 4 of keyword WRFTPLT) now includes the segment x- and y-coordinates. Their values may be entered in keyword WELSEGS to provide information for 3-D plots of well trajectories.

Reservoir Coupling

- As from the 2001A_1 release, Reservoir Coupling runs cannot be performed with an `EclipseLite` license. The master run (and any slaves run on different machines) will check out a full ECLIPSE license regardless of the number of wells. This change overcomes the problem of slaves checking out additional ECLIPSE licenses even though they are running on the same machine as the master.

ECLIPSE 300

AIM implicit cell selection

- The default selection criteria for implicit cells within an `AIM` run have changed. The pre-2002A method can be restored by setting the 69th switch in the `OPTIONS3` keyword. The 2002A settings are equivalent to having the `AIM1` mnemonic set in the `TOPT` keyword in the 2001A release. The changes made are:
  - 11th argument of keyword `TSCRIT` has changed; this limits the time step size after well modifications. For AIM runs this is now set to 40, thus allowing 1-month time steps, which is often desirable and achievable in history-matching datasets.
  - The default setting of the `AIMCON` keyword has changed, thus altering the default procedure for selection of implicit cells during an AIM run. The new setting often chooses less implicit cells for the early time in a history matching case (water-injection), and more implicit cells during a gas-injection prediction phase.

  These settings are believed to produce better answers more efficiently.

Performance

- The default linear solver method has been changed to use the WARP linear solver. The 2001A linear solver can be invoked by setting switch 66 in `OPTIONS3`. This change should not affect the results, but should improve the performance.
- Convergence criteria for the linear solver have been changed to be more like those in ECLIPSE 100 and to try and prevent the 'over-solving' that occurs as the solution is approached. As a result there is often a reduction in the number of linear iterations, and hence the CPU time. The solution should not in general be noticeably different. But in some cases, where there have been non-linear convergence problems and/or the non-linear tolerances have been relaxed, the differences can be more pronounced. Pre-2002A convergence can be reinstated by setting switch 61 in `OPTIONS3`.
- The `GAIN` option has been turned off by default. This means that ECLIPSE 300 will always solve the non-linear equations to the convergence tolerance. It can be turned on again using the `OPTIONS3` keyword switch 68.

Properties and equilibration

- The viscosibility in the `PVCO` keyword is now used by default. It can be ignored by setting switch 24 in `OPTIONS3` to -1.
- The `SWATINIT` data is now honored in a gas zone. In previous versions the capillary pressure scaling was based only on the water-oil pressure difference. In a gas zone the capillary scaling is now based on the water-gas pressure difference.
- A change has been made to the pressure used to calculate the phase densities in the equilibrator. The densities are thus set consistently with the subsequent simulation and are consistent with ECLIPSE 100. The change results in small differences in initial conditions. The old behavior can be restored by using switch 57 in `OPTIONS3`.

- The distribution of depth versus pressure nodes has been improved in the equilibrator. The old behavior can be restored by using switch 57 in OPTIONS3.
- The HCSCAL option in the FORMOPTS keyword has been made the default. It can be turned off with the NOHCSCAL option.
- The calculations in the choices 5 and 7 (oil wet system) in the second option of the EHYSTR keyword have been slightly modified. In particular, if you select choice 7, it requires a connate oil saturation value to be entered, that is normally in that case a non-zero value (see the "Saturation Functions" chapter of the ECLIPSE Technical Description).

Well model

- The default method of calculating the density of the fluid in the wellbore has been changed from the 'AVG' method to the 'SEG' method as described in item 12 of the WELSPECS keyword. The 'SEG' or segmented density method is more accurate than the other choice if different mixtures of fluids enter the well through each connection. This change has been found to improve convergence in some simulations and this default is now consistent with that of ECLIPSE 100. The previous default method ('AVG') can be restored by using the 65th item in OPTIONS3.
- The closure of additional connections "below" the offending one in +CON workovers in standard wells is now governed by the position of the connections along the wellbore (see keyword COMPORD). Similarly the segmented density method for calculating the wellbore hydrostatic head density (see item 12 of the WELSPECS keyword) now makes use of connection ordering along the wellbore rather than by depth. In both cases, setting the connection ordering to DEPTH in the COMPORD keyword will revert these calculations to their pre-2002A state.
- For wells connected to the inner-most grid block of a radial grid, the connection transmissibility factor is now calculated using the wellbore radius, (see the section entitled "The connection transmissibility factor" in the Well Inflow Performance chapter of the ECLIPSE Technical Description. Previously, the inner-most grid block radius was used in this calculation. (To revert to the former state, use switch 54 in OPTIONS3.) If the inner-most grid block radius and the wellbore radius differ, a warning message will now be generated.
- In the connection factor calculation for horizontal wells (see the HWELLS keyword) in a single layer model, unspecified Z direction permeabilities are now defaulted to the geometric average of the X and Y direction permeabilities. This is consistent with the default values used in ECLIPSE 100. Previously the Z direction permeability was set to 1.0e-10 in this situation.
- The operation of switch 46 in OPTIONS3, concerning the treatment of well revivals, has been changed to make it more consistent with the corresponding switch (15) in the ECLIPSE 100 OPTIONS keyword.
- Prior to this release, well potential rates may sometimes have been calculated at the well's BHP limit even though the rates at the well's THP limit are lower and thus should have been assigned as the well's actual potential. This has been corrected in the 2002A release. To assess any associated changes in results, please set item 54 of the OPTIONS3 keyword to 2.

Input/Output

- The behavior of the first data item on the TUNING keyword has been modified such an input value is taken as the maximum size of the next timestep.
- The AQUDIMS keyword has been modified to be more compatible with ECLIPSE 100.

- Summary file vectors for wells and groups that are not yet declared (pre-2002A they were output with the name ':+:+:+:+') are now not output at all to the RSM file.
- The summary section keyword PERFORMA now also outputs the CPU per time-step and the CPU per (simulated) day.
- The units used when outputting molar density data (MLSC in RPTSCHED or RPTRST) have been corrected to moles per reservoir volume, instead of moles per standard volume.
- The field units when outputting energy data (ENERGY in RPTSCHED or RPTRST; BENERGY and SENE in the SUMMARY section) have been corrected to BTU/barrel, instead of BTU/ft^3.
- NOMONITO has now become the default setting. Previous behavior can be produced by using the MONITOR keyword.

Flux boundary conditions

- In a full field DUMPFLUX run, checks are now in place to trap wells that are connected across flux regions and report this as an error. If you are certain that in the USEFLUX reduced run the flux regions crossed by wells are either all active or all inactive, then use switch 59 in OPTIONS3 to change this error message to a warning message.

Multi-segment wells and Thermal option

- In multi-segment wells, the water import options WIMPR and WIMPP in keyword WSEGEXSS, can now be used with the Thermal option.
- Default values for the multi-segment well convergence criteria (CVCRIT items 24-26) have be redefined.

Network option

- The 2001A version prevented extrapolation of the VFP table for the branch pressure losses. If the THP (outlet pressure), WFR or GFR value was outside the range of the table, the branch pressure drop was calculated at the nearest tabulated point. In the 2002A version the VFP table is linearly extrapolated in these circumstances, in common with the ECLPSE 100 Network option. To regain back-compatibility with the 2001A treatment, set OPTIONS3 keyword switch 63 = 1; this will prevent extrapolation of the VFP tables.

New keywords for 2002A

ECLIPSE 100

GLOBAL keywords

ENDSKIP	Causes keyword skipping (SKIP, SKIP100) to be terminated.
SKIP	Skips the following keywords until ENDSKIP is read.
SKIP100	Skips the following keywords until ENDSKIP is read.
SKIP300	Has no effect in ECLIPSE 100.

RUNSPEC section

MULTOUTS	Requests multiple Summary output files.
UNIFOUTS	Requests unified Summary output files.

GRID section

OPERATE	Performs arithmetic operations on arrays.

EDIT section

OPERATE	Performs arithmetic operations on arrays.

PROPS section

OPERATE	Performs arithmetic operations on arrays.

REGIONS section

OPERATE	Performs arithmetic operations on arrays.

SOLUTION section

OUTSOL	Equivalent to RPTRST.

SUMMARY section

BDEN(O,W,G)
: Block densities.

BGPC
: Gas-oil capillary pressure.

BKR(O,W,G)
: Alternatives to B(O,W,G)KR.

BS(OIL,GAS,WAT)
: Block saturations.

BV(OIL,WAT,GAS)
: Block viscosities.

BWPC
: Water-oil capillary pressure.

COPR, C(W,G)(I,P)R, C(O,W,G)PRL, C(W,G)IRL

Various connection flow rate quantities, for compatibility with ECLIPSE 300.

CTFAC Well connection transmissibility factor.

CVFR(L), CVPT(L), CVIT(L)

Reservoir fluid volume flow rates and cumulative production & injection totals, for individual well connections and lumped completions.

(F,G)SG(R,T)

Equivalent to (F,G)GS(R,T).

FRTM, RRTM, BRTM

Transmissibility modifier associated with rock compaction.

SDENM Segment mixture density.

S(GL,OW)(PP,VD)

Segment profile parameter and drift velocity for gas-liquid slip and oil-water slip in the Drift Flux slip model for multi-segment wells.

SPPOW Working power of a down-hole separator pullthrough pump.

W(W,G)IP Equivalent to W(W,G)PI.

ZIPEFF Predicted efficiency of time-step. (Mainly for developer use.)

ZIPEFFC Predicted efficiency divided by actual efficiency. (Mainly for developer use.)

SCHEDULE section

COMPORD Defines how the ordering of connections along the wellbore is determined.

OUTSOL Equivalent to RPTRST.

SIMULATE Counteracts a NOSIM keyword and turns on the time step simulation.

WCONINJH Observed rates and pressures for history matching injection wells.

Multi-segment wells WSEGDFPA Drift flux model parameters for multi-segment wells.

Multi-segment wells WSEGPULL Defines a segment to represent a pullthrough pump for a down-hole water separator.

ZIPP2OFF Turns off ‘intelligent time-stepping’.

ZIPPY2 Turns on ‘intelligent time-stepping’.

ECLIPSE 300

GLOBAL keywords

ENDSKIP Causes keyword skipping (SKIP, SKIP300) to be terminated.

SKIP Skips the following keywords until ENDSKIP is read.

SKIP100 Has no effect in ECLIPSE 300.

SKIP300 Skips the following keywords until ENDSKIP is read.

RUNSPEC section

FASTTRAC Enables faster tracer solution.

Gradient option FMTHMD Declares that the .HMD file is to be formatted.

Geomechanics option	GEODIMS	Sets number of geomechanics regions, dimensions of PERMSTAB and BIOTCTAB tables.
Geomechanics option	GEOMECH	Requests the geomechanics option.
Gradient option	HMDIMS	Specifies dimensions for the Gradient option.
	MULTOUTS	Requests multiple Summary output files.
Parallel option	PSPLITX	Specifies multiple domain decomposition in the X-direction.
Parallel option	PSPLITY	Specifies multiple domain decomposition in the Y-direction.
	REACTION	Specifies the number of chemical reactions.
	SKIPSTAB	Speeds up flash calculations by skipping some 1-phase stability calculations.
Gradient option	UNCODHMD	Requests that the .HMD file will contain unencoded derivatives.
	UNIFOUTS	Requests unified Summary output files.

GRID section

Geomechanics option	BIOTC	Set Biot's constant for each grid block.
Gradient option	HMMULTPV	Specifies pore volume modifiers (cumulative).
	OPERATE	Performs arithmetic operations on arrays.
Geomechanics option	POISSONR	Set Poisson's ratio for each grid block.
Geomechanics option	ROCKDEN	Set rock density for each grid block.
Local grid refinement	RPTGRIDL	Used to output GRID section data for local refined grids.
Geomechanics option	YOUNGMOD	Set Young's modulus for each grid block.

EDIT section

Gradient option	HMMULTPV	Specifies pore volume modifiers (cumulative).
	OPERATE	Performs arithmetic operations on arrays.

PROPS section

Geomechanics option	BIOTCTAB	Sets a Biot constant multiplier as a tabular function of maximum and minimum principal stress in each geomechanics region.
Thermal option	CCTYPE	Over-rides default formula for the component liquid phase compressibility.
Thermal option	CCTYPES	Over-rides default formula for the component liquid phase compressibility for surface calculations.
Thermal option	CVTYPE	Defines non-volatile and non-condensable components in live oil runs.
Thermal option	CVTYPES	Defines non-volatile and non-condensable components at surface conditions in live oil runs.
	DETAILMF	Defines the detailed initial conditions in terms of mole fraction versus depth for lumped simulations.
Thermal option	EQLDKVCR	Defines equilibrium values for equilibrium deviation reactions.
Thermal option	EQLDREAC	Defines equilibrium deviation reaction rates.
	FILLEPS	Requests output of endpoint data to the INIT file.
Geomechanics option	GEOYLDF	Sets properties of the yield function in each geomechanics region.
	INTPC	Invokes the Integrated Pc curve option for dual porosity

Thermal option	`KVCRS`	Defines K-values for surface calculations.
	`OPERATE`	Performs arithmetic operations on arrays.
Geomechanics option	`PERMSTAB`	Sets a permeability multiplier as a tabular function of maximum and minimum principal stress in each geomechanics region.
	`REACACT`	Defines the activation energy used in the chemical reaction rates.
	`REACCORD`	Defines the order of component terms in the chemical reaction rates.
Thermal option	`REACENTH`	Defines the reaction enthalpy.
	`REACPHA`	Defines the phases of the component terms in the chemical reaction rates.
	`REACPORD`	Defines the order of the porosity term in the chemical reaction rates.
	`REACRATE`	Defines the reaction rate constant for chemical reactions.
	`STOPROD`	Defines the stoichiometric coefficients of products in chemical reactions.
	`STOREAC`	Defines the stoichiometric coefficients of reactants in chemical reactions.

REGIONS section

Gradient option	`HMPORVM`	Specifies pore volume parameter region data.
	`OPERATE`	Performs arithmetic operations on arrays.
Geomechanics option	`YLDNUM`	Sets a geomechanics region number for each grid block.

SOLUTION section

RPTRST Equivalent to OUTSOL.

SUMMARY section

B(O,W,G)DEN

Block densities

B(O,W,G)KR

Alternatives to BKR(O,W,G).

B(O,W,G)VIS

Block viscosities.

CLFR(L), CLPT(L)

Connection liquid flow rate and production total (for E100 compatibility).

C(O,W,G)FR(L)

Connection flow rates (For ECLIPSE 100 compatibility).

CPR Connection pressure.

CTFAC Connection transmissibility factor.

CVFR(L), CVPT(L), CVIT(L)

Reservoir fluid volume flow rates and cumulative production & injection totals, for individual well connections and lumped completions.

FHPV Field hydrocarbon pore volume.

(F,G)MCTG Field and group gas injection control mode integer.

(F,G)MCTP Field and group production control mode integer.

(F,G)MCTW Field and group water injection control mode integer.

(F,G,W,C)OGR, COGRL

Oil-gas ratios.

(F,R)RPV Pore volumes.

FREAC, BREAC

Chemical reaction rate.

FREAT Chemical reaction total.

G(O,W,G,V)(P,I)RL and GLPRL

Group production/injection rate limits.

MAXCFL Outputs the maximum CFL value for an explicit cell when the stable time-stepping option is used (keyword CFLLIMIT).

SDENM Segment mixture density.

S(GL,OW)(PP,VD)

Segment profile parameter and drift velocity for gas-liquid slip and oil-water slip in the Drift Flux slip model for multi-segment wells.

TCPUDAY CPU time per day (or hour, in lab units).

TCPUTS CPU time per time-step.

WMCTL Well control mode integer.

WMVFP Well VFP table number.

SCHEDULE section

	CFLLIMIT	Turns on stable time-stepping for IMPES and AIM runs.
Local grid refinement	COMPAGHL	Sets the gravity head in local refinement completion cells.
Local grid refinement	COMPKRIL	Sets the injection relative permeability in local refinement completion cells.
Local grid refinement	COMPMBIL	Sets the injection mobility in local refinement completion cells.
	COMPORD	Defines how the ordering of connections along the wellbore is determined.
Gasfield operations	GASBEGIN	Denotes start of keywords to go into the Annual Scheduling File.
Gasfield operations	GASEND	Denotes end of keywords to go into the Annual Scheduling File.
Gasfield operations	GASFDELC	Specifies the definition of the delivery capacity.
Gasfield operations	GASMONTH	Denotes the times within each year that operations on the Annual Scheduling File should be performed, and requests printed reports.
Gasfield operations	GASPERIO	Advances the simulation through one or more complete contract periods.
Gasfield operations	GASYEAR	Advances the simulation through one or more complete contract years.
Gasfield operations	GDCQ	Sets the initial DCQs for individual contract groups.
Gradient option	GRADWELL	Specifies well quantities to be output to the .HMD file.
Gasfield operations	GSWINGF	Turns on the Gas Field Operations model in the Multiple Contract Groups mode. It identifies the groups having gas supply contracts, and sets their swing and profile factors.
Thermal option	REACCRIT	Interpolation criteria for chemical reactions.
	RPTRST	Equivalent to OUTSOL.
	SIMULATE	Counteracts a NOSIM keyword and turns on the time step simulation.
	SKIPREST	Makes ECLIPSE skip the subsequent keywords in a restart run until the restart time is reached.
Geomechanics option	STRESSBC	Sets stress and velocity boundary conditions on external and internal boundaries.
Gasfield operations	SWINGFAC	Specifies the Swing Factors and Profile Factors for each month, which are applied to the FIELD.
	WCONINJH	Observed rates and pressures for history matching injection wells.
Local grid refinement	WELMOVEL	Moves the well head to an LGR.
	WRFTPLT	Requests output of RFT, PLT and segment data to the RFT file.
Multi-segment wells	WSEGDFPA	Drift flux model parameters for multi-segment wells.

Altered keywords for 2002A

ECLIPSE 100

RUNSPEC section

ACTDIMS	The default value of item 2 has been increased to 50 to be the same as in ECLIPSE 300.
NOSIM	This keyword may now be entered in the SCHEDULE section to turn off time step simulation at a specific point in time.
PATHS	The maximum number of aliases has been increased to 100. The maximum length of a pathname has been increased to 132 characters.
ROCKCOMP	A new option REVLIMIT has been added.
TRACERS	It is now possible to set a minimum number of Newtons for the tracer calculation.

GRID section

USEFLUX	A second argument can be used to request extrapolation beyond the end of the original run.

PROPS section

EHYSTR	An eighth option has been added to select the wetting phase between oil and gas in the three-phase case.
TZONE	The keyword can now be used in runs without the mobile oil correction active.

SCHEDULE section

ACTIONW	New trigger on the well bottom hole gas liquid ratio (WBGLR).
WELOPEN	New option POPN in item 2 opens the well only if it is priority-closed.
WELOPENL	New option POPN in item 3 opens the well only if it is priority-closed.
WINJMULT	New item 4 to specify mode of operation.
	New items 5 - 7 to apply multiplier to individual connections.
RPTRST	ALLPROPS can be set to 2 to return the old output mnemonics for formation volume factors (1/FVFOIL etc.).
	The DRAIN mnemonic can now be set to 2 to output drainage region sinks.

OPTIONS keyword

Switch 53	The original usage of this switch has been replaced by the COMPORD keyword. The switch may now be used to revert to the pre-2002A method of determining which additional connections to close in +CON workovers.
Switch 83	This new switch can be used to ensure the saturation values that are output are between 0.0 and 1.0

ECLIPSE 300

GLOBAL keywords

OUTSOL	RS and RV added in compositional runs. FIP added.

RUNSPEC section

AQUDIMS	Now reads in 6 data items to be compatible with ECLIPSE 100.
MEMORY	The defaults have been increased.
NETWORK	Item 3 is added to allow input of maximum number branches connected to a node.
FORMOPTS	A new option BOILMOD has been added. The HCSCAL option is now the default, hence a NOHCSCAL option is available to turn off the option.

GRID section

DGRID	Will now will work with an LGR.
RPTGRID	When the multi-point flux option MPFA is used, cell reverse transmissibilities and k-orthogonality errors can be output using the mnemonics MPFTREV and MPFORTH.
USEFLUX	A second argument can be used to request extrapolation beyond the end of the original run.

PROPS section

The keywords PCRITW and TCRITW, which specified the critical pressure and temperature of water for the Thermal option have been removed since they are not used.

EHYSTR	An eighth option has been added to select the wetting phase between oil and gas in the three-phase case.
PVCO	The oil viscosibility (entered in the 6th column) is now recognized.
TZONE	The keyword can now be used in runs without the mobile oil correction active.

SOLUTION section

EQUIL	Item 10 can now be set equal to 2 or 3 in THERMAL simulations, for equilibrating with a gas cap.
OUTSOL	The PPCW, SWCR, SGCR, SOWCR, SOGCR mnemonics are available to output processed end-point arrays.
	The CFL mnemonic can be used to output the CFL value in each cell when stable time-stepping has been used (see keyword CFLLIMIT).
RPTSOL	The PPCW, SWCR, SGCR, SOWCR, SOGCR mnemonics are available to output processed end-point arrays.

SCHEDULE section

CVCRIT	New item to control convergence of drift-flux segments in multi-segment wells.
RPTRST	New mnemonic REAC added to output chemical reaction rates.
	The mnemonics BASIC and FREQ are now available in ECLIPSE 300.
	Geomechanics rock effective stresses, principal stresses and rock displacements can be output with EFFSTRES, PRSTRESS and ROCKDISP.
RPTSCHED	New mnemonic REAC added to output chemical reaction rates.
TUNING	The first data item is now interpreted as the maximum size of the next timestep.
WELOPEN	New option POPN in item 2 opens the well only if it is priority-closed.
WELOPENL	New option POPN in item 3 opens the well only if it is priority-closed.
WELSPECS	The default density calculation (item12) has been changed to 'SEG'.
WELSPECL	The default density calculation (item13) has been changed to 'SEG'.
WSEGEXSS	Options WIMPR and WIMPP in item 3 can now be used with the Thermal option.

OPTIONS3 keyword

Switch 24	If set to -1 will ignore the viscosibility (sixth column) in the PVCO keyword.
Switch 46	Modified to make it more consistent with the corresponding switch (15) in the ECLIPSE 100 OPTIONS keyword. These switches control how many times a well will be revived if it is unable to flow given the current reservoir state.
Switch 48	Geomechanics: if set to 3, turns off the rock conservation equation. In this case, porosity does not change from either initial values or the values at the time this switch is invoked. Setting this to 4 turns off the rock/fluid gravity contribution to the geomechanics force balance. A value of 10 turns off both of the above.
Switch 54	Overrides some corrections made to the well calculation for the 2002A release, restoring compatibility with earlier releases to allow you to assess the importance of the changes.
Switch 56	Can be used to modify the interpolation procedure in black oil cases.
Switch 57	If set=1 restores the pre-2002A equilibration behavior.
Switch 59	If set >0, in runs using flux boundary conditions certain error conditions are reduced to warnings to allow the run to proceed.
Switch 61	If set >0 it restores pre-2002A convergence criteria on the linear solver.
Switch 62	If set >0 in the RUNSPEC section, all wells are internally converted to single segment multi-segment wells and the multi-segment well model will be used in preference to the standard well model.

Switch 63	If set=1, prevents extrapolation of VFP tables in the Network option.
Switch 65	Can be used to alter the method of calculating the density of the fluid in the wellbore. This density is used to determine the pressure head between a well's bottom hole pressure and the pressure at a connection.
Switch 66	If set >0 it restores the 2001A linear solver.
Switch 67	Can be used to alter the convergence tolerance in the standard well model.
Switch 68	If set>0 turns on the 2001A "gain" Newton convergence method.
Switch 69	If set>0 reverts to the 2001A implicit cell selection criteria.

Developments in 2001A

New facilities for 2001A

ECLIPSE 100

Grid data

- New keyword added to the GRID and EDIT sections, MULTREGP. This keyword can be used to set a pore volume multiplier for a specific region. The regions must previously have been defined using keyword FLUXNUM or MULTNUM in the GRID section.

Properties and equilibration

- When checking the total compressibility of the hydrocarbon system, the minimum pressure and the number of pressure nodes used can be modified using the PMAX keyword.
- It now possible to enter the hysteresis curvature parameters by region. See the EHYSTRR keyword.
- Secondary drainage curves in hysteresis data sets with dissolved gas can now be recalculated so that if the gas saturation goes below the trapped gas due to the resolution of the gas, the gas will become mobile immediately at the start of secondary drainage. See the HYMOBGDR keyword.
- It is now possible to enter the datum depth for depth corrected pressures by FIP region. See the DATUMR keyword.
- It is now possible to use a flux limiting scheme to solve the energy equation and hence reduce the numerical dispersion in the temperature option. The scheme is analogous to the flux limitation scheme (TRACTVD) for single phase tracers. See the TEMPTVD keyword.

Well model

- The drawdown limit set in keyword WELDRAW may now be applied to restrict the **maximum** drawdown in the well's connections, as an alternative to the PI-weighted average drawdown of the connections.
- In the ACTIONW and ACTIONG keywords, triggers on the liquid and reservoir volume rates and totals, and the gas to liquid ratio, have been added.
- If the data is run in NOSIM mode, ECLIPSE will now perform some basic syntax checking on keywords within the ACTION / ENDACTIO bracket. However, full data checking of these keywords cannot take place until they are actioned during the simulation.
- In the GTMULT keyword, the group's injection rate target or limits may now be multiplied by a specified factor.
- There is an option in keywords WCUTBACK and WBHGLR to reverse completely any rate cutbacks previously applied to the well, when the well is worked over after violating an economic limit. For example, the closure of the well's lowest completion may make it less prone to water coning and consequently it could produce at a higher rate without increasing its water cut. Progressive cutback reversals would increase its rate, but there is now an option to increase its rate instantaneously by removing the rate limit previously applied by the cutback process.

Output

- Well and group guide rates for production and injection can be output to the Summary file. See the (G,W)(O,W,G,V)PGR and (G,W)(W,G)IGR keywords
- The Summary section keyword FMWSET may be used to request the complete set of well count mnemonics for all production and injection wells, together with the drilling queue and workover event mnemonics. Similarly, the keyword GMWSET requests this complete set for all wells subordinate to a given group.
- In a polymer run, with 'VISC' requested in RPTRST, the effective water viscosity will now also be output with the heading 'EWAT_VIS'.
- The MULTREGP keyword, which is used to set a pore volume for a specific region, can now be output to the Print file under the RPTGRID keyword.
- It is now possible to reduce the restart file size in cases with many wells. See the RPTRST keyword.

Environmental tracers

- New item 6 in the TRACERS keyword in the RUNSPEC section allows the user to specify the maximum number of non-linear iterations the tracer algorithm may perform.

Flux boundary conditions

- Keyword USENOFLO allows a USEFLUX data set to be run without the flux file.

Gradient option

- It is now possible to request gradients of porosity and phase densities at reservoir conditions using the GRADRESV keyword. These may be used to calculate gradients with respect to an external petro-elastic model.
- Gradients of field and group quantities can now be output to the HMD file. See the GRADGRUP, RPTHMG and RPTHMD keywords.
- In a gradient run, with 'PORO' requested in the RPTRST keyword, the effective porosity will now also be output under the heading, 'PORO_EFF'. Note this will only be the case when 'PORO' is also requested using the GRADRESV keyword.

Local grid refinement

- ECLIPSE 100 now allows the use of nested Local Grid Refinements. Nested local grids can only be solved "in place". The parent local grid must be a Cartesian grid refinement (see keyword CARFIN) and neither the parent nor the child can be part of an amalgamated local grid. This option may not be used with the Parallel option or the Gradient option.
- WELMOVEL allows a well which was specified in the global grid to be moved into a local grid.

Multi-segment wells

- A new device model is available, representing a sub-critical valve with a specified throat cross-section area. See keyword WSEGVALV.
- Water or gas may be imported into a segment from an external source. Applications include the modeling of lift gas injection and the injection of water used to power down-hole hydraulic pumps. See keyword WSEGEXSS.
- The initial pressure and fluid contents of each segment may be set manually. See keyword WSEGINIT.

- A new ACTION keyword has been introduced, ACTIONS, that allows a set of SCHEDULE section keywords to be processed when a segment of a multi-segment well satisfies a specified trigger condition.

ECLIPSE 300

Grid data

- The keyword GADJUST takes specified permeabilities as representing a permeability tensor specified as a diagonal matrix whose eigenvectors correspond to the grid I, J and K directions.
- The basis for representing a permeability tensor can now be specified by the TBASIS keyword.
- The application of 9-point transmissibilities can now be additionally controlled by a pore volume limit using keyword MINPORV9 or by orthogonality errors using keywords ORTHERRO, ORTHERRX, ORTHERRY and ORTHERRZ.
- The MULTPV keyword can now be used to specify the pore volume multipliers. This keyword can be used in both the GRID and EDIT sections.
- The keyword MINPVV can be used to set a minimum pore volume which a cell must exceed or it will be made inactive. One threshold pore volume value is input for each cell in the current box.
- The new keyword MULTREGP in the GRID and EDIT sections can be used to set a pore volume multiplier for a specific region. The regions must previously have been defined using keyword FLUXNUM or MULTNUM in the GRID section.
- The DPNUM keyword is now available in ECLIPSE 300.

Properties and equilibration

- The PMAX keyword is now available for black oil runs.
- The rock hysteresis data ROCKTABH can now be used to model cyclic deflation and dilation. The new option is controlled by the ROCKCOMP and ROCKOPTS keyword.
- It is now possible to specify the initial CO_2 concentration in CO2SOL cases with enumerated starts, using either the AMF or the RSW keyword.
- It is now possible to use the ZMF keyword in equilibrated cases (EQUIL keyword); in this case the ZMF values overwrite the hydrocarbon compositions. This can be useful to model areal variations in composition while maintaining a good approximation to the hydrocarbon / water equilibrium.
- It is now possible to use XMF, YMF and ZMF keywords within a BOX.
- In order to maintain backwards compatibility with pre-2000A versions the OLDEQUIL keyword will cause the grid to be equilibrated using non-tilted block equilibration with the number of layers specified in item 9 of the EQUIL keyword.
- The TZONE keyword is now available with ECLIPSE 300.
- The SWATINIT keyword is now available with ECLIPSE 300.
- The PPCWMAX keyword is now available with ECLIPSE 300.
- It is now possible to enter the datum depth for depth corrected pressures by FIP region. See the DATUMR keyword.

- Using the FIPSEP keyword, different fluid-in-place regions may now be assigned different separators. These separators are used in the calculation of the in-place oil and gas volumes as reported in the print file.
- The PEDERSEN viscosity calculation can now be used to determine the viscosities in a compositional run, instead of the default Lorentz-Bray-Clark method.

Well model

- The pressure equivalent radius for a connection may now be specified in the COMPDAT keyword. This value will be used in place of that calculated from the grid block dimensions according to Peaceman's formula. The pressure equivalent radius appears in the connection transmissibility factor formula for non-radial grids.
- The drainage radius for productivity/injectivity index calculations may now be specified in the WELSPECS keyword. If this radius is not supplied, the pressure equivalent radius of the grid blocks containing the well connections will be used.
- The productivity/injectivity index (P.I.) value of one or more wells may now be set to a specified value using the keyword WELPI.
- A crossflow ability flag may now be specified in item 10 of the WELSPECS keyword. If this is set at 'NO', the connections will act as one-way valves, preventing flow if there is a reverse drawdown.
- An alternative segmented method for calculating the density of the fluid in the wellbore may now be selected by setting item 12 of the WELSPECS keyword to 'SEG'. This density is used to calculate the hydrostatic head between the well's bottom hole reference depth and each of its connections. In this method the flowing mixture density is calculated between each adjacent pair of connections, and depends on the relative quantities of the phases flowing in these particular sections of the wellbore. This calculation is more accurate than the default average density calculation if different mixtures of fluids enter the well through each connection. However, it is treated explicitly in the numerical scheme (calculated once at the beginning of each time-step) and in some circumstances it may be less stable than the default averaging scheme which is implicit.
- Velocity Dependent Relative Permeability (VDRP) is now incorporated into the Pseudo-Pressure integral if VDRP is requested together with the Generalized Pseudo-Pressure option (see keywords VELDEP and PSEUPRES). Only capillary number dependence is implemented. This facility is activated by setting the 4th item of PICOND to 'YES'.
- The drawdown limit set in keyword WELDRAW may now be applied to restrict the **maximum** drawdown in the well's connections, as an alternative to the PI-weighted average drawdown of the connections.
- The production guide rate for a nominated phase may now be calculated from a general formula using the keyword GUIDERAT. This calculated guide rate will be used for wells under group control that do not have fixed guide rates set with the keyword WGRUPCON. It will also be used for groups whose guide rate definition is set to 'FORM' in keyword GCONPROD, when a flow target of a superior group is apportioned amongst subordinate groups.
- As an alternative to the **sequential** drilling queue for wells (see keyword QDRILL), a **priority** drilling queue may now be defined using the keyword WDRILPRI. If a group (including the field) cannot meet its production target, the production well with the highest priority in the drilling queue that is subordinate to that group, and not subordinate to any other group under the same mode of production control, will be opened automatically.

Similarly, if a group cannot meet its injection target, the injection well with the highest priority in the queue that is subordinate to that group, and not subordinate to any other group under injection control for the same phase, will be opened automatically. Priority values may either be set manually or calculated using the drilling priority equation (see keyword DRILPRI).

- For compatibility with ECLIPSE 100, the family of ACTION keywords has been extended to include ACTIONW, ACTIONG and ACTIONR. These keywords allow a set of SCHEDULE section keywords to be stored for future processing when a nominated well, group or fluid-in-place region satisfies a specified trigger condition. The user can choose whether the action is to be performed only once, or repeatedly with each timestep while the triggering condition is satisfied.
- If the data is run in NOSIM mode, ECLIPSE will now perform some basic syntax checking on keywords within the ACTION / ENDACTIO bracket. However, full data checking of these keywords cannot take place until they are actioned during the simulation.
- Well production rate and injection rate targets and limits may now be multiplied by a specified factor using the WTMULT keyword. Similarly, group production rate or injection rate targets and limits may be multiplied by a specified factor using the GTMULT keyword.
- The last four arguments of the CECON keyword have been enabled for use by ECLIPSE 300. Limits may now be placed on the minimum oil and gas flow rates in connections. Also, flags may now be set to control whether the economic limits apply to stopped wells and if a connection breaks a limit, whether or not all connections below it are additionally shut.
- Using the keyword COMPLUMP, connections in a well may now be lumped together into a completion for simultaneous closure in automatic workovers after violating limits set in WECON, CECON, GECON, GCONPROD or GCONPRI. Completions may also be used in WELOPEN and WPIMULT keywords to refer to a range of connections and in the SUMMARY section to output a selected quantity totaled over all the connections that belong to the same lumped completion. The COMPLMPL keyword may be used to lump well connections together into completions when the connections are located in local grid refinements.
- There is an option in the keyword WCUTBACK to reverse completely any rate cutbacks previously applied to the well, when the well is worked over after violating an economic limit. For example, the closure of the well's lowest completion may make it less prone to water coning and consequently it could produce at a higher rate without increasing its water cut. Progressive cutback reversals would increase its rate, but there is now an option to increase its rate instantaneously by removing the rate limit applied by the cutback process.
- The GPTABLE3 keyword may now be used to define a gas plant table with oil, NGL and gas recovery fractions specified by the user. This keyword should be used instead of GPTABLE or GPTABLEN if it is required that the sum of these factors is not unity, thus allowing a component to be reduced in or excluded from a separator calculation.

Output and restarts

- The MULTPV array specifying pore volume multipliers can now be output to the Print file (under the RPTGRID keyword), and also output to the INIT file.
- The MINPVV array, which is used to set the minimum pore volume that cells must have to be active, can be output both to the Print file (under the RPTGRID keyword) and to the INIT file.
- The MULTREGP keyword, which is used to set a pore volume multiplier for a specific region, can now be output to the Print file under the RPTGRID keyword.

- The total compressibility of the fluid (hydrocarbon plus water) can be output to the Print and Restart files by using the 'TOTCOMP' mnemonic in the RPTSCHED and OUTSOL keywords.
- Coloring and sub-grid data, used by the WARP linear solver, can be output to the Restart files by using the 'COLR' and 'SUBG' mnemonics in the OUTSOL keyword.
- A group hierarchy tree is now drawn in the Print file in the 'Production and injection data for groups' section of the report step following a GRUPTREE keyword.
- Field, group and well hydrocarbon component molar production and injection rates and totals may now be written to the Summary file. See the (F,G,W)CM(P,I)(R,T) keywords.
- Group and field potentials may be written to the Summary file. See the (F,G)(O,W,G)PP and (F,G)(W,G)PI keywords.
- Well and group guide rates for production and injection can be output to the Summary file. See the (G,W)(O,W,G,V)PGR and (G,W)(W,G)IGR keywords.
- A variety of well counts can now be output to the Summary file. They output the number of wells that are flowing, abandoned or unused. Counts can also be made of the number under group control, controlled by their own surface rate limit, controlled by their own reservoir volume rate limit, controlled by pressure and, for producers, controlled by their own oil rate limit. Separate counts are available for production wells and injection wells and the counts may either cover all wells in the field or only those that are subordinate to a given group. See the (F,G)MWP(T,R,A,U,G,S,V,P,O) and (F,G)MWI(T,N,A,U,G,S,V,P) keywords.
- Drilling queue and workover events for all wells subordinate to a given group may also now be written to the Summary file. See the (F,G)MWD(R,T) and (F,G)MWW(O,T) keywords.
- The keyword FMWSET may be used to request the complete set of well count SUMMARY section mnemonics for all production and injection wells together with the drilling queue and workover event mnemonics. Similarly the keyword GMWSET requests this complete set for all wells that can be hierarchically traced to a given group.
- The SUMMARY section keyword WMCON reports the number of open connections for a given well.

Tracers

- The TRACK option has been enhanced to allow tracking of components from user defined regions. See keywords TRACKREG and TRACK.
- To a greater or lesser extent most compositional runs have components that actually represent a lumping of several other components. It is now possible to partially model these detailed components using tracers. The option is switched on using the LUMPDIMS keyword in the RUNSPEC section. The detailed components are defined in the PROPS section using the keyword LUMPING together with DETAILVD to describe the detailed initial condition, and the physical properties of the detailed components are defined with the keywords ACFDET, MWDETAIL, TCRITDET, PCRITDET, ZCRITDET, VCRITDET, OMEGAADE and OMEGABDE. (The surface condition equivalent component property keywords are TCRITSDE, PCRITSDE, ZCRITSDE, VCRITSDE, OMEGAASD and OMEGABSD.) The injection characteristics of the detailed fluid are described using the WINJEDET keyword.

- Traced mass rate, mass totals, and mole fractions of hydrocarbons can be output to the Summary file with the keywords `(F,G,W)TM(R,T,F)`. Traced volume rates and volume totals are available for the liquid and vapor phases with the keywords `(F,G,W)T(V,T)(L,V)`.

Performance

- The speed of running `USEFLUX` cases has been improved by preventing the need to read through the entire flux file up to the current time location at each timestep.
- A number of options to modify the formulation in the compositional model have been added. These are particularly important when modeling gas injection in general and first contact miscible cases in particular. See the `FORMOPTS` keyword.

Local grid refinement

- Unstructured (PEBI) grids can now be run as local grid refinements. The `EXTFIN` keyword can be used to specify an unstructured local grid refinement. The `EXTHOST` keyword can be used to specify the global host cells for an unstructured local grid refinement, or similarly, the `EXTREPGL` keyword can be used to specify the replaced global cells for an unstructured local grid refinement. Finally, the `TRANGL` keyword can be used to specify the global-local connections and associated transmissibilities for an unstructured local grid refinement. Within any `EXTFIN/ENDFIN` bracket, either `EXTHOST` or `EXTREPGL` should be used, but not both. The 4 new keywords mentioned here are generally supplied by another program and will not be modified by the user.
- The keyword `WELOPENL` can be used to shut or reopen wells or well connections in local grids, without having to specify the rest of the well control data or connection data.

Multi-segment wells

- A new device model is available, representing a sub-critical valve with a specified throat cross-section area. See keyword `WSEGVALV`.
- A labyrinth device may now be modeled using the same keywords and having the same functionality as ECLIPSE 100. See keyword `WSEGLABY`.
- Water or gas may be imported into a segment from an external source. Fluid may also be removed from a segment to an external sink. Applications include the modeling of lift gas injection and the injection of water used to power down-hole hydraulic pumps, and modeling recirculating wells in Thermal simulations. See keyword `WSEGEXSS`.
- Multi-segment wells may now be used in conjunction with the Generalized Pseudo-Pressure option (`PSEUPRES`) and the `CO2SOL` option.
- A new `ACTION` keyword has been introduced, `ACTIONS`, that allows a set of `SCHEDULE` section keywords to be processed when a segment of a multi-segment well satisfies a specified trigger condition.
- Multi-segment wells can be used with the Thermal option. Heat storage and conduction in the tubing and casing can be specified using the `WELSEGS` keyword, and modified with the `WSEGPROP` keyword.

- In Thermal Option applications, a heat transfer coefficient may be defined at a segment node. With a suitably defined thermal resistance and thermal contact length, heat transfer can take place from the segment to a reservoir grid block, to another segment or to a specified fixed external temperature. The thermal resistance is defined by keyword `WSEGHEAT`. This keyword is also be used to define the thermal contact length for heat transfer between segments and between a segment and an external temperature. For heat transfer between a well and the reservoir, the thermal contact length (the length of well in each completion cell) is defined by the `COMPSEGS` keyword.
- In Thermal Option applications, the segment temperature, energy, enthalpy flow rate and steam quality can be output to the Summary file. See the keywords `STEM`, `SENE`, `SHFR` and `SSQU`.

Network option

- This facility is now available in ECLIPSE 300. Either the Standard or the Extended network model may be used, although for new data sets you are recommended to use the Extended model. See "The Network Option" chapter of the ECLIPSE Technical Description for further details. The differences between the ECLIPSE 300 and ECLIPSE 100 implementations are summarized in the section "The network model in ECLIPSE 300" in the same chapter.

Parallel option

- The WARP linear solver can be used with the parallel option. See "The ECLIPSE 300 WARP linear solver" section in the "Solution of the Linear Equations" chapter of the ECLIPSE Technical Description for further details.

Thermal option

- A heat of vaporization can be defined at the standard temperature using the keyword `HEATVAPS`.
- The steam trap can now be controlled by specifying a temperature offset instead of a pressure offset, using item 17 of the `WCONPROD` keyword.
- The summary vector `BPSAT` can now be used with the Thermal option.
- Field and group steam production can now be output to the Summary file with keywords `(F,G)STP(R,T)`. The cumulative oil-steam ratio and the cumulative steam-oil ratio can be output using keywords `(F,G)OSRC` and `(F,G)SORC`.
- Multi-segment wells can be used with the Thermal option.
- Temperature, energy, enthalpy flow rate and steam quality in a segment in a multi-segment well can be output to the Summary file. See the keywords `STEM`, `SENE`, `SHFR` and `SSQU`.

Behavioral changes for 2001A

These may cause changes to the results of a simulation, or alter the form or content of the output.

ECLIPSE 100

Properties

- If the SURFTENS option is active (SATOPTS keyword) with the integrated Pc option (INTPC keyword), the surface tension modification is now applied to the integrated Pc curve. To return to the pre-2001A behavior the SURFTENS option should be turned off.
- In previous versions of ECLIPSE, the SOM parameter in the STONE1 three-phase relative permeability model was not set to the minimum of the scaled SOWCR and SOGCR values, but to the minimum of the table values. The scaled values are now honored. The pre-2001A behavior can be achieved by setting switch 79 in the OPTIONS keyword to -1.
- In previous versions of ECLIPSE the SOMGAS and SOMWAT data was not scaled in end-point scaling runs, which could potentially lead to non-physical oil relative permeabilities. The SOMGAS and SOMWAT data is now scaled to honor the SOWCR and SOGCR values.

Well model

- Switch 73 of the OPTIONS keyword is no longer in use. Use item 4 of keyword WVFPEXP to prevent wells switching to THP control while they are constrained to operate on the unstable side of their VFP curve.

Output and restarts

- The default of the GRIDFILE keyword has changed such that the default grid file output uses the extensible grid file. Data sets which only specify the first argument of the GRIDFILE keyword will now produce two grid files.
- Though not specifically banned, historical group and field Summary file quantities (FOPRH, FOIRH, FOITH, FOPTH etc.) were not compatible with the use of the GEFAC keyword in releases prior to 2001A. This is now corrected, but the output will be inconsistent in restarts from previous releases.

OPTIONS keyword

- The OPTIONS keyword can now only be used in the RUNSPEC and SCHEDULE sections.

Polymer flood model

- Pre-2000A polymer concentrations were ‘capped’ at twice the maximum injected concentration. Thc cap has now been removed. Pre-2000A behavior can be restored by setting OPTIONS keyword switch 81. This caps the polymer concentration to (OPTIONS(81) + 1) times the maximum injected concentration; a value of 1 therefore reproducing pre-2000A behavior.

ECLIPSE 300

RUNSPEC section

- The EQLOPTS keyword now accepts the arguments MOBILE. This has the same effect in ECLIPSE 300 as in ECLIPSE 100.

Properties and equilibration

- Positive values in the 9th element of the EQUIL keyword will now cause a tilted block equilibration to be performed.

- The sign of the capillary pressure at the water-oil contact has changed. A positive value now implies that $P_{oil} > P_{water}$ at the contact.
- When using the CO2SOL model the water properties have to be entered using the PVTW keyword. If the WATERTAB keyword is used and PVTW has not been specified, an Error rather than a Warning is issued.

Well model

- For compatibility with ECLIPSE 100, if a connection transmissibility factor has been supplied in item 8 of the COMPDAT keyword, it will be used to calculate either the connection's effective Kh or the geometric denominator term according to how the Kh value is set in item 10 of this keyword. Setting switch 43 in the OPTIONS3 keyword to 1 will revert to the 2000A treatment, whereby an unset Kh value will be calculated from the grid block data regardless of whether the connection transmissibility factor is set; and the geometric denominator term will not be made consistent with the given connection transmissibility factor and Kh values.
- The procedure for handling THP controlled wells, in particular the initial guess and the detection of dead wells, has been improved. See the description switches 45 and 46 in the OPTIONS3 keyword for information on the various options and how to achieve backwards compatibility with the 2000A release of ECLIPSE 300.
- When using the old group control keywords (GRUPPROD or GRUPINJE), it is possible to set groups under the control of a higher level group (control mode 'GR') without explicitly setting a guide rate (with GRUPGR). They will now have their control mode automatically reset to passive and will be reported as such. Setting switch 43 in the OPTIONS3 keyword to 3 will revert to the 2000A treatment, whereby such a group is still reported as under 'GR' control. This switch also reverts to setting the default guide rates for groups equal to their potentials. The default value for group guide rates in 2001A is zero (which means 'undefined'), and groups will only have guide rates if they are set explicitly. This is now consistent with ECLIPSE 100.

Linear solver

- The WARP and ECLSOL linear solvers cannot be interchanged during the run.
- The WARP linear solver may only be initiated by the WARP keyword.

Input/Output

- The AQUANCON keyword has been made more compatible with ECLIPSE 100. Referring to the aquifer influx coefficient (9th item of keyword), if a reservoir cell is defined more than once, its previous value for the aquifer influx coefficient is now added to the present value. The exception is the case where the user enters a value of zero, which then sets the influx coefficient for all the cells in the box to zero. Previously, ECLIPSE 300 output all connections.
- Units previously output as nnn/H are now output as nnn/Hour.
- The XMF,YMF and ZMF keywords will now normalize any cells where the component sum is within 10% of unity. An error will be generated if the discrepancy is greater than 10%.
- Output to the Run Summary file will be in wider columns, consistent with ECLIPSE 100 output. The keyword NARROW can be used to output data in the pre-2000A format.
- The default of the GRIDFILE keyword has changed such that the default grid file output uses the extensible grid file. Data sets which only specify the first argument of the GRIDFILE keyword will now produce two grid files

OPTIONS3 keyword

- The OPTIONS3 keyword can now only be used in the RUNSPEC and SCHEDULE sections.

Local grid refinement

- An error message will now be generated when a well completed in an LGR is specified in keywords that require connection I,J,K values and are not designed for LGR wells (e.g. CECON, WELOPEN). The data will be accepted, however, if the connection I,J,K values are all defaulted so that the data applies to all connections in the well. Setting switch 43 in the OPTIONS3 keyword to 2 will revert to the 2000A treatment, whereby the connection I,J,K values are assumed to be in the simulation grid with the LGR positioned below the global grid. These I,J,K values are not the same as those used to specify the connections in the LGR using COMPDATL.

Thermal option

- The horizontal thermal transmissibility calculation is based on the full cell cross-sectional area and does not include a net-to-gross factor (see "Thermal conductivity"). The pre-2001A calculation, which does include a net-to-gross factor, can be retained by setting switch 52 in the OPTIONS3 keyword to 1.
- The calculation of the thermal hydrostatic head in the well model has been changed to bring it into line with the new explicit hydrostatic head option for compositional simulations. To retain the pre-2001A2001A calculation, item 44 of keyword OPTIONS3 needs to be set to 99.

New keywords for 2001A

ECLIPSE 100

GRID section

MULTREGP	Sets a pore volume multiplier for a specific region.
USENOFLO	Allow a use flux data set to be run without the flux file.

PROPS section

EHYSTRR	Allows input of the hysteresis curvature parameters by SATNUM region.
HYMOBGDR	Changes the way secondary drainage curves are calculated in hysteresis runs with dissolved gas.
TEMPTVD	Invokes the flux limiting scheme for the temperature option.

SOLUTION section

DATUMR	Allows input of the datum depth by FIP region.

SUMMARY section

(G,W)(O,W,G,V)PGR	Group and well production guide rates for oil, water, gas and reservoir voidage.
(G,W)(O,W,G)IGR	Group and well injection guide rates for oil, water, and gas.
S(W,G)IM(R,T)	Segment water and gas import rates and cumulative totals from an external source into a multi-segment well (see keyword WSEGEXSS).
FMWSET	Requests the complete set of well count mnemonics for all production and injection wells together with the drilling queue and workover event mnemonics.
GMWSET	Requests the complete set of well count mnemonics for all production and injection wells that can be hierarchically traced to a given group together with the drilling queue and workover event mnemonics.

SCHEDULE section

ACTIONS	Initiates a set of keywords to be processed when a segment of a multi-segment well satisfies a specified condition.
GRADGRUP	Defines group derivatives required by SimOpt.
RPTHMG	Nominates groups for which derivatives are requested by SimOpt.
WELMOVEL	Allows a well to be moved from the global grid to a local grid.
WSEGEXSS	Imports water or gas into a multi-segment well from an external source.
WSEGINIT	Sets the initial pressure and fluid contents of well segments.
WSEGVALV	Defines segments in a multi-segment well to represent a sub-critical valve.

ECLIPSE 300

RUNSPEC section

FORMOPTS	Options to modify the formulation.
LUMPDIMS	Dimensions for lumped/detailed simulation.
NETWORK	Activates the Extended Network Model.
WARP	Enable WARP linear solver

GRID section

DPNUM	Allows single porosity regions to be specified in dual porosity runs.
EXTFIN	Specifies an unstructured local grid refinement.
EXTHOST	Used to specify the global host cells for an unstructured local grid refinement.
EXTREPGL	Specifies the replaced global cells for an unstructured local grid refinement.
GADJUST	Specify a permeability tensor which is diagonal along the grid I, J and K directions.
MINPORV9	Specifies a threshold pore volume which a cell must exceed to employ 9-point transmissibilities.
MINPVV	Sets the threshold pore volume which a cell must exceed or it will be made inactive.
MULTPV	Specifies the cell pore volume multipliers.
MULTREGP	Sets a pore volume multiplier for a specific region.
ORTHERRO	Specifies a threshold orthogonality error to employ 9-point transmissibilities.
ORTHERRX	Specifies a threshold orthogonality error for interfaces in the X-direction to employ 9-point transmissibilities.
ORTHERRY	Specifies a threshold orthogonality error for interfaces in the Y-direction to employ 9-point transmissibilities.
ORTHERRZ	Specifies a threshold orthogonality error for interfaces in the Z-direction to employ 9-point transmissibilities.
TBASIS	Specifies the basis for representing the permeability tensor.
TRANGL	Specifies the global-local connections and associated transmissibilities for an unstructured local grid refinement.
WCOL	Controls the coloring of grids, needed for the WARP linear solver.

PROPS section

ACFDET	Acentric factor for detailed components
DETAILVD	Sets initial conditions of detailed components.
HEATVAPS	Defines a heat of vaporization at the standard temperature, for use with the thermal option.
LUMPING	Describes make-up of lumped components.
MWDETAIL	Molecular weights for detailed composition.
OMEGAADE	As OMEGAA but for detailed composition.
OMEGAASD	As OMEGAAS but for detailed composition.
OMEGABDE	As OMEGAB but for detailed composition.
OMEGABSD	As OMEGABS but for detailed composition.

PCRITDET	Critical pressures of detailed components.
PCRITSDE	Critical pressures of detailed components at surface conditions.
PEDERSEN	Allows use of Pedersen viscosity calculation.
PMAX	Sets range of pressures to check total compressibility of the hydrocarbon system.
PPCWMAX	Limits the capillary pressure used in the SWATINIT keyword.
SWATINIT	Defines the initial water saturation.
TCRITDET	Critical temperatures of detailed components.
TCRITSDE	Critical temperatures of detailed components at surface conditions.
TZONE	Scales end points in the transition zones.
VCRITDET	Critical volumes of detailed components.
VCRITSDE	Critical volumes of detailed components at surface conditions.
ZCRITDET	Z-factors of detailed components.
ZCRITSDE	Z-factors of detailed components at surface conditions.

REGIONS section

`TRACKREG` Allows the user to define the regions for the enhanced TRACK option.

SOLUTION section

`DATUMR` Allows input of the datum depth by FIP region.

`FIPSEP` Allows different separators (used in the calculation of the in-place oil and gas volumes) to be assigned to different fluid-in-place regions.

`GPTABLE3` Defines a gas plant table with oil, NGL and gas recoveries.

`OLDEQUIL` Causes the grid to be equilibrated using non-tilted block equilibration with the number of layers specified in item 9 of the `EQUIL` keyword.

`RSW` Allows input of the initial CO_2 concentrations in `CO2SOL` cases with enumerated initial conditions.

SUMMARY section

`COP(R,T)L` and `C(W,G)(P,I)(R,T)L`

Completion production and injection rates and totals for oil, water and gas. The completion is defined by specifying a connection within it.

`CWCTL,CGORL,CWGRL`

Completion watercut, gas-oil and water-gas ratios. The completion is defined by specifying a connection within it.

`(F,G,W)CM(P,I)(R,T)`

Field, group and well hydrocarbon component molar production and injection rates and totals.

`(F,G,W)TM(R,T,F)`

Traced hydrocarbon mass rate, mass totals and molar fractions.

`(F,G,W)T(V,T)(L,V)`

Traced hydrocarbon volume rates and volume totals for liquid and vapor phases.

`S(O,W,G)(IM,RM)(R,T)`

Oil, water and gas import and removal rates and totals to or from a segment in a multi-segment well (see keyword `WSEGEXSS`).

`SH(IM,RM)(R,T)`

Enthalpy import and removal rates and totals to or from a segment in a multi-segment well (see keyword `WSEGEXSS`) for the Thermal option.

`STEM`, `SENE`, `SHFR`, `SSQU`

Temperature, energy, enthalpy flow rate and steam quality in a segment in a multi-segment well, for the Thermal option.

`(F,G)(O,W,G)PP` and `(F,G)(W,G)PI`

Group and field production and injection potentials.

`(F,G)MWP(T,R,A,U,G,S,V,P,O)`

Counts of all production wells in the field (F) or those subordinate to a given group (G). The total number of production wells may be counted (T) or the count may be of flowing wells (R), abandoned wells (A), unused wells (U), wells under group control (G), wells under their own surface rate control (S), wells under reservoir voidage control (V), wells under pressure control (P) or wells under their own oil rate control (O).

(F,G)MWI(T,N,A,U,G,S,V,P)

Counts of all injection wells in the field (F) or those subordinate to a given group (G). The total number of injection wells may be counted (T) or the count may be of flowing wells (N), abandoned wells (A), unused wells (U), wells under group control (G), wells under their own surface control (S), wells under reservoir voidage control (V) or under pressure control (P).

(F,G)MWD(R,T)

Count of the number of drilling queue events that have occurred in the field or a given group in a timestep, and the total number to date.

(F,G)MWW(O,T)

Count of the number of workover events that have occurred in wells subordinate to a given group in a timestep, and the total number to date.

FMWSET Requests the complete set of well count mnemonics for all production and injection wells, together with the drilling queue and workover event mnemonics.

GMWSET Requests the complete set of well count mnemonics for all production and injection wells subordinate to a given group, together with the drilling queue and workover event mnemonics.

WMCON Reports the number of open connections for a given well.

(G,W)(O,W,G,V)PGR

Group and well production guide rates for oil, water, gas and reservoir voidage.

(G,W)(W,G)IGR

Group and well injection guide rates for water and gas.

(F,G)STP(R,T)

Field and group steam production rate and totals (Thermal option).

(F,G)OSRC

The cumulative oil-steam ratio (Thermal option).

(F,G)SORC

The cumulative steam-oil ratio (Thermal option).

GPR, GPRG, GPRW

Group or node pressures in the production and gas and water injection networks.

GPRB, GPRBG, GPRBW

Pressure drop along the group or node's outlet/inlet branch in the production and gas and water injection networks.

GALQ ALQ in the group or node's outlet branch in the production network.

G(O,W,G,L)PRNB

Oil, water, gas and liquid flows along the group or node's outlet branch in the production network.

`G(W,G)IRNB`

Water and gas flows along the group or node's inlet branch in the water/gas injection network.

SCHEDULE section

ACTIONG	Initiates a set of keywords to be processed when a group satisfies a specified condition.
ACTIONR	Initiates a set of keywords to be processed when a fluid-in-place region satisfies a specified condition.
ACTIONS	Initiates a set of keywords to be processed when a segment of a multi-segment well satisfies a specified condition.
ACTIONW	Initiates a set of keywords to be processed when a well satisfies a specified condition.
BRANPROP	Defines the branches in the 'extended' network.
COMPLMPL	Lump well connections together into completions for simultaneous closure in automatic workovers when the connections are located in local grid refinements.
COMPLUMP	Lump well connections together into completions for simultaneous closure in automatic workovers.
DRILPRI	Sets the coefficients that define the default priority formula for the prioritized drilling queue.
GNETINJE	Defines the structure of the 'standard' injection network.
GPTABLE3	Defines a gas plant table with oil, NGL and gas recoveries.
GRUPNET	Defines the structure of the 'standard' production network.
GTMULT	Multiplies a group production or injection rate target or limit by a specified factor.
GUIDERAT	Specify a general formula for calculating production guide rates used for group control.
NETBALAN	Sets parameters for the network balancing calculation.
NGASREM	Removes gas from a node in the 'extended' network.
NODEPROP	Defines node properties in the 'extended' network.
NWATREM	Removes water from a node in the 'extended' network.
WDRILPRI	Puts wells into the prioritized drilling queue and defines their drilling priority values.
WELOPENL	Shuts or reopens wells or well connections in local grids, without having to specify the rest of the well control data or connection data.
WELPI	Sets the productivity/injectivity index (P.I.) value of one or more wells to a specified value.
WINJEDET	Describes detailed composition of a lumped fluid for an injection well.
WNETDP	Applies an additional pressure drop between the tubing head of a well and the corresponding network node of its group.
WSEGEXSS	Imports/removes fluid in a multi-segment well from/to an external source/sink.
WSEGHEAT	Assigns a heat transfer coefficient to a segment in a thermal multi-segment well.
WSEGLABY	A labyrinth device may be modeled with the same functionality as E100.
WSEGVALV	A homogeneous sub-critical valve may be modeled with the same functionality as E100.
WTMULT	Multiplies a well control target or limit by a specified factor.

Altered keywords for 2001A

ECLIPSE 100

RUNSPEC section

RPTHMD	New item 6 allows the output of group and field derivatives for the Gradient Option.
TRACERS	New item 6 allows the user to control the maximum number of non-linear iterations used by the tracer solver.

GRID section

CARFIN	A parent local grid can now be specified.
GRIDFILE	The default has changed such that the default grid file output uses the extensible grid file.
PARAOPTS	Some additional options have been added.

PROPS section

PMAX	New items 3 and 4 apply to extend the range of pressures used to check total compressibility of the hydrocarbon system.

SOMGAS and SOMWAT are now scaled in end-point scaling cases.

SOLUTION section

GRADRESV	Allows output of gradients of porosity and phase densities.

SCHEDULE section

ACTIONG	Liquid and reservoir volume rates and totals, and the gas to liquid ratio are new triggering quantities.
ACTIONW	Liquid and reservoir volume rates and totals, and the gas to liquid ratio are new triggering quantities.
GRADRESV	Allows output of gradients of porosity and phase densities.
GTMULT	Group injection surface and reservoir volume rate targets and limits may now be multiplied by a user-specified factor.
RPTRST	It is now possible to reduce the restart file size in cases with large numbers of wells where a few wells have many connections. See the COMPRESS mnemonic. In a gradient run, with 'PORO' requested in the RPTRST keyword, the effective porosity will now also be output under the heading, 'PORO_EFF'. Note this will only be the case when 'PORO' is also requested using the GRADRESV keyword.
WBHGLR	New item 8 to remove any cutbacks when the well is worked over.
WCUTBACK	New item 14 to remove any cutbacks when the well is worked over.
WELDRAW	New item 5 to specify whether the drawdown limit applies to the maximum drawdown in the well's connections or to their PI-weighted average.

`WELSEGS` Two items have been appended to each record, allowing the user to specify the X and Y coordinates of the segment nodes. These are stored on the restart or graphics files to enable graphics applications to produce plots of the well trajectory. The data is not used for any other purpose and may be left defaulted if trajectory plots are not required.

`WVFPEXP` New item 4, to prevent individual wells switching from rate control to THP control while they are constrained by a low rate limit to operate on the unstable side of their VFP curve. This item replaces switch 73 in the `OPTIONS` keyword.

Global keywords

OPTIONS	The keyword may now only be entered in the RUNSPEC and SCHEDULE sections. Switch 73 is no longer in use.Use item 4 of keyword WVFPEXP instead. Switch 79 can be used to restore the pre-2001A behavior of the STONE1 model in end-point scaling cases. Switch 80 has been superseded by the HYMOBGDR keyword. New switch 81 restores pre-2000A behavior with the Polymer Flood model. New switch 82 removes the restriction on updating grid block sizes in a local grid refinement.

ECLIPSE 300

RUNSPEC section

EQLOPTS	Can now be used with the MOBILE option.
PARALLEL	The ECLIPSE 100 option SERIAL is now available (this turns on the WARP linear solver).
REGDIMS	New item 5 to define the maximum number of regions for the enhanced TRACK option
ROCKCOMP	Item 1 can now take a new option BOBERG which allows hysteretic dilation as well as compaction.
TRACK	New 'REGONLY' option for item 1, when only user defined regions are required in the TRACK option.

GRID section

GRIDFILE	The default has changed such that the default grid file output uses the extensible grid file.
MPFA	Can now be applied to individual local grids between the CARFIN and ENDFIN pairs.

PROPS section

ROCKOPTS	New item 4 specifies the initial rock compressibility for the ROCKCOMP hysteresis option HYSTER and the new cyclic compaction and dilation option BOBERG.
ROCKTABH	Data from this keyword can now be used to model cyclic compaction and dilation with the BOBERG option in ROCKCOMP.

REGIONS section

RPTREGS	There is a new keyword, TRACKREG, to allow output of the region for the enhanced TRACK option.

SOLUTION section

AMF	The AMF keyword can now be used to set the initial solution in a CO2SOL case as well as the GASWAT case.
OUTSOL	Allows output of total compressibility using the 'TOTCOMP' mnemonic. In addition, sub-grid and coloring data, used by the WARP linear solver, can be output using the 'SUBG' and 'COLR' mnemonics respectively.
XMF, YMF, ZMF	These keywords can now be used within a BOX and the data will be normalised if the component sums to within 10% of unity.
ZMF	The ZMF keyword can now be used with the EQUIL keyword to overwrite the equilibrium hydrocarbon composition.
EQUIL	The behavior of items 4 and 9 has changed.

SCHEDULE section

CECON	The last four arguments of the keyword have been enabled for use by ECLIPSE 300. Limits may now be placed on the minimum oil and gas flow rates in connections. Also, flags may now be set to control whether the economic limits apply to stopped wells and if a connection breaks a limit, whether or not all connections below it are additionally shut.
COMPDAT	The pressure equivalent radius for a connection may now be specified in item 14 of this keyword.
COMPSEGS	One item has been added to specify the length of well in thermal contact with the completion cell. This item is only used in the Thermal option.
DGRDT	Two items have been appended, allowing the user to specify both the smallest permitted lower bound and the largest permitted upper bound on the fractional change in guide rate between the current time step and the previous time step.
GPMAINT3	New item 14 to allow an alternative interpretation of the reinjection fraction in this keyword. In this alternative the fraction limits the amount of produced gas that will be used for reinjection, with any additional injection gas required for pressure maintenance being supplied from the make-up gas stream. Any unused produced gas is available for fuel or sales usage.
GRUPGR	The new type POTN in item 4 allows the group injection and production guide rates to be set to the appropriate sum of the group's children's potentials.
GRUPINJE	The new control mode NONE in item 4 instructs no immediate control of a group's injection rate.
GRUPPROD	The new control mode NONE in item 2 instructs no immediate control of a group's production rate.
LSCRITS	Now controls convergence criteria of sub-grids as well as of LGRs.
LSCRITL	Retired in 2001A
NODEPROP	New items 6 and 7 to specify the corresponding group name and the network type.
OUTSOL	Allows output of total compressibility using the 'TOTCOMP' mnemonic. In addition, sub-grid and coloring data, used by the WARP linear solver, can be output using the 'SUBG' and 'COLR' mnemonics respectively.
PICOND	New item 4 activates the Velocity Dependent Relative Permeability option within the Generalized Pseudo-Pressure option.

RPTSCHED Allows output of total compressibility using the 'TOTCOMP' mnemonic

WCUTBACK New item 14 to remove any cutbacks when the well is worked over.

WEFAC Item 3 is now recognized by ECLIPSE 300, to specify whether the full flow or the time-averaged flow of the well should be passed into the Extended Network.

WELDRAW New item 5 to specify whether the drawdown limit applies to the maximum drawdown in the well's connections or to their PI-weighted average.

WELOPEN Items 6 and 7 have been enabled to define a range of completions within which all connections may be opened or shut.

WELSEGS Two items have been appended to each record, allowing the user to specify the X and Y coordinates of the segment nodes. These are stored on the restart or graphics files to enable graphics applications to produce plots of the well trajectory. The data is not used for any other purpose and may be left defaulted if trajectory plots are not required.

Three items have been appended to describe heat conduction in the tubing or casing for Thermal Option runs. The items specify the cross-sectional are of the tubing/casing, the heat capacity and the conductivity.

WELSPECS The drainage radius for productivity/injectivity index calculations may now be specified in item 7 of this keyword.

A crossflow ability flag may now be specified in item 10 of this keyword. If this is set at 'NO', the connections will act as one-way valves, preventing flow if there is a reverse drawdown.

An alternative, segmented method for calculating the density of the fluid in the wellbore may now be selected by setting item 12 of this keyword to 'SEG'. This density is used to calculate the hydrostatic head between the well's bottom hole reference depth and each of its connections.

WPIMULT Items 6 and 7 have been enabled to define a range of completions within which the transmissibility factors of all connections are multiplied by the specified factor.

WPIMULTL Items 7 and 8 have been enabled to define a range of completions within which the transmissibility factors of all connections are multiplied by the specified factor.

WSEGPROP Three items have been added that modify heat conduction in the tubing or casing for thermal runs. The items specify the cross-sectional area of the tubing/casing, the heat capacity and the conductivity.

Global keywords

OPTIONS3 The keyword may now only be entered in the RUNSPEC and SCHEDULE sections.

- New switch 43 reverts to a number of pre-2001A well model treatments.
- New switch 44 allows the user to change the amount of damping in the calculation of explicit hydrostatic heads (both segmented and averaged).
- New switch 45 allows the user to select whether or not a predictor is to be used when solving THP controlled wells and which type of scanning algorithm will be used should the well prove difficult to solve (usually close to the point when it is unable to flow).

- New switch 46 sets the number of iterations within a timestep in which a well can be revived after being closed because it cannot operate at the reservoir conditions of the previous iteration.
- New switch 47 controls the number of network balancing iterations in which a well that has just died may be revived.
- New switch 49 activates prototype group control logic which may help in problematic runs when wells and/or groups have significant flows that are unavailable for higher group control.
- New switch 51 controls the frequency of recalculating the well potentials during the network coupling iterations, which depend on the THP limits obtained from the network.
- New switch 52 reverts to the pre-2001A method of calculating horizontal thermal transmissibilities.
- New switch 53 reverts to the pre-2001A method of handling 3-phase relative permeabilities in the Generalized Pseudo Pressure option.

Developments in 2000A

New facilities for 2000A

ECLIPSE 100

Grid data

- An alternative transmissibility option based on upscaling half-block permeabilities is now available when the HALFTRAN keyword is used. This facility relies on half block upscaling to be used in FloGrid. See "Transmissibility Calculations" on page 55-1 of the "ECLIPSE Technical Description" for more information.
- The Vertical Equilibrium option now accounts for radial geometries.
- Generalized input of numerical aquifer non-neighbor connections can now be entered with the AQUNNC keyword.

Properties and equilibration

- When using the initial fluid mobility correction with the Hysteresis option active (with the flags MOBILE in keyword EQLOPTS and HYSTER in keyword SATOPTS), it is now possible to modify the imbibition as well as the drainage critical end points during initialization, to preserve the correct volumes of mobile fluid in place for both imbibition and drainage processes. Previously, only the drainage critical end points were modified to achieve the mobile fluid correction. See keyword EHYSTR.
- When modeling oil-wet systems with the End Point Scaling option activated, it is advantageous to be able to scale the water-oil capillary pressure at the maximum water saturation (using keyword PCW) rather than at the entry (connate) water saturation. An OPTIONS keyword switch (74) has been introduced to cater for this situation. Of rather less interest, but for completeness, it is possible to scale the gas- oil capillary pressure (keyword PCG) at connate rather than maximum gas saturation using this option switch. If OPTIONS (74) is non-zero and divisible by 2 then the water-oil capillary pressure is scaled at the maximum water saturation; and if OPTIONS (74) is non-zero and divisible by 3 then the gas-oil capillary pressure is scaled at the connate gas saturation.
- The keyword NOWARNEP suppresses all warning messages about (potentially) inconsistent end points.
- It is now possible to modify the way in which the initial relative permeability table critical saturations are determined by using the TOLCRIT keyword. Normally, the critical water saturation S_{wcr} is set to the value of S_w in the last table entry (in SWOF or SWFN) for k_{rw}, for which $k_{rw} \le 1.0 \times 10^{-6}$. The user can override this by entering TOLCRIT such that the critical saturation is the value of S_w in the last table entry where $k_{rw} \le TOLCRIT$, and similarly for S_{gcr}, S_{owcr} and S_{ogcr}. This will only be used in pathological cases involving end point scaling (keyword ENDSCALE), and may be useful for comparison with certain other simulators.
- The maximum pressure used to check the total compressibility of an oil-gas system can now be increased by setting the second item in the PMAX keyword. This would be useful, for example, to verify whether PVTO table extrapolations on the saturated bubble point line above the maximum entered (R_s, P_{bub}) are likely to cause difficulty during a simulation.

- The warnings for total compressibility have been improved to account for unphysical extrapolated values while checking the compressibilities in the pressure range spanned by the initial PVTO, PVDG, PVTG and PVDO tables. The explanation in the technical description has been improved; see "Total Compressibility Checks" on page 53-1 of the "ECLIPSE Technical Description".
- It is now possible to use the SGCWMIS keyword in MISCIBLE cases as well as in SOLVENT runs.
- When the Temperature option is active, the initial reservoir temperature can be specified using RTEMPA as an alias for RTEMP.
- Generalized input of analytic aquifer connections can now be provided by the AQANNC keyword.
- It is possible to switch off the Hysteresis option by adding the NOHYST keyword to the RUNSPEC section. Any IMBNUM values entered in the REGIONS section and imbibition endpoints in the PROPS section will then be ignored.
- The reporting of PVT table extrapolations has been improved. Setting the flag in EXTRAPMS greater than 3 will result in diagnostic output to the print file of the (P_o, R_s) values for the worst case extrapolations on the saturated and undersaturated portions of the PVTO tables. Similarly for (P_g, R_v) lookups in the PVTG table. This will report the cells where the most extrapolation occurs and report the type of extrapolation in the PVTO or PVTG tables, depending on whether the hydrocarbon is saturated or undersaturated.

Well model

- Wherever a well name root can be entered in the SCHEDULE section well keywords, a well list name may be entered instead as an alternative method of specifying several wells at once. These well list names must begin with an asterisk (*). A well list comprises a set of well names; there is no limit to the number or length of the well lists, and they may be changed at any time in the simulation, but each well cannot belong to more than one list at a time. The new keyword WLIST provides a set of facilities for constructing and managing well lists.
- A new option MAXR is available for sales gas control (see keyword GCONSALE) in the set of actions that may be taken when the sales gas rate exceeds the upper limit because the available injection capacity is insufficient to re-inject all the surplus gas. This option switches the group to gas production rate control, but also removes the re-injection fraction limit and continues to recalculate the production rate target to maintain the sales target according to the previous iteration's injection rate. The option thus maximizes the production rate by re-injecting as much gas as possible; a subsequent increase in the injection capacity will allow the production rate to increase correspondingly.
- The control of group targets/limits by prioritization has been extended to allow for different criteria to be used to decide which wells to close depending on which group rate limit is being exceeded. See keywords GCONPRI, WELPRI and PRIORITY.
- • The well block average pressures calculated for output to the Summary file (see keyword WPAVE) may now be corrected to a reference depth which is different from the well's BHP reference depth. A reference depth may be entered for each well with keyword WPAVEDEP, the default being the well's BHP reference depth.
- The NUPCOL keyword can now be entered in the SCHEDULE section as well as the RUNSPEC section.
- It is now possible to use oil injectors in API tracking cases. See the WAPI keyword.

Output

- Entering the mnemonic NOTHING in any of the RPTGRID, RPTPROPS, RPTREGS, RPTSOL and RPTSCHED keywords switches the output off for that keyword.
- The mnemonic AQUTAB in RPTPROPS outputs the Carter-Tracy influence function tables to the Print file.
- The mnemonic FIPOWG in RPTREGS outputs the FIPOWG region data to the Print file.
- Numerical aquifer performance data can now be obtained from SUMMARY section mnemonics ANQR and ANQT which represent inflow rate and cumulative inflow, and ANQP for the water volume weighted average aquifer pressure. The corresponding field quantities are FNQR and FNQT.
- When using the Killough capillary pressure Hysteresis Model, summary quantities can now be reported for the departure saturations of water and gas (BWSHY, BGSHY) and for the maximum wetting saturation for water capillary pressure hysteresis (BWSMA) and the trapped gas saturation for gas capillary pressure hysteresis (BGSTRP).
- The mnemonics PCOW and PCOG in keyword RPTRST will output the capillary pressures to the Restart file.
- The SAVE keyword can now be used in the SCHEDULE section, with no arguments, to request a restart record to be written to the Save file.

Performance

- Scratch files for processing non-neighbor connections within the GRID and EDIT sections have been removed on UNIX systems. This will show performance improvement in pre-processing datasets with large numbers of NNCs and local grids. The OPTIONS keyword switch 78 restores the scratch files where not enough virtual memory is available to process the NNCs dynamically.

Miscellaneous

- A facility has been added to import arrays from general binary files such as Initial and Restart files from a previous simulation run. The GETDATA keyword enables grid array data from an initial or restart file to be imported as an array with a new name into the current run. For example, it can be useful to import SWCR from a previous Initial file into the current run, or to import SWAT from a previous restart file as SWATINIT into a new case. In principle this is similar to the GETSOL keyword in ECLIPSE 300 but more flexible.
- The INCLUDE keyword now accepts file names up to 132 characters in length.
- Single phase tracers are solved, by default, using the cascade algorithm. It is now possible to force ECLIPSE 100 to use the linear solver algorithm for single phase tracers by specifying the NOCASC keyword in the RUNSPEC section.
- The Vertical Equilibrium Model can now be used in radial geometries (as well as with radial local grid refinements).

Environmental tracers

- If the Surfactant model or Environmental Tracer option is activated, there is now the facility to define partitioned tracers. These are tracers that exist in two phases, not necessarily two hydrocarbon phases. The partitioning between the two phases is governed by user input tables. See the keywords PARTTRAC, TRACER, TRACERKP and TRKPF for more details.

Flux boundary option

- The active tracer concentrations (API, BRINE, FOAM and SURFACT tracers) are now exported to the flux file in a DUMPFLUX run when the pressure flux boundary condition is employed. These will be used in the reduced USEFLUX run to account for the influx or efflux of tracer from outside the pressure boundary. Passive tracers are not included on the flux file, but may be added in the reduced flux run.

Gas field operations model

- It is possible to specify contract periods which are less than a year, for example to review the DCQ on a quarterly basis. The keyword GASPERIO may be used instead of GASYEAR to advance the simulation over a number of contract periods. A contract period can be any number of months that is a factor of twelve, i.e. 1, 2, 3, 4, 6 or 12.

GI pseudo-compositional model

- Dew point and bubble point pressures can now be output when running the GI model. The multipliers on saturated R_s, R_v, B_o and B_g can also now be reported. See the RPTSOL, RPTRST and RPTSCHED keywords.
- Extrapolation warnings for PVT tables (PVTO, PVTG) are now reported when using the GI-model (see keyword EXTRAPMS). Extrapolation messages are also issued when extrapolation occurs for (P, GI) in any of the GINODE, RSGI, RVGI, BOGI, BGGI or GIALL keywords.

Gradient option

- It is now possible to obtain gradients of the solution with respect to rock compressibility parameters using the HMROCK keyword, and gradients with respect to rock compaction modifiers using the HMROCKT keyword. Cumulative modifiers on rock compaction and compressibility parameters during the regression using SimOpt can be supplied using the HMMROCK, HMMROCKT and HMRREF keywords. See also the HMDIMS keyword.
- Gradients can now be obtained with respect to multipliers on the transmissibilities between MULTNUM regions. See the HMMULRGT keyword. Modifiers for inter-region transmissibility can also be specified using the HMMMREGT keyword. See also the HMDIMS keyword.
- Gradients with respect to numerical aquifer porosity, permeability and transmissibility multipliers for aquifer-grid connections can now be obtained using the HMAQUNUM keyword. Modifiers for numerical aquifers can also be specified using the HMMLAQUN keyword.
- It is now possible to request gradients of RFT observations for nominated wells, using the GRADRFT keyword. See also the RPTHMW keyword. This is used by SimOpt to match against observed RFT data.
- The cumulative number of linear iterations for each gradient parameter can now be output to the Summary file using the HSUMLINS keyword in the SUMMARY section.
- The Gradient option is now compatible with the Wellbore Friction option.
- The Gradient option is now compatible with the Multi-segment Well option.
- It is now possible to request gradients of cumulative production totals (see the GRADWELL and RPTHMD keywords).

Multi-segment wells

- Specific segments can be defined to represent a flow limiting valve, which imposes a pressure drop that increases sharply when the flow rate exceeds a specified limit. These may be used to limit the production of associated water or gas in a section of tubing, or to control the distribution of production/injection between branches of a multilateral well, or to prevent branch-to-branch crossflow. See keyword `WSEGFLIM`.

Local grid refinement

- Unstructured (Pebi) grids can now be run as local grid refinements. See keywords `EXTFIN`, `EXTHOST`, `EXTREPGL`, `PEBI`, `TRANGL`, `SOLVDIMS` and `SOLVNUM`. These are generally expected to be supplied by another program and will not be modified by the user.
- Radial local grids can now be used in runs where the global grid is run with the Vertical Equilibrium model.
- The wellhead location can now be defaulted in `COMPDATL`. If the I and J locations are defaulted, they are set to the wellhead I and J indices provided that the current local grid is the same as that used in `WELSPECL`, otherwise the I and J indices must still be specified for wells completed in different local grids from that specified in `WELSPECL`.

Parallel option

- `DUMPFLUX` is now available with the Parallel option.
- Horizontal pinchouts are now permitted with the Parallel option.
- In place local grid refinements are now available with the Parallel option.
- Non-neighbor connections which cross domain boundaries are now allowed with the Tracer Tracking and Parallel options.
- Auto-refinement is now compatible with the Parallel option.
- Import files can now be used with the Parallel option.
- The Parallel option now produces coarsened grids which are identical to those produced in a serial run.
- LGRs can be used in conjunction with tracers.

Polymer flood model

- Mnemonic RK in `RPTRST` reports the water relative permeability reduction factor to the restart files. The mnemonic VELOCITY reports the water flow velocity for shear thinning.

Surfactant model

- A new feature in the Surfactant Model allows you to model the change in wettability of the rock as a function of the concentration of adsorbed surfactant. The immiscible relative permeabilities (and capillary pressure) are then taken to be a weighted average of the user supplied oil-wet and water-wet saturation functions. The feature is enabled by using the `SURFACTW` keyword in the `RUNSPEC` section. The table numbers to use for the water-wet saturation look up and the fraction of oil-wet to water-wet to use are input with the `SURFWNUM` keyword in the `REGIONS` section, and the actual tables giving the fraction to use as a function of concentration of adsorbed surfactant are entered with the `SURFADDW` keyword in the `PROPS` section.

- Diffusion can now be applied to a surfactant if the Environmental Tracer option is active (Item 4 of the TRACERS keyword in the RUNSPEC section). This is done by using the TRDIFSUR keyword (that is the TRDIF keyword for the tracer called 'SUR' which is the surfactant).
- The SURFBLK mnemonic has been added to the RPTRST keyword for output of surfactant concentrations and related quantities.
- The requirement that the values of the saturated concentration of surfactant adsorbed by the rock formation input in the SURFADS keyword should be monotonic can be overridden by setting the 76th item in the OPTIONS keyword to be non-zero.
- If the Surfactant model or Environmental Tracer option is activated, there is now the facility to define partitioned tracers. These are tracers that exist in two phases, not necessarily two hydrocarbon phases. The partitioning between the two phases is governed by user input tables. See the keywords, PARTTRAC, TRACER, TRACERKP and TRKPF for more details.

ECLIPSE 300

Grid data

- The GRIDOPTS keyword can now be used to request additional options for processing the grid data. Only the second argument specifying the maximum number of MULTNUM regions in the GRID section should be used in this current release. All other arguments in this keyword are for ECLIPSE 100 only.
- The REGDIMS keyword can now be used to specify the maximum number of regions associated with miscellaneous keywords in the REGIONS section. It may be useful with the new GRID section keyword MULTNUM. Currently, only the first and fourth argument of the REGDIMS keyword can be used in ECLIPSE 300. Any other argument read in will be ignored.
- It is now possible to define the regions for applying inter-region transmissibility multipliers, instead of applying them to flux regions. This can be done using the MULTNUM keyword.
- The MULTREGT keyword can now be used to multiply the transmissibility between flux regions or regions set up using the MULTNUM keyword. The MULTREGT keyword can be used in both the GRID and SCHEDULE sections.
- The NONNC keyword is now available for compatibility with ECLIPSE 100.
- The facility to model a full tensor permeability has been added. Normally, the tensor permeability data would be supplied by an external program. ECLIPSE 300 uses a 19 or 27 point discretization scheme in 3D to account for the tensor nature of the permeability field. The permeability tensor components are supplied by keywords PERMX, PERMY and PERMZ for the diagonal components (or their aliases PERMXX, PERMYY and PERMZZ), and PERMXY, PERMYZ, and PERMZX for the off diagonal components. Only symmetric tensors are dealt with in the simulator at present. The calculation of transmissibility coefficients for tensor permeabilities is invoked using the MPFA keyword. It is still possible to use a conventional two point discretisation in conjunction with tensor permeabilities if required; ECLIPSE 300 will choose the components of the tensor along the coordinate axes to diagonalise. The MPFANUM keyword may be used to indicate regions in which the multipoint discretisation is to be applied. The multipoint transmissibilities can alternatively be imported into ECLIPSE 300 from the unstructured grid module in FloGrid by using the MPFNNC keyword. See "Tensor Permeability" on page 50-1 of the "ECLIPSE Technical Description" for further information.
- The MPFNNC keyword from ECLIPSE 100 is now available.

- The ADDZCORN and EQLZCORN keywords are now available in ECLIPSE 300 to modify the ZCORN data.

Properties and equilibration

- It is now possible to specify initial bubble point versus depth (PBVD) and initial

dew point versus depth (PDVD) in black oil runs for ECLIPSE 100 compatibility.

- The initial composition keywords RSVD, RVVD and RSWVD may now all be specified in the SOLUTION section for ECLIPSE 100 compatibility.
- It is now possible to modify the way in which the initial relative permeability critical table saturations are determined by using the TOLCRIT keyword. Normally, the critical water saturation S_{wcr} is set to the value of S_w in the last table entry (for SWOF or SWFN) for k_{rw}, for which $k_{rw} \le 1.0 \times 10^{-6}$. The user can override this by entering TOLCRIT such that the critical saturation is the value of in the last table entry where $k_{rw} \le TOLCRIT$, and similarly for S_{gcr}, S_{owcr} and S_{ogcr}. This will only be used in pathological cases involving end point scaling (keyword ENDSCALE). and may be useful for comparison with certain other simulators.
- It is now possible to use the initial grid block pressures as reference pressures for the ROCK keyword such that the initial ROCK is the initial reservoir pore volume. When using the ROCK keyword, setting item 2 of the ROCK keyword to STORE copies the initial equilibrated pressure to the overburden pressure (this functionality was already available with the rock compaction model; see keyword ROCKCOMP). This references the input pore volumes and porosities to reservoir conditions for compatibility with certain other simulators.

Well model

- Group production and injection controls may now be specified using the ECLIPSE 100 compatible keywords GCONPROD and GCONINJE, as an alternative to using GRUPPROD and GRUPINJE. When specifying gas injection controls with GCONINJE in a compositional run, the nature of the injected gas must also be specified with keyword GINJGAS. Well guide rates may also be set using the ECLIPSE 100 compatible keyword WGRUPCON, which also allows scaling of calculated guide rates.
- Individual group production rate targets and limits may be modified using the ECLIPSE 100 compatible keyword GRUPTARG.
- Group name roots can now be read for all group keywords except GRUPTREE.
- Wherever a well name root can be entered in the SCHEDULE section well keywords, a well list name may be entered instead as an alternative method of specifying several wells at once. These well list names must begin with an asterisk (*). A well list comprises a set of well names; there is no limit to the number or length of the well lists, and they may be changed at any time in the simulation, but each well cannot belong to more than one list at a time. The new keyword WLIST provides a set of facilities for constructing and managing well lists.
- A drilling queue for wells may now be defined using the keyword QDRILL. If a group (including the field) cannot meet its production target, the first production well in the drilling queue that is subordinate to that group, and not subordinate to any other group under the same mode of production control, will be opened automatically. Similarly, if a group cannot meet its injection target, the first injection well in the queue that is subordinate to that group, and not subordinate to any other group under injection control for the same phase, will be opened automatically.

- The time taken to drill a well from a drilling queue may be specified with the WDRILTIM keyword. With this keyword, the user may select whether a well drilled within a timestep flows from the beginning of the timestep, or flows only when fully drilled. The keyword also includes an option to prevent repeated premature opening of wells from the drilling queue.
- The rates and compositions of both oil and gas for up to nine stages of a separator may now be accessed using the appropriate summary mnemonics. (See the ECLIPSE 300 SUMMARY sub-section in "New keywords for 2000A".) Previously, the mnemonics reported gas data for the first three stages only. The gas from any stage of a separator may now also be used as a source of re-injection fluid. (See GINJGAS, WINJGAS, GRUPINJE, WELLINJE, WINJMIX and WINJORD.)
- You can select whether a well is to be shut or stopped when it violates an economic limit (WECON, GECON) or a maximum flow rate (GCONPROD, GCONPRI) or is unable to flow. See item 9 of keyword WELSPECS.
- The actions CON and +CON have been added to item 7 of GECON, allowing workovers to be performed if a group violates an economic ratio limit.
- When a well is defined with WELSPECS (or WELSPECL), its 'preferred phase' must be entered in item 6 (or 7). This is used to determine the worst offending connection or well when a group production rate limit is exceeded in GCONPROD or GCONPRI.
- The well block average pressures calculated for output to the SUMMARY file (see keyword WPAVE) may now be corrected to a reference depth which is different from the well's BHP reference depth. A reference depth may be entered for each well with keyword WPAVEDEP, the default being the well's BHP reference depth.
- In the well block average pressure calculation, the pressure of the inner block and its outer ring of neighboring block may now be averaged according to their pore volumes. This method of averaging pressure over grid blocks, used in the connection factor weighted average pressure calculation, may be activated by setting the first weighting factor in the WPAVE keyword to a negative value.
- The NUPCOL keyword can now be entered in the RUNSPEC section as well as the SCHEDULE section.
- Imbibition saturation table numbers may now be set for well completions using the COMPIMB keyword.
- By using the keyword VFPCHK, the checking and correcting of crossing curves in a VFP production table may be limited to crossings below a given BHP value. This allows a table to retain BHP values that are set artificially high to indicate conditions where the flow is supersonic or exceeds specified erosion velocity limits. These are legitimate, and serve to prevent the well from operating at these conditions.

Output and restarts

- When using the Rock Compaction option with an overburden pressure (keyword OVERBURD), the effective fluid pressure can be output using the mnemonic PRES_EFF in the OUTSOL keyword. The effective pore volume multiplier can be output with the mnemonic PORV_MOD.
- It is now possible to specify a NOSIM data checking run in the LOAD keyword and to specify the formatted/unformatted input flag in the LOAD keyword.
- Rock compaction data tables and overburden pressure tables can now be output to the Print file using the mnemonics ROCKTAB, ROCKTABH (for the Hysteretic Option), and OVERBURD in the RPTPROPS keyword.

- • The MULTNUM array can now be output to the Print file (under the RPTGRID keyword) and also output to the INIT file. In addition, information concerning the MULTREGT keyword can now be output to the Print file.
- For Dual Porosity runs, output in the Print file will refer to specific parts of the grid, namely, the matrix part and the fracture part.
- In black oil runs, the mnemonics RS, RSSAT, RV and RVSAT will output the R_s, saturated R_s, R_v and saturated R_v values when used as arguments to keywords RPTSOL, RPTSCHED and OUTSOL.
- If the SOF3 mnemonic is used the RPTPROPS keyword, the ECLIPSE 100 style 3- phase tabular output to the Print file appears in addition to the input tables.
- If the PVTW mnemonic is used in the RPTPROPS keyword, this results in tabular output of the water PVT data to the Print file.
- If the transmissibilities are overwritten (using the OVERT− keywords, for example OVERTX), or multiplied (using the MULT/MULTT keywords, for example MULTX), in the SCHEDULE section, this information can be output to the Print file using the mnemonic MULT in the RPTSCHED keyword. If a combination of both is used, then the final result will be output.
- A new file format, the extensible grid file (EGRID), can be used to transfer data to and from the pre- and post-processing programs. See the GRIDFILE keyword.
- Inter-block flows of hydrocarbon components can now be reported to the Print file using mnemonics FLOCn in the RPTSOL and RPTSCHED keywords (where n is an integer representing component n). The mnemonic FLOWAT requests output of inter-block water flow rates in the RPTSOL, RPTSCHED and OUTSOL keywords.
- The INSPEC and RSSPEC files now contain a list of descriptors for each output array and a list of descriptors for the unit system of all output quantities in the INIT and Restart files.
- Historical flowing rates and cumulatives at the field and group level may now be written to the Summary file (see keyword WCONHIST).
- Additional SUMMARY section keywords (NEWTFL, NNUMFL, NNUMST, NBAKFL) are available to output performance data for the flash calculation.

Performance

- A new linear solver has been implemented which accounts for local grid refinements in a more efficient way. The LSCRIT keyword sets convergence criteria for the linear solver. Keywords LSCRITL and LSCRITS provide control over sub-iterations for local grids. For more information, see the chapter on “Solution of the Linear Equations” on page 45-1 of the “ECLIPSE Technical Description”.
- A new way for controlling the choice of implicit cells within the AIM solution procedure is available (keyword AIMCON).
- A number of pre-set tunings are available in keyword TOPT.
- A new OPTIONS3 keyword switch (42) has been added which may help convergence in dual porosity cases.

Miscellaneous

• When the required memory is not available to process the dynamic parts of the simulation for setting up the Jacobian matrices and the linear solver, ECLIPSE 300 now counts up the total memory required in many cases and reports this total before aborting. There must already be sufficient memory allocated already to advance the simulation up to the end of the SOLUTION section, but the memory required to advance the timesteps is by far the largest memory request, especially in implicit cases. Once the total memory requested is known, the MEMORY keyword can be set if resources are available.

- A Brine Tracking option is available, to model the mixing of waters with different salt concentrations. See keyword BRINE.
- It is now possible to add tracers (but not the TRACK option) to a restart run where the base run did not use any tracers. However, it is still not possible to add extra tracers at the restart when the base case already used tracers

Local grid refinement

- Both the MINPORV and MINROCKV keywords can now be used for individual local grid refinements. These keywords and their data should be placed between the CARFIN or RADFIN and ENDFIN keywords for the respective local grid.
- The Dual Porosity option is now compatible with the use of local grid refinements.
- LGRs can be added to, or removed from, the grid during flexible restarts (keyword RESTART).

Multi-segment wells

• The Drift-flux Slip Model is now available. This can be selected for the entire well in item 7 of record 1 in keyword WELSEGS, and parameters may be adjusted for individual segments with keyword WSEGFMOD. With the Drift Flux Model, the phases may flow at different velocities, and at low flow rates countercurrent flow may occur, where the heavy and light phases flow in opposite directions.

- A variable pressure drop multiplier can now be specified in keyword WSEGMULT. The multiplier allows you to specify a scaling factor by which the frictional pressure drop calculated across a segment will be multiplied. The scaling factor may be either a constant or vary as a function of the water-oil ratio and gas-oil ratio of the mixture flowing through the segment's outlet junction. A variable scaling factor may be applied, for example, to progressively choke off a branch as its produced gas-oil ratio increases, or to model the operation of an adjustable flow control device whose purpose is to cut off production from completions with high water cuts. The value of the scaling factor can be output to the Summary file using the mnemonic SPRDM in the SUMMARY section.
- Specific segments can be defined to represent a flow limiting valve, which imposes a pressure drop that increases sharply when the flow rate exceeds a specified limit. These may be used to limit the production of associated water or gas in a section of tubing, or to control the distribution of production/injection between branches of a multilateral well, or to prevent branch-to-branch crossflow. See keyword WSEGFLIM.
- Multi-segment wells may now have a D-factor for flow-dependent skin (keyword WDFAC).
- Some of the iteration parameters for the Multi-segment well calculation can be controlled with the keyword CVCRIT. These are the maximum number of iterations to converge the well and convergence tolerances for molar densities, molar flow rate and energy flow rate. The pressure tolerance is the same as for the reservoir grid and can also be set with CVCRIT.

- Segment component flow rates (SCFRn) and segment saturation pressures (SPSAT) can be written to the Summary file.
- The facility in the ECLIPSE 100 Wellbore Friction option to output information which GRAF uses to produce specialized plots is also available with Multi-segment Wells (see "Graphical presentation of friction effects in horizontal wells" on page 58-10 of the "ECLIPSE Technical Description"). GRAF recognizes the presence of multi-segment wells, and will produce the same series of prompts as it does for friction wells.

Parallel option

- Multi-segment wells are now available.
- The flux options are now available:
  - `DUMPFLUX`: to create a flux file
  - USEFLUX: to use the flux file.
- The `COARSEN` keyword is now available.

Thermal option

- A new keyword `OILVISCC` allows the temperature dependence of the oil viscosity to be specified using standard correlations. Four correlations can be used, ASTM, Andrade formula, Vogel formula and a logarithmic relationship. Correlation coefficients can be entered directly, or values of temperature and viscosity can be entered, in which case the simulator will calculate the coefficients.
- A screening of oil, water and gas saturations can be undertaken if a problem occurs due to a negative saturation. This may happen, for example, with saturations from an ECLIPSE 100 simulation. If this is desired, set the 40th switch of keyword `OPTIONS3` to a value > 0.
- The steam production rate from a well can now be controlled (see keyword `WCONPROD`).
- In dead oil thermal runs, steam production can now be prevented by limiting the well BHP at a fixed amount above the saturated water pressure. This option can also be used in live oil thermal runs, but some steam may be produced at pressures above the water saturation pressure. (See keyword `WCONPROD`.)

Behavioral changes for 2000A

These may cause changes to the results of a simulation, or alter the form or content of the output.

ECLIPSE 100

Grid data

- The default behavior of `MULTREGT` (also `MULTREGD` and `MULTREGH`) has been altered when there are numerical aquifers. Transmissibilities for connections relating to numerical aquifers are now accounted for between `MULTNUM` regions. Setting the flag NOAQUNNC in item 2 of the `MULTREGT` keyword will not apply the transmissibility multipliers to numerical aquifer connections for back compatibility with pre-2000A versions.

- When using numerical aquifers (AQUNUM) and the Vertical Equilibrium (VE) options, the method of calculating the flows across the NNCs connecting the aquifer to the grid (specified by AQUCON) has been altered. The default is now to use an effective VEFRAC = 1.0 for the relative permeability calculation and VEFRACP = 1.0 for the capillary pressure calculation for the aquifer to grid water flows. Previously, an effective VEFRAC and VEFRACP both equal to 0.0 (pure rock curves for relative permeability and capillary pressures) were used in the 99A version. The VEFRAC and VEFRACP values can be controlled by the 12th and 13th items in the AQUCON keyword.
- The non-orthogonal pressure correction nine-point scheme controlled by the NOPC9 keyword has been deprecated.

Output and restarts

- Various defaults in the MESSAGES keyword have been increased for compatibility with ECLIPSE 300.
- The LOAD keyword has been modified to be more compatible with the ECLIPSE 300 behavior. The second data item can now be used to specify the report step to restart from, provided that the corresponding restart record is available in the SAVE file. The old format (without the report step data item) will still be recognized by ECLIPSE and interpreted as before, effectively defaulting the report step.
- The INSPEC and RSSPEC files are now output by default. Previously these files were only output if the INSPEC or RSSPEC keywords were specified. New keywords NOINSPEC and NORSSPEC can be used to disable INSPEC or RSSPEC output.
- Runtime monitoring is now on by default. This used to be switched on optionally by the MONITOR keyword. The NOMONITO keyword disables runtime monitoring.

Disking option

- The disking options have been deprecated.

Hysteresis

- The Capillary Pressure Hysteresis Model has been changed such that the capillary pressure will follow a scanning curve when the initial wetting phase saturation is below the critical value. This should have little effect when the drainage and imbibition curves meet at the connate saturation, but if the two curves have different capillary pressures at the connate saturation then a sensible scanning curve will be used. This is compatible with the default behavior in ECLIPSE 300. The pre-2000A behavior, in which the bounding imbibition curve is used instead of a scanning curve, may be restored by setting the 79th switch of the OPTIONS keyword.

Gradient option

- ECLIPSE no longer bans the use of active tracers (API, Brine and the Temperature Option or the Foam Model) in Gradient runs. In most cases the effect should be small and can often be neglected, so that it is still possible to perform a sensitivity run to screen redundant parameters or enlist a regression run. However, the gradients will not be fully accurate as the contributions of the tracer properties to the sensitivities are ignored. See “The Gradient Option” on page 24-1 in the “ECLIPSE Technical Description”.

Local grid refinement and coarsening

- The format of the COMPDATL keyword in ECLIPSE 100 is changed in the 2000A release, with the addition of the local grid name in item 2. This makes it compatible with the ECLIPSE 300 form of the keyword. It is now the same as the COMPDATM keyword in ECLIPSE 100, which is treated as an alias for COMPDATL. For back compatibility, ECLIPSE 100 will still recognize and read the old form of COMPDATL if this is entered. The new form of the keyword is distinguished by item 6 being an integer (it must not be defaulted), or alternatively item 2 being a character string which is not a pure integer, or item 7 being a character string beginning with a letter. If any of these conditions is true ECLIPSE 100 recognizes the keyword as the new form, otherwise it reads it as an old form of the keyword.
- The TOPS value derived for a coarsened cell is now the volume weighted average of the TOPS values for its constituent cells. Previously it was the value supplied for the representative cell. This is now consistent with the treatment of DEPTH values.

Multi-segment wells

- When using the Drift Flux Model, the adjustment of the drift velocity to allow for pipe inclination has been changed. The adjustment is now calculated from the formula of Hasan and Kabir (see “The drift velocity” in the “ECLIPSE Technical Description”), which gives a larger drift velocity in pipes deviated from the vertical. Also, the original model for oil-water slip has been replaced with the model proposed by Hasan and Kabir (see “Three-phase flow and oil-water slip” in the “ECLIPSE Technical Description”). It is possible to revert to the pre-2000A treatment by setting switch 77 in the OPTIONS keyword.

ECLIPSE 300

RUNSPEC section

- The behavior of DPCDT has changed:
  - The second argument of DPCDT no longer exists.
  - The limiting of the capillary pressure is only performed on explicit cells (The limiting can be applied to all cells if switch 38 of OPTIONS3 is set >0)
  - The DPCDT limiting is performed on the pressures associated with a phase. (Pre-2000A, the limiting was performed after table look-up; this behavior can be restored if OPTIONS3 switch 37 is set >0) This change only effects 3-phase cells.
- The NORSM keyword is deprecated.

Grid data

- •The SIGMA and SIGMAGD keywords have been replaced by SIGMAV and SIGMAGDV for ECLIPSE 100 compatibility. SIGMA and SIGMAGD now expect scalar data items and an error will result if array data is supplied.
- The DZMTRX keyword now expects scalar data items for ECLIPSE 100 compatibility. The DZMTRXV keyword (or its alias DZMATRIX) is still available to enter DZMTRX values on a cell by cell basis.

Non-Darcy flow

- The velocity dependent relative permeability Forchheimer models (see keywords VDKRO and VDKRG) have been fixed to be consistent with the Non-Darcy flow technical description and units (see "Non-Darcy Flow" on page 35-1 of the "ECLIPSE Technical Description"). The fixes will generate different results from previous version of ECLIPSE 300. One way to validate the fixes is to compare the velocity dependent relative permeability Forchheimer models (keywords VDKRO and VDKRG) to the standard Forchheimer model with a β factor (keyword VDFLOW). To accomplish this, set all parameters in keywords VDKRO and VDKRG to zero except the fifth argument (a_o and a_g) which should be set to the same value as the β factor for VDFLOW. In ECLIPSE 300, the velocity dependent Forchheimer models (keywords VDKRO and VDKRG) are the preferred models since effects such as relative permeability are treated implicitly, whereas explicit treatment is used in Forchheimer model with the β factor.

Properties and equilibration

- The 9th item in the EQUIL keyword has been altered for ECLIPSE 100 compatibility. Horizontal block averaging over 2*N layers for the more accurate initial fluid-in-place calculation is now selected by setting the 9th item to -N. Setting the 9th item to +N still implies horizontal block averaging over N layers for backward compatibility with pre-2000A versions. Tilted block integration is not at present available in ECLIPSE 300.
- The initial composition versus depth keywords RSVD, RVVD and RSWVD have been moved from the PROPS to the SOLUTION section for ECLIPSE 100 compatibility. These keywords can all still be entered equivalently in the PROPS section for back-compatibility.
- The calculation of the hydrocarbon-water relative permeabilities has changed. The pre-2000A method interpolated between the gas-water and oil-water curves, without scaling the end-points prior to interpolation. This interpolation was only performed if the Li-correlation indicated the current state was near the critical point. The 2000A method scales the end-points of the curves prior to interpolation. The pre-2000A behavior can be restored by setting OPTIONS3 keyword switch 41 > 0. The limits of the Li correlation, which specify when this interpolation occurs, can be entered using the LILIM keyword.

Well model

- The default treatment for using advanced gas during re-injection is a sequential model. In this model the advanced gas is used prior to the available produced gas for re-injection. If the re-injection requirements of a group or well can be met by the advanced gas stream alone, then no produced gas will be re-injected and all of it will remain available for fuel and/or sales usage. If the advanced and available produced gas streams are not sufficient for the re-injection requirements of a well, make-up gas, if specified, will be used in the normal manner. The advanced gas rate is added to a group's produced gas rate to form the available gas rate for reinjection subject to a group's fuel and sales gas commitments. The default mixing tank treatment of the 99A release may be used as an alternative by setting the 30th integer in the OPTIONS3 keyword.
- The injection calculation is now iterated a number of times in order to provide a more accurate solution in situations when the composition of the injected gas depends on the injection rate, such as when advance gas or make-up gas is present, or when the injection gas is made up of a mixture defined by WINJORD. The calculation is also iterated a number of times to provide more accurate availability limits for the case when two or more injection groups take their gas from the same source group and availability control has been requested (see keyword WAVAILIM). When advance gas or make-up gas or a WINJORD mixture is not present, and availability limits have not been requested, the injection

calculation is only executed once each Newton iteration. The keyword GCONTOL may be used to alter the default convergence tolerance and limit on the number of iterations. Setting the 36th integer in the OPTIONS3 keyword to 9 will cause the simulator to revert to the 99A treatment of the injection calculation (one iteration, regardless of the nature of the problem).

- When the source of injection gas is a single separator stage (see GINJGAS item 5), the treatment has been altered for cases where there are fuel or sales targets that take precedence over injection (see WTAKEGAS). Previously, the fuel and sales gas composition was that of the complete separator, whereas in reality the composition must change to reflect the fact that an amount of injection gas with the composition of a single stage has been removed. The new treatment allows for this change in composition, by performing the injection calculation first at a rate that is limited by higher priority fuel and sales requirements. It is possible to revert to the pre-2000A treatment by setting the 36th integer in the OPTIONS3 keyword to 10.
- The data format of keyword GPMAINT has been changed to be compatible with the ECLIPSE 100 keyword. Items 8 to 14 have been removed; use GINJGAS instead to specify the nature of the injected gas. Item 2 entries that specify the nature of the injected gas have also been removed. The old data format can still be read if the keyword name is changed to GPMAINT3.
- For compatibility with ECLIPSE 100, positive valued completion D-factors entered in COMPDAT or COMPDATL are now treated as if they apply to the well and will be scaled to a D-factor appropriate to the completion (see WDFAC for a description of this scaling). A D-factor may be entered directly and not scaled by setting it to a negative value. Alternatively, setting the 36th switch in the OPTIONS3 keyword to 2 will revert to the 99A treatment of completion D-factors.
- For compatibility with ECLIPSE 100, in the calculation that scales well D-factors to completion D-factors, the sum of transmissibility factors is now made over all currently open completions. Prior to this release, this sum was made over all completions regardless of whether they were open or closed. Setting the 36th integer in the OPTIONS3 keyword to 2 will revert to the 99A method of D-factor scaling.
- For compatibility with ECLIPSE 100, the D-Factor mobility multiplier in a crossflowing gas injector now applies to water phase as well as hydrocarbon phases. Setting the 36th integer in the OPTIONS3 keyword to 8 will revert to the 99A treatment, whereby this multiplier is only applied to the hydrocarbon phases.
- The well's preferred phase must be entered in item 6 of WELSPECS or item 7 of WELSPECL when this keyword is used to define a well.
- The end run flag in WECON and GECON now stops the run if the well (or all the producers in the group) have been shut/stopped for any reason, not just on violation of the minimum economic rate limit. Setting the 36th integer in the OPTIONS3 keyword to 3 will revert to the 99A treatment.
- The WREV option in WECON item 7 has been removed (it is interpreted as WELL). The WELL option now allows wells to be revived if requested in WTEST (with an E in item 3), to be consistent with ECLIPSE 100. Previously, wells closed due to ratio violations could only be revived if the WREV option was selected.
- The procedure for opening connections that were automatically closed in workovers, when a well is revived with the WTEST facility, has been changed to be compatible with ECLIPSE 100. Previously, closed connections were not re-opened when the well was revived. Setting the 36th integer in the OPTIONS3 keyword to 4 will revert to the 99A treatment.

- Well cumulative production histories are now incremented according to their actual production rates when they are no longer history matching wells. By using this cumulative in the calculation of group level cumulatives, a discontinuity is avoided in the group level cumulative when a production well switches from a history matching well to a standard well.
- For compatibility with ECLIPSE 100, a follow-on well (that is fully specified but shut) is now opened should its associated well be automatically closed for **any** reason apart from priority control. Previously, this action was limited to closure due to well economic limits. (See item 9 of the `WECON` keyword for more details.)
- If a well's bottom hole pressure reference depth is defaulted in `WELLSPEC`, `WELSPECS` or `WELSPECL`, its value is now set equal to the centre depth of the grid block containing the top-most connection in this well, as is the case in ECLIPSE 100. Setting the 36th integer in the `OPTIONS3` keyword to 5 will revert to the 99A treatment, whereby a defaulted bottom hole pressure reference depth is set equal to the centre depth of the grid block containing the first connection defined by the user for this well.
- For compatibility with ECLIPSE 100, when a well is automatically closed, item 9 in the `WELSPECS` keyword now determines whether the well is shut or stopped. The default action, which applies if the well is specified with the `WELLSPEC` keyword, is to shut the well. In the case when the well is closed because it is unable to flow, setting the 36th integer in the `OPTIONS3` keyword to 6 will cause the simulator revert to the 99A treatment, whereby the well will be stopped, regardless of what is specified in the `WELSPECS` keyword.
- In production well VFP tables, BHP values at 1.0e10 (the default value used to indicate supersonic or erosional flow in a VFPi pressure traverse calculation) that result in crossing curves are now not automatically corrected so that the curves no longer cross. Such BHP values have been made artificially high on purpose to distinguish regimes of supersonic or erosional flow, and the correction would remove this distinction and allow the well to operate under these conditions.
- The well block average pressures (see the WBP series of summary mnemonics) have been modified for the case of horizontal wells. The stencil of grid blocks around well completions is now perpendicular to the penetration direction of horizontal wells, as is the case in ECLIPSE 100. (See keyword `WPAVE`.)

Output

- Various defaults in the `MESSAGES` keyword have been increased for compatibility with ECLIPSE 100.
- Any errors encountered now force the simulator into `NOSIM` mode. In addition, if the number of errors is either greater than 10 (the new `MESSAGES` default) or a user-specified error limit set in `MESSAGES`, the simulator stops. This now provides the same error condition (level 5) handling as ECLIPSE 100.
- Runtime monitoring is now on by default. This used to be switched on optionally by the `MONITOR` keyword. The `NOMONITO` keyword disables runtime monitoring.
- The `INSPEC` and `RSSPEC` files are now output be default. Previously these files were only output if the `INSPEC` or `RSSPEC` keywords were specified. New keywords `NOINSPEC` and `NORSSPEC` can be used to disable `INSPEC` or `RSSPEC` output.

Performance

- The allocation of ROWSUM and COLSUM constraints has changed in the Nested Factorization solver (see "Solution of the Linear Equations" on page 45-1 of the "ECLIPSE Technical Description"). The default in fully implicit cases is now COLSUM instead of ROWSUM. This can be over-ridden using the 20th switch in the `OPTIONS3` keyword.

Local Grid Refinement

- The minimum pore volume for all local grid refinements has been changed from 1.0e-21 to 1.0e-6. A specific value can now be set via the `MINPORV` or `MINPV` keyword, which is entered between the `CARFIN` or `RADFIN` and `ENDFIN` keywords, otherwise the new default value will be used.

Multi-segment wells

- When a workover with an action '+CON' is performed on a multi-segment well, that is close the worst-offending connection and all below it, is now interpreted as meaning close the connection and all connections **upstream** of it. Thus instead of selecting the connections to close according to their depth, they are selected according to their length down the tubing—all connections further from the wellhead are closed. In multi-lateral wells, the branch topology is taken into account so that only branches subordinate to the worst-offending connection are closed off.

Thermal option

- The ECLIPSE 300 Thermal option keyword `TEMP` has been changed to TEMPI to avoid compatibility issues, as `TEMP` is used by the `ECLIPSE 100` Temperature option.
- The maximum temperature economic limit has been moved from item 11 of `WECON` to item 15, because items 11 to 14 already contain items used by ECLIPSE 100.

New keywords for 2000A

ECLIPSE 100

RUNSPEC section

`BIGMODEL`	Makes ECLIPSE allocate memory in 4-byte words rather than bytes. This allows large models requiring between 2 GBytes and 8 Gbytes of memory to be run with the standard ECLIPSE executable. (This is, of course, subject to any limits on memory allocation imposed by the user's machine or its operating system.)
`FULLIMP`	Forces the run to be fully implicit (for ECLIPSE 300 compatibility).
`IMPES`	Forces the run to be always IMPES (for ECLIPSE 300 compatibility).
`NOCASC`	Forces the linear solver tracer algorithm to be used for single phase tracers.
`NOHYST`	Disables hysteresis.
`NOINSPEC`	Disables the Initial Index file (`INSPEC`) output.
`NOMONITO`	Disables output for runtime monitoring.
`NORSSPEC`	Disables the Restart Index file (`RSSPEC`) output.
`PARTTRAC`	Activates the Partitioned Tracer Model.

	SOLVDIMS	Specifies the solver dimensions for an unstructured grid generated by FloGrid.
Surfactant Option	SURFACTW	The immiscible relative permeabilities and capillary pressure will be calculated as an average of oil-wet and water-wet saturation functions dependent upon the mass of adsorbed surfactant.

GRID section

	AQUNNC	Specifies non-neighbor connections explicitly for numerical aquifers.
LGR Option	EXTFIN	Specifies an unstructured local grid refinement.
LGR Option	EXTHOST	Specifies the global host cells for an unstructured local grid refinement.
LGR Option	EXTREPGL	Specifies the replaced global cells for an unstructured local grid refinement.
	HALFTRAN	Activates the half-block transmissibility upscaling option.
Gradient Option	HMAQUNUM	Activates the calculation of gradients for numerical aquifer parameters.
Gradient Option	HMMLAQUN	Specifies cumulative multipliers on numerical aquifer parameters.
Gradient Option	HMMMREGT	Specifies cumulative inter-region transmissibility multipliers.
Gradient Option	HMMULRGT	Activates the calculation of gradients with respect to inter-region transmissibility multipliers.
	PEBI	Specifies that an unstructured grid is being used.
LGR Option	SOLVDIMS	Specifies the solver dimensions for an unstructured local grid refinement generated by FloGrid.
	SOLVNUM	Specifies the map from user numbers to solver numbers for an unstructured grid generated by FloGrid.
LGR Option	TRANGL	Specifies the global-local connections and associated transmissibilities for an unstructured local grid refinement.

PROPS section

Gradient Option	HMMROCK	Specifies cumulative multipliers on rock compressibility.
Gradient Option	HMMROCKT	Specifies cumulative modifiers of rock compaction parameters.
Gradient Option	HMROCK	Activates the calculation of gradients with respect to rock compressibility multipliers.
Gradient Option	HMROCKT	Activates the calculation of gradients with respect to rock compaction parameters.
Gradient Option	HMRREF	Specify reference pressures for HMROCKT.
	NOWARNEP	Suppresses all warning messages associated with checking consistency of end points when the end point scaling option is activated.
	SGCWMIS	Input tables of critical gas saturation as a function of water saturation in MISCIBLE cases.
Surfactant Option	SURFADDW	Tables of mass of adsorbed surfactant vs. fraction of oil-wet and water-wet curves to use in calculating immiscible relative permeabilities.
	TOLCRIT	Controls the method of determining the table critical saturations in pathological end point scaling cases.
Environmental and Surfactant Options	TRACERKP	Tables giving the K(P) partitioning functions for partitioned tracers.

REGIONS section

Surfactant Option	SURFWNUM	Gives the table numbers for the water-wet saturation functions and the mass of adsorbed surfactant vs. fraction of oil-wet to water-wet curves, to use for each grid block.
Environmental and Surfactant Options	TRKPF	Region numbers for selecting which K(P) table to use for each partitioned tracer.

SOLUTION section

AQANNC	Specifies analytic aquifer connections explicitly.
RTEMPA	Specifies initial reservoir temperature as an alias for RTEMP.

SUMMARY section

	ANQR	Numerical aquifer influx rate.
	ANQT	Numerical aquifer influx total.
	ANQP	Numerical aquifer average water pressure.
	FNQR	Field influx rate for all numerical aquifers.
	FNQT	Field influx total for all numerical aquifers.
	BGSHY	Departure saturation from drainage to imbibition for gas capillary pressure hysteresis.
	BGSTRP	Trapped gas critical saturation for gas capillary pressure hysteresis.
	BWSHY	Departure saturation from drainage to imbibition for water capillary pressure hysteresis.
	BWSMA	Maximum wetting saturation for water capillary pressure hysteresis. This also represents the maximum wetting phase saturation used to determine scanning curves in relative permeability hysteresis, whether the Carlson or Killough Model is used.
Gradient Option	HSUMLINS	Outputs the total number of linear iterations for each gradient parameter.
	NOMONITO	Disables output for runtime monitoring.

SCHEDULE section

	DIMPES	This replaces the IMPES keyword in the SCHEDULE section of ECLIPSE 100, which took a different form from the current ECLIPSE 300 compatible IMPES keyword in the RUNSPEC section. However, ECLIPSE 100 will still accept the old IMPES keyword in the SCHEDULE section for backward compatibility.
	DIMPLICT	This replaces the IMPLICIT keyword in the SCHEDULE section of ECLIPSE 100. However, ECLIPSE 100 will still accept the IMPLICIT keyword in the SCHEDULE section for backward compatibility.
Gas Field Operations Model	GASPERIO	Advances the simulation over a number of contract periods which may be less than a year.
Gradient Option	GRADRFT	Requests gradient output of RFT quantities for nominated wells.
Gradient Option	HMWPIMLT	Requests gradients with respect to well productivity index.
	NEXT	ECLIPSE 100 now also accepts NEXT as an alternative to NEXTSTEP.
	WAPI	Specify the API of injected oil in API tracking cases.

	`WLIST`	Constructs and manages well lists.
	`WPAVEDEP`	Specifies reference depths for the calculation of well block average pressures.
Multi-segment Wells	`WSEGFLIM`	Defines segments to represent a flow limiting valve.

Global keywords

GETDATA	Imports named array data from previous Initial or Restart files. GRID, EDIT, PROPS, REGIONS and SOLUTION sections.

ECLIPSE 300

RUNSPEC section

BRINE	Enables brine tracking.
GRIDOPTS	Requests additional options for processing the grid data.
NODPCDT	Disables the DPCDT option.
NOINSPEC	Disables the Initial Index file (INSPEC) output.
NOMONITO	Disables output for runtime monitoring.
NONNC	Disables the generation or use of non-neighbor connections.
NORSSPEC	Disables the Restart Index file (RSSPEC) output.
REGDIMS	Describes the maximum number of regions associated with miscellaneous keywords in the REGIONS section.

GRID section

ADDZCORN	Adds a constant to the corner point depth array.
EQLZCORN	Resets part of the corner point depth array.
MPFA	Activates the multipoint flux approximation and sets options for handling tensor permeability.
MPFANUM	Specifies regions in which to apply the multipoint flux approximation for handling tensor permeability data.
MPFNNC	Specifies multipoint NNCs and transmissibility coefficients.
MULTNUM	Defines the regions for applying inter-region transmissibility multipliers.
MULTREGT	This keyword can be used to set a transmissibility multiplier between regions that have already been defined using the FLUXNUM or MULTNUM keyword. The MULTREGT keyword can also be used in the SCHEDULE section.
PERMXX	Specifies the xx component of the permeability tensor.
PERMXY	Specifies the xy component of the permeability tensor.
PERMYY	Specifies the yy component of the permeability tensor.
PERMYZ	Specifies the yz component of the permeability tensor.
PERMZX	Specifies the zx component of the permeability tensor.
PERMZZ	Specifies the zz component of the permeability tensor.
SIGMAGDV	Vectorized form of the scalar keyword SIGMAGD.
SIGMAV	Vectorized form of the scalar keyword SIGMA.

PROPS section

BDENSITY	Enables the surface density of water to vary with the salt concentration.
LILIM	Specifies the range for Li-interpolation.

	MISCSTRR	Allows the reference miscible surface tension to be set by saturation region number.
Thermal Option	OILVISCC	Oil viscosity-temperature correlation for the Thermal option.
	PVTWSALT	FVF, compressibility and viscosity of water as functions of salt concentration.
	TOLCRIT	Controls the method of determining the table critical saturations in pathological end point scaling cases.

SOLUTION section

	PBVD	Initial bubble point versus depth in black oil runs with dissolved gas.
	PDVD	Initial dew point versus depth in black oil runs with vaporized oil.
	SALT	Initial salt concentrations in each grid block, for the Brine option.
	SALTVD	Initial salt concentration versus depth, for the Brine option.
Thermal Option	TEMPI	Initial temperatures, often used when specifying non-equilibrium initialization with NEI.

SUMMARY section

(F,G)(O,W,G,L)P(R,T)H,

(F,G)(W,G)I(R,T)H,

(F,G)(WCT,GOR,OGR,WGR)H

Field and group historical production and injection rates, totals and ratios.

(F,G,W)(O,G)nP(R,T)

Field, group and well oil and gas production rates and totals from the nth separator stage (n = 1 to 9).

(G,W)(X,Y)MFn

Group and well liquid and vapor mole fractions for the nth separator stage (n = 1 to 9).

	NBAKFL	Number of calls made to scalar fallback routines in the flash calculation per timestep.
	NEWTFL	Cumulative average Newton iterations per cell in the two phase flash calculations.
	NNUMFL	Total number of two phase flash calculations performed.
	NNUMST	Total number of stability test calculations performed.
	NOMONITO	Disables output for runtime monitoring.
	RGIPA	Region gas in place above TRACK depth.
	RGIPB	Region gas in place below TRACK depth.
	ROIPA	Region oil in place above TRACK depth.
	ROIPB	Region oil in place below TRACK depth.
	ROFR(+/-)	The positive and negative contributions to the inter region oil flow rate.
		Similarly a + or - can be added to all the inter-region summary keywords.
Multi-segment Wells	SCFRn	Multi-segment well segment component n flow rate.

Multi-segment Wells	SPRDM	Multi-segment well segment frictional pressure loss multiplier.
Multi-segment Wells	SPSAT	Multi-segment well segment Psat.

SCHEDULE section

	AIMCON	Controls implicit cells within an AIM run.
	COMPIMB	Sets imbibition saturation table numbers for well completions.
	GINJGAS	Specifies the nature of the injected gas when group injection controls are entered with GCONINJE.
	GCONINJE	Sets injection controls for groups and the field.
	GCONPROD	Sets production controls for groups and the field, using guide rate group control.
	GCONTOL	Items 3 and 4 may be used to control the iterations of the injection calculation.
	GRUPTARG	Resets an individual production rate target or limit for a group.
	LSCRIT	Linear solution convergence criteria.
LGR Option	LSCRITL	Linear solution convergence criteria for all local grids.
LGR Option	LSCRITS	Linear solution convergence criteria for each individual local grids.
	MULTREGT	This keyword can be used to set a transmissibility multiplier between regions that have already been defined using the FLUXNUM or MULTNUM keyword.
	QDRILL	Defines a drilling queue for wells that may be opened in sequence when they are needed to maintain a group rate target under guide rate group control (GCONPROD).
	TOPT	Pre-set tuning options.
	VFPCHK	Sets a threshold BHP for VFP table checks, or turns them off.
	WGRUPCON	Sets well guide rates for group control.
	WLIST	Constructs and manages well lists.
	WPAVEDEP	Specifies reference depths for the calculation of well block average pressures.
	WSALT	Defines the salt concentration in water injectors, for the Brine option.
Multi-segment Wells	WSEGFLIM	Defines segments to represent a flow limiting valve.
Multi-segment Wells	WSEGFMOD	Specifies the multiphase the flow model in individual segments.

Altered keywords for 2000A

ECLIPSE 100

RUNSPEC section

Gradient Option	HMDIMS	Item 4 specifies the maximum number of inter-region multiplier parameters. New item 8 to specify the maximum number of rock compaction or rock compressibility parameters.
Gradient Option	RPTHMD	Item 6 allows gradients of cumulative quantities.

GRID section

AQUCON	Items 12 and 13 set the effective VEFRAC and VEFRACP when using the VE option.
MULTREGT	Item 5 controls application to numerical aquifer connections.
MULTREGD	Item 5 controls application to numerical aquifer connections.
MULTREGH	Item 5 controls application to numerical aquifer connections.

PROPS section

EHYSTR	New item 7 to apply the initial fluid mobility correction to the imbibition end points as well as to the drainage end points.
PMAX	New item 2 to increase the range of pressures for checking total compressibility of the oil-gas system.
RPTPROPS	New mnemonic AQUTAB (switch 25) outputs the Carter-Tracy influence tables.
Environmental and Surfactant Options TRACER	New items 4 and 5 for defining partitioned tracers.

SOLUTION section

RPTRST	Mnemonics PCOW and PCOG (switch 27) output the capillary pressures. Mnemonic RK (switch 29) outputs the water relative permeability reduction factor due to polymer retention, for the Polymer Flood model. The mnemonic GIMULT will output the RSGI, RVGI, BOGI and BGGI multipliers for the current (P,GI) values in each cell, when running the GI Model.

SCHEDULE section

LGR Option COMPDATL	New item 2 inserted (the local grid name), to make the keyword compatible with the ECLIPSE 300 form. Items 3 and 4 (connection I and J locations) now default to those of the wellhead in WELSPECL, if the connection is located in the same local grid as specified in WELSPECL.
LGR Option COMPDATM	This is now an alias for COMPDATL, which has the same data format.
GCONPRI	An additional option, PR2, in items 3, 5, 7 and 9 controls the group/field production rate by prioritization using the second priority formula specified in keyword PRIORITY.
GCONSALE	New option MAXR in item 5.
Gradient Option GRADWELL	Well cumulative production gradients are now available.
IMPES	Has been replaced (in the SCHEDULE section only) by DIMPES (Dynamic IMPES) to avoid confusion with ECLIPSE 300.
IMPLICIT	Has been replaced (in the SCHEDULE section only) by DIMPLICT (Dynamic Implicit) to avoid confusion with ECLIPSE 300.
NUPCOL	This keyword can now be entered in the SCHEDULE section as well as in RUNSPEC.
PRIORITY	Has been extended; items 10-17 allow the definition of a second priority formula.
Gradient Option RPTHMW	New item 3 can be used to nominate wells for output of RFT gradients in conjunction with the GRADRFT keyword.

RPTRST Mnemonics PCOW and PCOG (switch 27) output the capillary pressures. Mnemonic RK (switch 29) outputs the water relative permeability reduction factor due to polymer retention, for the Polymer Flood model. The mnemonic GIMULT will output the RSGI, RVGI, BOGI and BGGI multipliers for the current (P,GI) values in each cell, when running the GI Model.

RPTSCHED Setting the GI mnemonic > 1 will now output the RSGI, RVGI, BOGI and BGGI multipliers, when running the GI Model.

WELPRI New items 4 and 5 provide a means of defining a fixed value to the well's second priority and providing a multiplier for the well's second priority value.

WLISTNAM Now allows well list names beginning with an asterisk. Well list names must begin with an asterisk if they are to be recognized wherever well name roots may be entered in the SCHEDULE section well keywords. For these well lists, it is best to construct them with the new keyword WLIST.

Multi-segment Wells

WSEGFMOD New items 9 and 10 supply parameters for the oil-water slip calculation.

Global keywords

EXTRAPMS Setting the value 4 outputs additional diagnostics for PVT table extrapolations.

INCLUDE Now accepts files names with up to 132 characters.

MESSAGES Various defaults have been increased.

OPTIONS New switches 74 - 79:

Switch 74 controls vertical scaling of capillary pressures when using end point scaling in oil-wet or gas-wet systems.

Switch 76 turns off the monotonicity check in the SURFADS keyword.

Switch 77 restores the original drift flux calculation in multi-segment wells.

Switch 78 restores the use of scratch files for processing NNC data.

Switch 79 restores the pre-2000A behavior of capillary pressure hysteresis.

RPTGRID, RPTPROPS, RPTREGS, RPTSOL, RPTSCHED

Mnemonic NOTHING switches off previously activated reporting for these keywords.

ECLIPSE 300

RUNSPEC section

DPCDT The second argument has been removed.

NUPCOL This keyword can now be entered in the RUNSPEC section, as well as in the SCHEDULE section.

GRID section

DZMTRX This keyword now takes a single value for the entire grid rather than a number for each cell. (Use DZMTRXV to enter cell-by-cell data.)

EDITNNC The default value for the 7th item of this keyword, the transmissibility multiplier, has been altered for ECLIPSE 100 compatibility.

	GDFILE	For ECLIPSE 100 compatibility, this keyword now has 2 data arguments. The first item specifies the name of the grid file to be imported, and the second item specifies whether this file is formatted or unformatted.
	GRIDFILE	New item 2 requests output of the new Extensible Grid file (EGRID).
LGR Option	MINPORV	This keyword can be used to specify the minimum pore volume for both global and local cells. For local grid refinements, the keyword can now be specified between the CARFIN or RADFIN and ENDFIN keywords.
LGR Option	MINROCKV	This keyword can be used to specify the minimum rock volume for global and local cells. For local grid refinements, this keyword can now be specified between the CARFIN or RADFIN and ENDFIN keywords.
	RPTGRID	Mnemonic AQUCON outputs numerical aquifer connection data. Mnemonic AQUNUM outputs numerical aquifer property data. Mnemonics PERMXY, PERMYZ and PERMZX output the off diagonal physical components of the permeability tensor. Mnemonic MPFTRAN outputs the 9 point transmissibilities. Mnemonic MPFANUM outputs the nine point indicator array MPFANUM. Instead of DIFFTX, DIFFTY, DIFFTZ, the mnemonics DIFFX, DIFFY, DIFFZ are now used to output the diffusivities. Mnemonic MULTNUM outputs the inter-region transmissibility multiplier array MULTNUM.
	SIGMA	This keyword now takes a single value for the entire grid rather than specifying an array of values. (Use SIGMAV to enter cell-by-cell data.)
	SIGMAGD	This keyword now takes a single value for the entire grid rather than specifying an array of values. (Use SIGMAGDV to enter cell-by-cell data.)

EDIT section

DIFFT*	For ECLIPSE 100 compatibility, the keywords DIFFTX, DIFFTY, DIFFTZ, DIFFTR and DIFFTTHT have been aliased to DIFFX, DIFFY, DIFFX, DIFFR and DIFFTHT.

PROPS section

MISCSTR	The default reference surface tension has changed from 50 Dynes/cm to zero.
ROCKOPTS	Setting item 2 to STORE now has an effect with ROCK (as well as ROCKTAB and ROCKTABH). The initial equilibrated pressures will be used as reference pressures in the ROCK compressibility.
RPTPROPS	Mnemonic PVTW outputs water PVT data.

SOLUTION section

GETGLOBAL	For ECLIPSE 100 compatibility, the GETGLOBAL keyword has been aliased to GETGLOB.
OUTSOL	Mnemonics RS, RSSAT, RV and RVSAT output the black oil properties Rs, Rssat, Rv and Rvsat.

	RPTSOL	Mnemonics RS, RSSAT, RV and RVSAT output the black oil properties Rs, Rssat, Rv and Rvsat. Aquifer grid block connection data can be output using the mnemonic AQUANCON, instead of AQUCON. Inter-block flow rates of component n can be requested with the FLOCn mnemonic. Inter-block flow rates of water can be output using the FLOWAT mnemonic. In black oil cases only, inter-block flow rates of oil and gas can be output by the FLOOIL and FLOGAS mnemonics.
	RSVD	This has been moved from the PROPS section.
	RSWVD	This has been moved from the PROPS section.
	RVVD	This has been moved from the PROPS section.
Thermal Option	TEMP	This Thermal option keyword has been changed to TEMPI, as a keyword TEMP is already used for the ECLIPSE 100 Temperature option.

SCHEDULE section

	COMPDAT	Positive D-factors (item 12) are now treated as if they apply to the well and will be scaled to a D-factor appropriate to the completion. Negative D-factors are applied directly to the completion. This is now compatible with ECLIPSE 100.
LGR Option	COMPDATL	Items 5 and 6 (connection K location range) can no longer be defaulted. Positive D-factors (item 13) are now treated as if they apply to the well and will be scaled to a D-factor appropriate to the completion. Negative D-factors are applied directly to the completion. This is now compatible with ECLIPSE 100.
Multi-segment Wells	CVCRIT	New items 23 - 26 specify items for Multi-segment wells: the maximum number of iterations and convergence tolerances for molar density, molar flow rate and energy flow rate.
	GCONINJE	Wet gas control mode added to item 3. New item 13 sets wet gas injection rate target/limit.
	GCONPROD	Wet gas control mode added to item 2. Wet gas guide rate added to item 10. New item 16 sets wet gas production rate target/limit.
	GCONTOL	New item 3 sets the tolerance fraction to which the most rapidly changing well gas injection rate estimate or well availability fraction must be met for convergence in the injection calculation iterative loop. New item 4 sets the maximum number of iterations over the injection calculation.
	GECON	New actions CON and +CON allowed in item 7.
	GPMAINT	The data format has been changed to be compatible with the ECLIPSE 100 keyword. Items 8 to 14 have been removed; use GINJGAS instead to specify the nature of the injected gas. Item 2 entries that specify the nature of the injected gas have also been removed. The old data format can still be read if the keyword name is changed to GPMAINT3.
	GRUPTARG	Wet gas rate target/limit added to item 2.
	MULTT*	For ECLIPSE 100 compatibility, the keywords MULTTX, MULTTY, MULTTZ, MULTTR and MULTTTHT have been aliased to MULTX, MULTY, MULTX, MULTR and MULTTHT.

	OUTSOL	Mnemonic PRES_EFF outputs the effective fluid pressure when using rock overburden pressures. Mnemonic PORV_MOD outputs the pore volume multiplier when using rock compaction. Mnemonics RS, RSSAT, RV and RVSAT output the black oil properties Rs, Rssat, Rv and Rvsat.
	RPTSCHED	Mnemonics RS, RSSAT, RV and RVSAT output the black oil properties Rs, Rssat, Rv and Rvsat. Mnemonic MULT outputs the transmissibilities that have been modified using the OVERT* or MULT* keywords. Inter-block flow rates of component n can be requested with the FLOCn mnemonic. Inter-block flow rates of water can be output using the FLOWAT mnemonic. In black oil cases only, inter-block flow rates of oil and gas can be output by the FLOOIL and FLOGAS mnemonics.
	TSCRIT	New items 19 and 20 allow the termination of a run after a given number of consecutive forced or reduced timesteps.
	WCONPROD	Control modes for wet gas, total molar rate, steam rate and saturation pressure added to item 3. New items 13 - 16 set targets/limits for these quantities.
	WDRILTIM	This keyword may now be used to control drilling times for wells opened from the drilling queue (see keyword QDRILL).
	WECON	In item 7 the WREV option has been removed (it is interpreted as WELL). The WELL option now allows wells to be revived if requested in WTEST. The maximum temperature limit for the Thermal option has been moved from item 11 to item 15.
Thermal Option	WELLPROD	Control modes for steam rate and saturation pressure added to item 2. New items 14 and 15 set the targets/limits for these quantities.
LGR Option	WELSPECL	The well's preferred phase must be entered in item 7. Item 10 specifies whether the well is to be shut or stopped when it violates an economic limit.
	WELSPECS	The well's preferred phase must be entered in item 6. Item 9 specifies whether the well is to be shut or stopped when it violates an economic limit.
Thermal Option	WELTARG	Can set target or limiting steam rate and saturation pressure, for the Thermal option.
	WGRUPCON	Wet gas guide rate added to item 4.
	WINJGAS	New option GRUP in item 2 to specify that the nature of the injected gas is defined in a superior group. Item 2 now defaults to this.
	WPAVE	The first weighting factor may now be set to a negative value to request pore volume weighted averaging of inner block and outer ring of neighboring block pressures in the connection factor weighted average pressure calculation.
Multi-segment Wells	WSEGMULT	Items 5-9 allow you to specify a variable friction pressure drop multiplier as a function of WOR and GOR.

Global keywords

`INCLUDE`	Now accepts files names up to 132 characters.0
`MESSAGES`	Various defaults have been increased.
`OPTIONS3`	Setting the 30th integer to a non-zero value activates the ‘mixing tank’ advanced gas model. The ‘sequential’ model is now the default. New switch 36 reverts to a number of pre-2000A well model treatments. New switch 37 reverts to a pre-2000A treatment of DPCDT. New switch 38 applies DPCDT limiting on implicit as well as explicit cells. New switch 40 enables screening of saturations from GETSOL for the Thermal option. New switch 41 reverts to pre-2000A treatment of end points for the Li correlation. New switch 42 may aid convergence in dual porosity cases.

ECLIPSE 100: developments in 99A

New facilities ECLIPSE 100 99A

Grid data

- It is now possible to import binary data as double precision (as well as single precision) when using the GDFILE or IMPORT keywords.
- It is now possible to calculate the dual porosity sigma factor from the matrix block dimensions (LX,LY,LZ) and the three matrix permeabilities (PERMX,PERMY,PERMZ). See the LTOSIGMA keyword.
- It is now possible to apply the NINEPOIN nine point scheme in a nominated region of the reservoir. See the NINENUM keyword.

Properties & equilibration

- The keyword FILLEPS specifies that the grid cell values of all the saturation endpoints (SWL, SWCR, SOWCR etc.) will be written out to the Initial file in all cases. By default, the grid cell endpoints are only written out when the run contains endpoint scaling, for those cells in which the user has entered endpoint data, or for which ECLIPSE calculates the critical endpoints for the initial mobile fluid correction. Other values are undefined, meaning that the unscaled tabular data is to be used.
- The keyword input for modelling gas-water systems has been simplified. The relative permeability and capillary pressure data can be entered with a single keyword SGWFN.
- Item 3 in the ROCKOPTS keyword can now be used to choose to address ROCK tables via SATNUM data rather than PVTNUM data. This replaces OPTIONS keyword switch 43.
- The COPYBOX keyword is now available in the PROPS and REGIONS sections.
- The BOX and ENDBOX keywords can now be used in the SOLUTION section.
- An alternative gravity drainage model is now available for dual porosity systems. The model is based on a formulation proposed by P. Quandalle and J.C. Sabathier in “Typical Features of a Multipurpose Reservoir Simulator” SPE Reservoir Engineering, November 1989. The model can improve the match with finely discretized models in mixed wettability cases (where the *Pc* curve is both positive and negative depending on the saturation). See the GRAVDRM keyword in the RUNSPEC section and the Dual Porosity chapter of the “ECLIPSE 100 Technical Description”.

Well model

- The keyword GEFAC allows efficiency factors to be applied to groups. This is appropriate when all the wells subordinate to a group have their downtimes synchronized, so that the group either flows with its full rate or not at all. The group’s flow rate is then calculated as the sum of the wells’ full rates, but this is then multiplied by the group’s efficiency factor when adding its rate to its parent group’s flow.

- A scale deposition model has been implemented that accounts for the cumulative effects of scale deposited about producing well connections, which result in a degradation of the productivity index. The model relies on two tables (assigned to a well using the keyword WSCTAB):
  - Scale deposition table. This defines the rate of scale deposition per unit water flow rate as a function of the fraction of sea water present in the water flowing into a connection (see keyword SCDPTAB). The sea water fraction is represented by a passive water tracer configured by the user (see keyword SCDPTRAC).
  - Scale damage table. This defines how the current amount of deposited scale alters the PI of the well connections (see keyword SCDATAB).
- The user can now dictate which Group Production Rules actions should take precedence over Gas Lift Optimization, and which should only be performed subsequent to a gas lift optimization calculation, when the ECLIPSE 200 Gas Lift Optimization facility is being used in conjunction with the Group Production Rules (see keyword PRORDER). Previously, whenever a group production target could not be met, gas lift optimization was only performed after all the requested production rule actions had been attempted.
- Extensions to the keyword WPAVE now allow the user to avoid two potential discontinuities in the average grid block pressures as reported to the Summary file under the control of the WBP series of mnemonics. The first discontinuity occurs when a well is shut and the depth correction to the grid block pressures (which uses an average wellbore density) is set to zero. This problem is avoided by using a reservoir density for the depth correction, or switching the correction off altogether. The second discontinuity in the reported pressures occurs when previously closed connections are opened and thus included in the default averaging over all **open** connections. This can be avoided by electing to perform the pressure averaging over all currently **defined** connections, whether open or closed.
- OPTIONS keyword switch 73 may be used to alter the treatment of production wells that are forced by a rate constraint to operate in the unstable part of their VFP curve. In this region the THP decreases as the flow rate decreases, and at a low enough flow rate the THP may fall below its minimum limit. The well would normally then switch to THP control, and it could either start hunting between control modes or die prematurely. However, if the OPTIONS switch is set > 0 the well will be prevented from changing to THP control as long as it would produce at a **higher** rate under THP control. Under these circumstances the well will continue to operate at its rate constraint, while its calculated THP is allowed to fall below the minimum limit.

Output

- It is now possible to write Restart file data more frequently than at each report time. If the user enters 'BASIC=6' in the RPTRST keyword (or 'RESTART=6' in the RPTSCHED keyword) then Restart file data will be written at **every** time-step. Whilst this option can be very useful in some circumstances, it should be used with great care as a great deal of output can be produced! The option will not work with the ECLIPSE 200 Gasfield Operations Model or the Parallel option.
- SUMMARY section keywords GEFF and WEFFG output respectively the group efficiency factor and the product of the well's efficiency factor and that of all its superior groups.
- SUMMARY section keywords CDSM, CDSML and CDSF output respectively the current mass of scale deposited about a connection, the mass of scale per unit perforated length, and the PI multiplicative factor due to scale damage.
- SUMMARY section keywords (F,G)GIM(R,T) output field/group gas import rates and their cumulative totals.

- SUMMARY section keyword CPI outputs a well connection's productivity/injectivity index for the well's preferred phase.
- SUMMARY data can now be written to the RSM file in a format suitable for loading into Excel. See the SUMMARY section keyword EXCEL.
- The RFT file (see keyword WRFTPLT) contains additional items describing the local flowing conditions in the wellbore and more connection properties. The PLT data includes a vector CONVTUB which records the local volumetric flow rate of the mixture in the tubing at each connection, based on the flows of the upstream connections. It also includes vectors CONFAC and CONKH which respectively record the connection transmissibility factors and the Kh values. The Segment data includes vectors SEGOVEL, SEGWVEL, SEGGVEL recording the flow velocities of the free oil, water and free gas phases in each segment, and vectors SEGOHF, SEGWHF, SEGGHF recording their holdup fractions.

Performance

- An extra item has been added to the TUNING keyword to control the maximum length of the time step following a well modification.

Miscellaneous

- The character workspace in ECLIPSE is now dynamically allocated. This will allow ECLIPSE to accept models with a very large number of Summary items. See the MEMORY keyword in the RUNSPEC section.
- FLEXlm licenses for particular ECLIPSE 200 features can be reserved at the beginning of the run, using the LICENSES keyword in the RUNSPEC section. This avoids the situation when a run stops part way through after activating an ECLIPSE 200 feature and finding that there is no available license for it.

Behavioral changes ECLIPSE 100 99A

These may cause changes to the results of a simulation, or alter the form or content of the output.

Grid data

- The behavior of OPTIONS keyword switch 48 has been extended. When this switch is set greater than 1 in a dual porosity, single permeability run, any active matrix cell is inactivated if the corresponding fracture cell is inactive for whatever reason. Previously this action was only taken if the fracture cell was inactive due to a minimum pore volume limit.
- The calculation of the matrix-fracture transmissibility for gravity drainage has been changed to use the Z-direction permeability. This prevents the need to modify the sigma factor to account for permeability effects. It is possible to restore the pre-99A behavior either by setting the Z permeability equal to the X permeability in the matrix or by using the 70th switch in the OPTIONS keyword.
- When an INCLUDEd or IMPORTed grid geometry file has a different unit system, specified by the GRIDUNIT keyword, from the unit convention specified in the RUNSPEC section, ECLIPSE will convert the geometry data to the prevailing unit convention for the dataset. This applies to all data within an INCLUDE, IMPORT or GDFILE having a different unit convention from the prevailing unit convention.

Well model

- A subtle change has been made to the well initialization. In pre-99A versions of the code, every time some control keywords were called (WCONPROD for example), the well flows were re-initialized to zero. Now, unless the well has changed type, the well flows are not re-initialized. This has an effect on the guide rate calculation which can lead to small differences of behavior under group control; immediately after one of these keywords the well guide rate phase ratios can now be calculated from the latest flow rate ratios instead of resorting to the potential ratios. The 99A code should be regarded as giving the more 'correct' behavior but the pre-99A behavior can be restored by using OPTIONS keyword switch 71.

Output

- When using the DATE keyword in the SUMMARY section, the year is now output as a 4-digit number.

  Note that under the 'YEAR' heading it always has been a 4-digit number and it is only under the 'DATE' heading itself that a change has been made. However, if the NARROW keyword is used the year remains a 2-digit number.
- If OPTIONS keyword switch 31 is set greater than 1, the field and region average **potentials** (as well as the pressure) will be weighted by pore volume rather than phase pore volumes.
- The RFT file contains additional vectors describing local fluid conditions in the wellbore and more connection properties (see keyword WRFTPLT). Also the WELLETC vector contains two additional items recording the units for local volumetric flow rate (item 10) and flow velocity (item 11), and the WELLPLT vector contains an additional item (item 6) recording the reservoir fluid volume rate of the well.
- The right hand column of the production and injection reports in the Print file now contains the connection productivity/injectivity indices as well as the well's PI.

New keywords ECLIPSE 100 99A

FILLEPS	Specifies that the grid cell values of all the saturation endpoints will be written out to the Initial file in all cases. By default, the grid cell endpoints are only written out when the run contains endpoint scaling, for those cells in which the user has entered endpoint data.
GEFAC	Sets group efficiency factors.
NINENUM	Applies the nine point schemes (NINEPOIN and NOPC9) only in a nominated region of the reservoir.
SCDATAB	Inputs tables of well PI multiplier vs. the amount of scale deposited per unit length of perforated interval (scale damage tables).
SCDPTAB	Inputs tables of the rate of scale deposition per unit water flow rate vs. the fraction of sea water present in the water flowing into the well (scale deposition tables).
SCDPTRAC	Sets the passive water tracer that will represent the fraction of sea water present in the water flowing into a well.
SGWFN	Inputs gas-water saturation functions.
WSCTAB	Assigns scale deposition and scale damage tables to wells.

New RUNSPEC section keywords

GRAVDRM	Invokes the alternative gravity drainage model for dual porosity runs.
LICENSES	Reserves ECLIPSE 200 option licenses at the beginning of the run.
SCDPDIMS	Sets the dimensions of the scale deposition and scale damage tables.

New SUMMARY section keywords

CDSF	PI multiplicative factor due to scale damage for a connection.
CDSM	Current mass of scale deposited about a connection.
CDSML	Current mass of scale per unit perforated length deposited about a connection.
CPI	Productivity/injectivity index of a well connection, for the well's preferred phase.
EXCEL	Writes the RSM file in a format suitable to load into Excel™.
GEFF	Group efficiency factor.
(F,G)GIMR	Field/group gas import rate.
(F,G)GIMT	Field/group gas import cumulative total.
WEFFG	Product of efficiency factors for a well and all its superior groups.

Altered keywords ECLIPSE 100 99A

BOX	Can now be used in the SOLUTION section.
COPYBOX	Can now be used in the PROPS and REGIONS sections.
ENDBOX	Can now be used in the SOLUTION section.

GDFILE	Can now accept double precision input.
IMPORT	Can now accept double precision input.
LTOSIGMA	New item 5 provides an option to calculate the matrix-fracture transmissibility from all three matrix permeabilities (PERMX, PERMY, PERMZ).
OPTIONS	Switch 31 set > 1 specifies that field and region average potentials are to be weighted by pore volume rather than phase pore volume. Switch 43 is now redundant. ROCK tables will be addressed by SATNUM rather than PVTNUM values if item 3 of the ROCKOPTS keyword is set to the string 'SATNUM'. Switch 48 set > 1 inactivates matrix cells when their corresponding fracture cells in a dual porosity, single permeability run are inactive for whatever reason. Switch 70 set > 0 restores the pre-99A operation of the SIGMAGD keyword, basing the gravity drainage transmissibility on the X-direction permeability. Switch 71 set > 0 restores the pre-99A initialization of wells immediately after reading certain control keywords (such as WCONPROD). This can have a small effect on the guide rates calculated for the phases, for group control. Switch 73 set > 0 allows wells to continue operating under rate control when their rate constraints force them to operate in the unstable part of their VFP curve.
PRORDER	The LIFT mnemonic is now allowed in runs using the ECLIPSE 200 Gas Lift Optimization facility.
ROCKOPTS	New item 3 specifies whether rock tables are to be addressed via SATNUM or PVTNUM. This item replaces the OPTIONS keyword switch 43.
RPTGRID	New SIGMAK mnemonic outputs the processed matrix-fracture transmissibility term in dual porosity runs.
RPTRST	Setting 'BASIC=6' results in data being written to the Restart file at every timestep. New SGTRAP mnemonic to output trapped gas saturation in WAG hysteresis. New RSSAT and RVSAT mnemonics to output saturated *Rs* and *Rv* values.
RPTSCHED	Setting 'RESTART=6' results in data being written to the Restart file at every timestep.
SIGMAGD, SIGMAGDV	The transmissibility is now calculated using the PERMZ rather than the PERMX values.
TUNING	New item 10 on record 1 specifies the maximum length of the time step following any well modifications.
WPAVE	New items 3 and 4 allow control over how reported grid block pressure averages are depth corrected and whether the pressures are averaged over all connections or only those currently open.

Altered RUNSPEC section keywords

MEMORY	New item 2 allocates character memory in 1000 character*8 chunks.
PATHS	The maximum number of path aliases is increased to 20.

ECLIPSE 100: developments in 98A

This section summarizes the changes to ECLIPSE between the 97A release and the 98A release.

New Facilities

Grid data

- A facility now exists to discretise the matrix blocks in a dual porosity run. This makes it possible to model the transient response of the matrix in, for example, a well test. Each matrix cell still only communicates with its corresponding fracture cell; but the matrix, rather than being represented by a single cell, can be sub-divided into a user-defined number of cells. The geometry of this discretization can be assumed to be linear (slab), radial or spherical. See the `NMATRIX` keyword in the `RUNSPEC` section and the `NMATOPTS` keyword in the `GRID` section. See also the "Dual Porosity" chapter of the ECLIPSE Technical Description.
- When multiplying transmissibilities between regions using `MULTREGT`, it is now possible to specify alternative regions using `MULTNUM` rather than using regions defined by `FLUXNUM`, in cases where `FLUXNUM` may be specific to the ECLIPSE 200 Flux Boundary Option.
- The "To" or "From" region numbers in `MULTREGT` can now be defaulted.
- Analogously to `MULTREGT`, the thermal conductivity and diffusivity can be modified between `FLUXNUM` or `MULTNUM` regions. See the `MULTREGH` and `MULTREGD` keywords.
- Pinchout data can now be specified on a regional basis. See the `PINCHREG` and `PINCHNUM` keywords.
- It is now possible to alias path names for included files. See the `PATHS` keyword in the `RUNSPEC` section.

Properties & equilibration

- It is now possible to specify the maximum capillary pressure values and the `SWLPC/SGLPC` values as functions of depth for each end point scaling region. See the `ENPCVD`, `IMPCVD`, `ENSPCVD` and `IMSPCVD` keywords.
- Extra data checks for consistency between the drainage and imbibition endpoints can be requested when the hysteresis model is active, by entering the `HYSTCHCK` keyword in the `PROPS` section.
- It is now possible to specify the initial reservoir temperature as a function of depth when using the Temperature Option. Note that the resulting model is not strictly in thermal equilibrium as there are no source/sink terms at the top and bottom of the reservoir. See the `RTEMPVD` keyword.
- It is now possible to modify the treatment of drainage scanning curves for capillary pressure hysteresis. The 6th switch in the `EHYSTR` keyword will control whether a reversal from scanning imbibition to secondary drainage will re-traverse the scanning imbibition curve or whether scanning secondary drainage curves are constructed.

Well model

- The Action facility has been greatly extended. Actions can now be triggered by well quantities (keyword ACTIONW), group quantities (keyword ACTIONG) and region quantities (keyword ACTIONR) as well as field quantities (keyword ACTION). Each action can be repeated a specified number of times when its triggering condition is satisfied again, and its triggering condition can be incremented each time that it is executed. Actions can be nested, so that an action can result in a new action being defined.
- The Generalized Pseudo-Pressure method of Whitson & Fevang has been added as a new option in Item 8 of the WELSPECS keyword, which can be used in gas condensate runs. The total hydrocarbon mobility is integrated between the grid block pressure and the well connection pressure, taking account of changes in relative permeability caused by condensate dropout.
- Keyword WBHGLR allows economic limits and cutbacks to be performed based on the Gas-Liquid Ratio at well bottom hole conditions. This is useful for reducing the local free gas flow when it may interfere with the operation of a down hole pump.
- Plugbacks can now be performed from the top or bottom of the worst offending completion, instead of the top or bottom of the well's perforations. See keyword WPLUG.
- An extra 'C' option has been added to the WTEST keyword to allow connections closed on automatic workovers to be tested for re-opening **before** the whole well is closed.
- A group can now perform re-injection or voidage replacement based on the production rate of another named group. See keyword GCONINJE.
- Well and group guide rates that are calculated from a formula (keyword GUIDERAT) may be averaged with their values at the previous time step to damp oscillations that result from rate-dependent water cut or GOR.
- The coefficients C and E in the GUIDERAT keyword may now be negative, to provide an additional way to make the guide rates increase with increasing values of R_1 and/or R_2. ECLIPSE will stop the denominator from becoming negative. If the power D or F is negative, the value of R_1 or R_2 will be prevented from falling below 1.0E-6.
- Two new keywords, WTMULT and GTMULT, similar in nature to WELTARG and GRUPTARG, allow the user to multiply a well or group production rate target or limit by an input factor.
- Keyword WPIMULT has been extended to allow connection factors of individual connections to be multiplied by user-defined factors.
- Connections to be opened or closed with keyword WELOPEN may be specified as a range of completion numbers, as an alternative to their i,j,k locations.
- The pressure equivalent radius used in Peaceman's formula can now be input by the user as Item 14 in the COMPDAT keyword.
- When using stabilized VFP tables from VFPi it is possible for wells to flow when they should really be shut in, because the unstable region at low flow rates has been replaced with a horizontal line. By setting Item 3 of the WVFPEXP keyword, ECLIPSE can be instructed to shut the well in at the end of the time step if it is operating in the unstable region of the VFP curve, where the BHP is constant. (Note also that the well will not be re-opened by keyword WTEST if it is only flowing because it is on a stabilized part of the VFP curve.)

- The artificial lift quantity (ALQ) can now be equated to the surface density of the oil or gas produced by a well. VFPi can produce a table spanning a range of surface densities, and ECLIPSE will interpolate this at the current surface density. This facility is particularly useful in API tracking runs. See keywords `WALQCALC` and `VFPPROD`.

Output

- The RFT file output facility has been extended. As well as RFT data, it is possible to output PLT data (including inflows and tubing flow at each connection), and also segment data for ECLIPSE 200 Multi-segment Wells. The data can be written at user-defined times or at each report time, or just when the well is first opened. See the `WRFTPLT` keyword and the "File Handling in ECLIPSE" chapter in the ECLIPSE Technical Description.
- There is now an option to write out an Initial Index File. This lists the arrays written to the `INIT` file. See the `INSPEC` keyword in the `RUNSPEC` section and the "File Handling in ECLIPSE" chapter in the ECLIPSE Technical Description.
- Entering '`VFPPROD=2'`, or greater, in the `RPTSCHED` keyword results in a table being output to the Print file comparing the maximum dimensions defined in the `RUNSPEC` section keyword `VFPPDIMS` with those actually found in the supplied VFP tables. This information can help to save significant amounts of memory.
- The maximum capillary pressure values for each cell can be output to the Print file by entering the mnemonics `PPCW` and `PPCG` in the `RPTSOL` keyword.
- `SUMMARY` section keywords `BPBUB` and `BPDEW` output the grid block bubble point and dew point pressures. `BRSSAT` and `BRVSAT` output the grid block saturated gas-oil (in liquid phase) ratio and oil-gas (in gas phase) ratio. `BSTATE` outputs an indicator of the hydrocarbon state.
- `SUMMARY` section keywords `(F,G)MWD(R,T)` output counts of the number of automatic drilling events that have occurred in a given group.
- `SUMMARY` section keywords `(F,G)MWW(O,T)` output counts of the number of automatic workover events that have occurred in a given group.
- `SUMMARY` section keywords `(F,G)GC(R,T)` output the gas consumption rate and cumulative total at or below a given group.
- `SUMMARY` section keyword `WALQ` outputs the well's ALQ value.
- `SUMMARY` section keyword `CDBF` outputs the blocking factor obtained from the pseudo-pressure option in gas condensate runs (keyword `WELSPECS` item 8).
- `SUMMARY` section keywords `C(O,W,G)FRU` have been extended to give upstream flow rate sums for all types of well, instead of just friction wells. Note that these sums depend on ECLIPSE knowing the structure of the well and are hence primarily aimed at friction and multi-segment wells where the structure is defined by the user. For standard wells, where the user has not defined an order for the connections, ECLIPSE will produce its own structure for the well and will base the upstream sums on this. For wells in local grids, particularly friction wells and multi-segment wells, the global grid counterpart of the well has no structure so the local versions of the keywords, `LC(O,W,G)FRU`, should be used instead.
- New `SUMMARY` section keywords have been introduced to output flow rates, production/injection totals and production ratios in lumped completions (see keyword `COMPLUMP`). These take the form of the corresponding connection quantity, with an '`L`' appended to the end. The resulting output shows the corresponding quantity for the lumped completion that contains the specified connection.

Performance

- An improved treatment of the way in which the well terms are treated in the preconditioner for the solution of the linear equations has been implemented. In some cases this leads to a moderate reduction in the number of linear iterations at the expense of some extra work in the preconditioner. Pre-98A functionality can be restored by setting switch 68 of the `OPTIONS` keyword greater than zero.

Behavioral changes

These may cause changes to the results of a simulation, or alter the form or content of the output.

Properties

- When using the miscible flood model, the effect of hysteresis together with end-point scaling or the initial mobile fluid correction is now accounted for.
- The Leverett J-function is now applied to imbibition capillary pressures when hysteresis is applied.

Well model

- When the API Tracking option is active, in wells using the segmented head option (`WELSPECS` item 12 is '`SEG`') ECLIPSE now calculates the oil density at each connection using the flow-weighted average of the oil surface densities entering each upstream connection. This provides a more accurate value for the local density and the wellbore hydrostatic head than the previous treatment which used the whole well's average oil surface density throughout. The pre-98A treatment can be restored by setting switch 35 in the `OPTIONS` keyword.
- The `QDRILL` option in keyword `WDRILTIM` no longer has the blanket effect of preventing all further drilling while **any** well is currently being drilled. This option now only prevents the drilling of wells while a well of the same type (producer, water injector etc.) is currently being drilled subordinate to the group requesting the drilling. Thus the drilling of an injector, or a producer in a different part of the group hierarchy, no longer stops a producer being drilled (assuming there are no drilling rig restrictions). Pre-98A functionality can be restored by setting switch 66 in the `OPTIONS` keyword.
- Item 1 of the second record of keyword `PRORDER` now prevents the further drilling of more producers while a producer is currently being drilled. The drilling of an injector subordinate to the group no longer stops a producer being drilled. This change is equivalent to the one described above for the `QDRILL` option in `WDRILTIM`, and pre-98A functionality can be restored by setting switch 66 in the `OPTIONS` keyword.

Output

- The region numbers associated with additional sets of fluid-in-place regions entered using the `FIP` keyword are now written to the `INIT` file.
- A value of 3 for the `VFPPROD` mnemonic in the `RPTSCHED` keyword results in **no** VFP tables being output.
- The default maximum number of Summary file quantities has been increased to 10000. This can be reduced to save memory (as well as increased) by using the `SMRYDIMS` keyword in the `RUNSPEC` section.

New keywords

ACTIONG	Defines actions to be triggered by group quantities.
ACTIONR	Defines actions to be triggered by region quantities.
ACTIONW	Defines actions to be triggered by well quantities.
ENPCVD	Inputs the maximum capillary pressures as functions of depth.
ENSPCVD	Inputs the scaled connate saturation for P_c curves only as a function of depth.
GTMULT	Multiplies a group production rate target or limit by a given factor.
HYSTCHCK	Activates additional consistency checks between drainage and imbibition endpoints when using the hysteresis model.
IMPCVD	The imbibition counterpart of keyword ENPCVD.
IMSPCVD	The imbibition counterpart of keyword ENSPCVD.
MAPUNITS	Defines the units used for the MAPAXES data.
MULTNUM	Specifies alternative regions to apply inter-region transmissibility multipliers.
MULTREGD	Specifies inter-region diffusivity multipliers.
MULTREGH	Specifies inter-region thermal conductivity multipliers.
NMATOPTS	Controls the discretized matrix dual porosity option.
PICOND	Controls the calculation of the generalized pseudo-pressure option for well inflow in gas condensate runs.
PINCHNUM	Identifies pinchout regions for use by PINCHREG.
PINCHREG	Specifies pinchout data within regions.
RTEMPVD	Specifies tables of initial reservoir temperature versus depth.
WALQCALC	Defines the well ALQ to be the oil or gas surface density.
WBHGLR	Specifies cutback and economic limits for the well GLR at bottom hole conditions.
WRFTPLT	Governs output of RFT, PLT and SEG data to the RFT file.
WTMULT	Multiplies a well target or limit by a given factor.

New RUNSPEC section keywords

INSPEC	Requests the output of an Initial Index File
PATHS	Specifies aliases for path names.
NMATRIX	Invokes the discretized matrix dual porosity option.

New SUMMARY section keywords

BPBUB, BPDEW	Grid block bubble point and dew point pressures
BRSSAT	Saturated gas-oil ratio (in liquid phase)
BRVSAT	Saturated oil-gas ratio (in gas phase)
BSTATE	Oil-gas state indicator
CDBF	Connection generalized pseudo-pressure blocking factor
C(O,W,G)FRU	Sums of flow rates in upstream connections
C(O,W,G,L)(FR,PT,IT)L	Flow rates and production/injection totals in the connection's lumped completion

C(WCT,GOR,GLR,OGR,WGR)L	Flow ratios in the connection's lumped completion
(F,G)MWD(R,T)	The number of drilling events this time-step or in total
(F,G)MWW(O,T)	The number of workover events this time-step or in total
(F,G,W,C)GLR	Gas-Liquid ratio
(F,G,W)GLRH	Gas-Liquid ratio history (WCONHIST)
(F,G)GC(R,T)	Gas consumption rate or cumulative total
MSUMNEWT	Total number of Newton iterations
WALQ	Well ALQ value
WBGLR	Well Gas-Liquid ratio at bottom hole conditions

Altered keywords

COMPDAT	New item 14 allows the user to specify the pressure equivalent radius used in the calculation of Peaceman's formula.
EHYSTR	New item 6 to control the shapes of drainage scanning curves following a reversal from imbibition to drainage.
GCONINJE	New items 11 and 12 allowing re-injection and voidage replacement targets to be derived from the production of other groups.
GUIDERAT	New item 10 supplies a factor to damp out oscillations caused by rate-dependent water cut or GOR. Items 5 and 7 may now be given negative values.
OPTIONS	
Control 35	Restores pre-98A method of calculating explicit wellbore heads in API tracking runs.
Control 62	Ensures back-compatibility with pre-98A post-processors.
Control 63	Damps explicit head correction in multi-segment wells.
Control 64	Activates developer options for tracer model.
Control 65	Allows DUMPFLUX runs to proceed in spite of certain errors concerning flux regions.
Control 66	Restores pre-98A functionality of 'QDRILL' option in WDRILTIM and Item 1 of record 2 of PRORDER.
Control 67	Activates developer options for Gradient facilities.
Control 68	Restores pre-98A treatment of well terms in the linear solver.
RPTGRID	New PINCH(REG) and PINCHNUM mnemonics to output pinchout data.
	New MULTNUM mnemonic to output transmissibility multiplier regions.
RPTSCHED	Setting 'VFPPROD=2' or greater outputs a table comparing the maximum VFP table dimensions with those set in RUNSPEC.
RPTSOL	New PPCW and PPCG mnemonics to output the maximum capillary pressure values for end-point scaling.
VFPPROD	New definitions of ALQ added for the surface density of oil or gas and the bean size.
WELOPEN	New items 6 and 7 allow the opening/closing of connections belonging to a range of completion numbers.
WELSPECS	New 'GPP' option in Item 8 invokes the Generalized Pseudo-pressure method.
WPIMULT	New items 3 to 7 allow PI multipliers to be defined for individual connections or a range of completion numbers.
WPLUG	New item 4 for prescribing whether plugbacks should be made from the top/bottom of the well or the worst offending completion.
WTEST	A new 'C' option has been added to allow connections closed in automatic workovers to be tested for re-opening before the whole well is closed.
WVFPEXP	New item 3 instructs wells with stabilized VFP tables to be shut in when they are operating in a part of the table that corresponds to the unstable region.

Altered RUNSPEC section keywords

DISKING	The matrix disking option (item 1) is no longer available.

GRIDOPTS	Item 2 allows use of the MULTNUM keyword.
	Item 3 allows use of the PINCHNUM keyword.
SMRYDIMS	The default maximum number of summary quantities has been increased to 10000.

Altered SUMMARY section keywords

BPR,BRV,BRS, BOSAT,BWSAT, BGSAT	The matrix ring number can be appended to the keyword if the discretized matrix model is active in the dual porosity option.

ECLIPSE 100: developments in 97A

This section summarizes the changes to ECLIPSE between the 96A release and the 97A release.

New Facilities

Grid Data

- A facility has been added to multiply transmissibilities between flux regions. See the `MULTREGT` keyword.
- It is now possible to read in a complete set of x, y and z coordinates for every grid cell in order to allow complete flexibility in the corner point representation. See the `GDFILE` keyword.
- An alternative 9-point scheme is now available. This introduces additional non-neighbor connections between cells which are diagonal neighbors in each areal plane. See the `NINEPOIN` keyword in the `RUNSPEC` section.
- The `IMPORT` keyword may now be specified in the `EDIT`, `PROPS` and `REGIONS` sections.

Properties & equilibration

- A new facility has been added to model rock compaction where the compaction is a function of water saturation as well as pressure. This water induced compaction model allows the reservoir pore volume to vary as the product of independent functions of pressure and water saturation, or as a full two-dimensional function of pressure and water saturation. The facility is initiated using the `ROCKCOMP` keyword in the `RUNSPEC` section, and the data is entered using the following keywords in the `PROPS` section: `ROCKTABW`, `ROCKWNOD`, `ROCK2D`, `ROCK2DTR`.
- It is now possible to turn off the rock compaction model by specifying '`NONE`' in the `ROCKCOMP` keyword in the `RUNSPEC` section.
- An option has been added to copy the pressure at equilibration to the overburden array, hence referencing the rock compaction tables to the initial pressure rather than the reference pressure. See the `ROCKOPTS` keyword.
- An new hysteresis option has been added to improve the modelling of WAG (water alternating gas) floods. In a WAG flood all three phases are present together in the flood front and this three phase nature can have a significant effect on the reservoir performance. Traditional hysteresis models, including the existing options in ECLIPSE, look on hysteresis as a 2-phase phenomenon. The WAG hysteresis model provides a simple method of describing the fluid behavior in 3-phase situations. See the `WAGHYSTR` keyword and "Hysteresis in WAG Floods" on page 12-1.
- It is now possible to apply the hysteresis model to the capillary pressure curves alone. See the `EHYSTR` keyword.
- The Killough non-wetting phase hysteresis model has been modified to remove a feature which tended to cause convergence difficulties. See the `EHYSTR` keyword.

- It is now possible to input dry gas properties by entering the compressibility (Z) factor for the gas as a function of pressure, instead of the formation volume factor. See the PVZG keyword.
- A new option has been added for runs containing vaporized oil which use the API tracking facility. By default the surface density tracer is conserved in the oil component present in both the liquid and gas phases. The new option requests that the surface density tracer is assumed to be present only in the liquid phase. See the OPTIONS keyword switch 58.
- The restriction that the RSCONST keyword cannot be used with API tracking in a dead oil system has been removed. But it is still not possible to use RSCONSTT with API tracking.
- A new model has been added in dual porosity (and dual permeability) runs to account for the displacement of fluid from the matrix block due to viscous effects. Typically the pressure gradient in the fracture system will be applied to the matrix blocks, causing flow both into and out of the matrix block. See the LX, LY and LZ keywords and the "Dual Porosity" chapter of the ECLIPSE Technical Description.
- It is now possible to modify the fracture relative permeability for the fracture to matrix flow automatically to reflect the maximum matrix relative permeability at the residual saturation of the displaced phase. See the DPKRMOD keyword and "Dual Porosity" chapter of the ECLIPSE Technical Description.
- The RSVD and RVVD keywords now allow tables after the first to be defaulted to the previous table.
- The EQUALS keyword may now be used in combination with the TNUM keyword.

Well model

- The ACTION and ENDACTIO keywords allow the user to specify a set of SCHEDULE section keywords to take effect when a given condition is satisfied. The set of triggering conditions is currently confined to FIELD quantities such as production or injection rate or cumulative total, GOR, water cut or average pressure.
- Individual wells may have their guide rates or priority values multiplied by a user defined scaling factor, to increase or decrease their relative contributions to a group or field flow target. See keywords WGRUPCON and WELPRI.
- Wells can now be opened from the drilling queue whenever a group's production or injection potential falls below a specified value. If this value is set sufficiently above the group's rate target, the drilling process can be completed in time for when a new well is needed to maintain the group's target production or injection rate. See keyword GDRILPOT.
- There is a new option for placing wells in the drilling queue, which is an alternative to the QDRILL keyword. In a drilling queue set up with QDRILL wells are opened in the order they were placed in the queue. The alternative is to set up a 'prioritized drilling queue' with keyword WDRILPRI. Wells in this queue are opened in decreasing order of their drilling priority, which can be set manually for individual wells or calculated as a function of their potential rates according to a formula specified with the DRILPRI keyword. It is also possible to 'lump' wells into drilling units, so that when one well is drilled all the other wells in the same unit will also be drilled. Only one type of drilling queue may exist at any given time, so the keywords QDRILL and WDRILPRI cannot be used together.
- It is now possible to maintain reservoir pressure (with the GPMAINT keyword) by varying the surface injection rate, which allows more than one phase to be injected into different parts of the reservoir.

- Wells can be switched to their secondary ALQs or THPs in user defined increments or decrements, instead of a single transition. See items 10 and 11 of keyword WLIFT.
- Wells may have their tubing pressure loss (BHP - THP) scaled by multiplying the value interpolated from the VFP table by a user defined factor. See keyword WVFPDP.
- Dynamic scaling of the productivity index of a well according to the (explicit) value of its maximum water-cut can be achieved by use of the PIMULTAB and WPITAB keywords. This feature may be useful in approximating performance decline due to backflow of water.
- The flash transformation ratios in keyword COMPFLSH can optionally be calculated from the properties of bubble point oil obtained from two oil PVT tables: the differential table used within the reservoir, and a nominated flash table.
- A minimum economic liquid production rate for a well can now be specified in the WECON keyword.
- There are two new ways of using the secondary water-cut in item 11 of the WECON keyword. These are enabled by entering 'LAST' or 'RED' in item 12. The 'LAST' option applies the secondary water-cut limit just to the last open connection in the well, allowing the well to be operated at a higher water-cut than its primary limit when it has only one connection left. The 'RED' option causes the well to be worked over when its primary water-cut limit is exceeded, as usual, but continues closing connections until the water-cut falls below the secondary limit. This reduces the number of distinct workovers the well may undergo, and clearly in this case the secondary limit must be smaller than the primary limit.
- When an economic limit at a connection is violated (see keyword CECON) it is now possible to specify whether only the offending connection is closed or all connections below it are closed also (see item 9). There is also an option to continue checking the connection economic limits even when the well is stopped (item 10). Items 11 and 12 allow minimum economic oil and gas production rates to be applied to connections.

Output

- A family of fluid-in-place regions can be set up automatically to represent the gas, oil and water zones at equilibration. See the FIPOWG keyword.
- Saturated dissolved gas oil ratios and vaporized oil gas ratios can be written to the Print file using mnemonics 'RSSAT' and 'RVSAT' in the RPTSOL and RPTSCHED keywords (controls 55 and 56).
- Inter-block flows of each phase may be written to the Print file using mnemonics 'FLOOIL', 'FLOWAT' and 'FLOGAS' in the RPTSOL and RPTSCHED keywords (controls 72-74).
- It is now possible to output the reservoir pore volume and the reservoir volume occupied by each phase for a block, region or the field. A table of the regional values can be output to the Print file using the mnemonic FIPRESV in the RPTSCHED and RPTSOL keywords (control 76). The information can also be written to the Summary file using the (B,R,F)(R,O,W,G,H)PV family of SUMMARY section keywords.
- The well productivity index based on the well block average pressures (see keyword WPAVE) can be written to the Summary file using the SUMMARY Section keywords WPI(1,4,5,9). Also, an alternative averaging technique is available for calculating the well block average pressures, weighting the average pressures in the inner blocks and the outer ring according to their pore volume. See item 1 of keyword WPAVE.
- When the timestep is constrained by the minimum or maximum limits, a mnemonic indicating the next constraint in the absence of the minimum or maximum limits is now also output to the Print file.

- When warning messages have been suppressed for part of the simulation by the NOWARN keyword, they can be switched back on by using the WARN keyword.
- Output is now written to the INIT file to allow a post processor program to mark which cells are connected to aquifers. See the AQUCON and AQUANCON keywords.
- There is now an option to write out a Restart Index File at the end of the run. The file lists the arrays written to the Restart file(s) at each restart time. See the RSSPEC keyword in the RUNSPEC section and the "File Handling" chapter of the ECLIPSE Technical Description.

Performance

- The two-point upstreaming table look-up code has been optimized, significantly reducing the run time for miscible cases with the two-point upstreaming option active.
- An improved solver is available for dual permeability runs which treats the matrix-fracture coupling as a fourth nesting in the nested factorization preconditioner. For back-compatibility, the pre-97A solver may be enabled using switch 60 in the OPTIONS keyword.
- A new tuning keyword, TUNINGDP, has been added to supply a modified set of convergence parameters for some high throughput cases, typically dual porosity runs. The keyword should be used with care as by default it has the effect of relaxing the non-linear convergence criteria.

Behavioral changes

These may cause changes to the results of a simulation, or alter the form or content of the output.

Properties

- The Killough wetting phase hysteresis model has been modified to prevent convergence problems in cells with a small maximum attained non-wetting phase saturation. The original model can be run by setting the 4th item in the EHYSTR keyword to zero.
- Data checks have been added to verify the drainage and imbibition end points when the hysteresis model is active.
- The method of requesting relative permeability hysteresis without capillary pressure hysteresis and the method of calculating the capillary pressure have been modified. Prior to this release, relative permeability hysteresis without capillary pressure hysteresis was requested by setting the 6th switch in the OPTIONS keyword. The switch has been replaced by the 4th item in the EHYSTR keyword. The capillary pressure in this case is taken as the drainage (SATNUM) value, where previously a scaled imbibition curve was used. This modification prevents discontinuities in the capillary pressure function.
- The 24th switch of the OPTIONS keyword has been replaced by the 'STESS' option in the ROCKOPTS keyword.

Well model

- The calculation of field and group cumulative production histories (FOPTH or GWPTH for example) has been amended. Previously these cumulatives were the sum over each well of either its actual cumulative or its historical cumulative, depending on whether the well was currently a history matching well or not. This resulted in discontinuous changes in the historical cumulative when a well was changed from a history matching well to a standard producer or injector. The group cumulative production histories are now calculated in a way that avoids these discontinuous changes. Essentially the group cumulative production history is now equal to the cumulative historical production of all its wells, where the wells' cumulative production histories are incremented according to their actual production rates when they are no longer history matching wells.
- The first record (basic data) of the production and injection well VFP tables (keywords VFPPROD and VFPINJ) have been extended by an additional item which identifies the quantity tabulated in the main body of the table. (The VFPi program can output several different types of table, but ECLIPSE only accepts tables of BHP values.) The new item defaults to 'BHP', so that existing tables do not need to be modified.
- A warning is now printed if the THP limit of a production well lies below the range of THPs in its VFP table, unless it is defaulted to zero (keyword WCONPROD item 10). Defaulting this item will now prevent the well from changing to THP control, even if the THP goes negative. The THP will still be calculated if the well has a VFP table number entered in WCONPROD item 11.
- When the top (or bottom) perforation depth in COMPVE or COMPVEL lies outside the grid block, ECLIPSE adjusts it to back to the top (or bottom) corner depth of the block. Now, however, if this occurs for a single connection entered in a record, ECLIPSE will also adjust the depth of the bottom (or top) of the perforations by a similar amount, provided that the perforated length remains entirely within the grid block. Thus the length of the perforations should be preserved if the depth has to be adjusted due to a mis-match with the grid.
- When the '+CON' option is used in item 7 of the WECON keyword or item 9 of the CECON keyword with the ECLIPSE 200 Wellbore Friction Option, "all connections below" is interpreted as meaning "all connections upstream".
- In item 2 of the GPMAINT keyword all characters are now significant, as new mnemonics have been added which only differ in their 4th character.
- The 54th item of the OPTIONS keyword has been replaced with item 3 in the WEFAC keyword, governing the interpretation of well efficiency factors in a network.

Aquifers

- A warning is now printed if the Carter-Tracy influence function for analytic aquifers is invalid, leading to backflow which could cause unphysical behavior and convergence problems.
- The Analytic Aquifer arrays output to the Restart file have been modified to reduce the disk space used, if the NCAMAX parameter has been set to a value greater than required. The minimum required NACMAX for the run is output as a comment at the end of the SOLUTION section.

Output

- Well report tables have been expanded to allow for 3-digit connection block (i,j,k) coordinates. This also affects the output produced by keyword `RUNSUM` in the `SUMMARY` section, which now by default has fewer columns per page. The keyword `NARROW` can be used in the `SUMMARY` section to restore the pre-97A format.
- Blank columns in the `RUNSUM` output of the `SUMMARY` data, which were caused when group names, well names or connections were defaulted in the `SUMMARY` section keywords, are now removed from these tables.

Performance

- Overriding the principal solver directions is now done using the `SOLVDIRS` keyword instead of using the 3rd switch in the `OPTIONS` keyword.

New keywords

ACTION	Initiates a set of actions (defined by SCHEDULE section keywords) to be executed when a specified condition is satisfied.
DRILPRI	Defines the default priority formula for the prioritized drilling queue set up with keyword WDRILPRI.
ENDACTIO	Indicates the end of the set of actions initiated with the ACTION keyword.
FIPOWG	Request a family of FIP regions to represent the gas, oil and water zones at equilibration.
GDFILE	Imports a Grid file.
GDRILPOT	Causes wells to be drilled from the drilling queue when a group's production rate potential falls below a specified value.
LTOSIGMA	Provides an easy way to calculate SIGMA from LX, LY, LZ.
LX, LY, LZ	Inputs the representative matrix block size for the viscous displacement option.
MULTREGT	Multiplies transmissibilities between flux regions.
PIMULTAB	Inputs tables of well PI multiplier vs. maximum water cut.
PVZG	Inputs dry gas properties using the Z-factor.
ROCK2D	Inputs the two-dimensional water induced compaction tables.
ROCK2DTR	The two-dimensional transmissibility modifier tables for water induced compaction.
ROCKOPTS	Controls options associated with the rock compaction facilities.
ROCKTABW	Supplies rock compaction data as a function of water saturation.
ROCKWNOD	Inputs the water saturation nodes for the two-dimensional water induced compaction tables.
SOLVDIRS	Overrides the principal directions for the linear solver.
TUNINGDP	Supplies a modified set of convergence parameters for some high throughput cases.
WAGHYSTR	Initiates the WAG Hysteresis model.
WARN	Reactivates warning messages after suppression by NOWARN.
WDRILPRI	Places wells in the prioritized drilling queue.
WPITAB	Assigns PI multiplier tables (input with PIMULTAB) to wells.

New RUNSPEC section keywords

ACTDIMS	Sets dimensions for the ACTION keyword facility.
NINEPOIN	Turns on the nine-point disc re tis at ion option.
PIMTDIMS	Sets dimensions of the PI multiplier tables (PIMULTAB keyword).
RSSPEC	Requests the output of a Restart Index file.
VISCD	Enables the viscous displacement option in dual porosity systems.

New SUMMARY section keywords

B(R,O,W,G,H)PV	Block reservoir pore volumes
R(R,O,W,G,H)PV	Region reservoir pore volumes
F(R,O,W,G,H)PV	Field reservoir pore volume
	R = Total pore volume
	O = Pore volume containing oil
	W = Pore volume containing water
	G = Pore volume containing gas
	H = Pore volume containing hydro-carbon
BWPC	Block water/oil capillary pressure
BGPC	Block gas/oil capillary pressure
BGTRP	Trapped gas saturation (WAG hysteresis model)
BGTPD	Dynamic trapped gas saturation (WAG hysteresis model)
CLFR	Connection liquid flow rate
CLPT	Connection liquid production total
MLINEARS	Outputs the number linear iterations for each timestep
MSUMLINS	Outputs the total number of linear iterations since start of run
NARROW	Increases the number of columns per page in the RUNSUM output by restoring the pre-97A format with narrower columns
TELAPLIN	Outputs the elapsed time per linear iteration in seconds
WPI1	Well PI based on the WBP well block average pressure
WPI4	Well PI based on the WBP4 well block average pressure
WPI5	Well PI based on the WBP5 well block average pressure
WPI9	Well PI based on the WBP9 well block average pressure

Altered keywords

CECON	New item 9 to define what action to take upon violating the connection economic limit. New item 10 to allow the continued checking of connection economic limits in stopped wells.
	New items 11 and 12 allow minimum economic oil and gas rates to be applied to connections.
COMPFLSH	New item 8 nominates a flash PVT table number for calculating the transformation ratios.
DPKRMOD	New item 3 scales the fracture relative permeability curves for fracture to matrix flow.
EHYSTR	Item 4 has been added to modify the Killough wetting phase hysteresis model.
	A new item 5 allows the hysteresis model to be applied to the relative permeability or the capillary pressure alone.
GPMAINT	Item 2 has an extended range of options to allow for oil, water and gas surface injection rates to be used to maintain the average pressure of a region.

GSATINJE	New item 5 sets the mean calorific value of the gas injected by a satellite group for, use by the ECLIPSE 200 Gas Energy Production Control option.	
GSATPROD	New item 7 sets the mean calorific value of the gas produced by a satellite group, for use by the ECLIPSE 200 Gas Energy Production Control option.	
IMPORT	This keyword may now be used in the EDIT, PROPS and REGIONS sections.	
	New item 2 allows formatted instead of binary files to be input.	
OPTIONS	Control 3	This is no longer used; it has been replaced by the SOLVDIRS keyword.
	Control 6	This is no longer used; it has been replaced by item 4 in the EHYSTR keyword.
	Control 24	This is no longer used; it has been replaced by item 1 in the ROCKOPTS keyword.
	Control 54	This is no longer used; it has been replaced by item 3 in the WEFAC keyword.
	Control 56	Turns local grids on or off before the global grid is solved.
	Control 57	Controls iterations of the Extended Network with automatic chokes.
	Control 58	Controls API tracking option in vaporized oil runs.
	Control 59	Reserved for Extended Network solver.
	Control 60	Restores pre-97A dual permeability solver.
RPTGRID	Control 47 outputs inter-region transmissibility multipliers.	
	Controls 48, 49 and 50 output matrix block sizes.	
RPTSCHED	Controls 55 and 56 output saturated Rs and Rv if set = 2.	
	Controls 72, 73 and 74 output inter-block flows.	
	Control 75 outputs WAG Hysteresis trapped gas.	
	Control 76 outputs region reservoir volumes.	
RPTSOL	Controls 55 and 56 output saturated Rs and Rv if set = 2.	
	Controls 72, 73 and 74 output inter-block flows.	
	Control 76 outputs region reservoir volumes.	
RSVD, RVVD		
	These keywords now allow tables to be defaulted to a copy of the previous table.	
VFPINJ	New item 6 in record 1 identifies the quantity tabulated in the main body of the table (defaults to 'BHP').	
VFPPROD	New item 9 in record 1 identifies the quantity tabulated in the main body of the table (defaults to 'BHP').	
WECON	Item 12 includes two new mnemonics allowing for different interpretations of the role of the secondary water-cut.	
	New item 14 sets a minimum economic liquid production rate.	
WEFAC	New item 3 governs application of the efficiency factor in the ECLIPSE 200 Extended Network Option, replacing OPTIONS keyword switch 54.	
WELPRI	New item 3 applies a scaling factor to well priority values.	
WGRUPCON	New item 5 applies a scaling factor to well guide rates.	

WLIFT	New items 10 and 11 allow secondary ALQ and THP values to reached in increments or decrements rather than at once.
WPAVE	A negative value for item 1 gives a pore volume weighted pressure average between the inner and outer grid blocks.
WVFPDP	New item 3 applies a scaling factor to the tubing pressure loss.

Altered RUNSPEC section keyword

ROCKCOMP	Additional option in item 1 to turn off rock compaction. New item 3 activates the water induced rock compaction option.

ECLIPSE 100: developments in 96A

This section summarizes the changes to ECLIPSE between the 95A release and the 96A release.

New Facilities

Dynamic Memory

- It is now possible to configure ECLIPSE to allocate memory dynamically, allowing a single executable to run any size of problem, provided the memory is available when the program is run. For large runs it may be convenient to allocate the memory in one "go" at the start of the run. This can be achieved by using the `MEMORY` keyword in the `RUNSPEC` section.

Grid Data

- A facility has been added to allow unformatted input for much of the data in the `GRID` section, to save file store and execution time. Typically the data will be generated by the GRID program. (See the `IMPORT` keyword.)
- An option has been added to the `PINCH` keyword to account for the inactive pinched-out cells when calculating the non-neighbor connection transmissibility. This replaces the `OPTIONS` keyword switch 17.
- It is now possible to refer to fault name roots (ending with an asterisk) in the `MULTFLT` and `THPRESFT` keywords.
- It is now possible to specify additional transmissibility multipliers (`MULTX-`, `MULTY-`, `MULTZ-`) which apply to the X-, Y-, and Z- faces of each grid block. This supplies additional control over the regular and non-neighbor transmissibilities calculated by ECLIPSE. Analogously, it is now possible to specify faults using the `FAULTS` keyword with faces 'X-', 'Y-' or 'Z-'. Corresponding diffusivity multipliers applying to the X-, Y- and Z- faces of grid blocks may also be supplied (`DIFFMX-`, `DIFFMY-`, `DIFFMZ-`). The `RUNSPEC` section keyword `GRIDOPTS` must be used to enable these facilities.
- A option has been added to apply the 9-point scheme only in the areal dimensions. (See the `OPTIONS` keyword switch 50.)

Properties & VE

- Dead oil PVT data may be input in a simplified form which assumes constant compressibility and "viscosibility". (See the `PVCDO` keyword.)
- It is now possible to scale the imbibition capillary pressure curves as well as the drainage curves. (See the `IPCW` and `IPCG` keywords.)
- It is now possible to limit the calculated maximum `PCW` generated when using the `SWATINIT` keyword. Typically this can occur if the input water saturation distribution has water saturations above connate in blocks well above the transition zone. The input water distribution can be honoured in these cases by modifying the connate saturation for these blocks. (See the `PPCWMAX` keyword.)

- Two new facilities have been added to the dual porosity model. Using the INTPC keyword invokes the integrated P_c option, where ECLIPSE will use an integrated version of the input capillary pressure curve. The facility improves the "final recovery" prediction from a matrix cell. The DPKRMOD keyword gives the user a tuning parameter on the oil relative permeability functions. This parameter has no physical meaning, but can be useful when matching a dual porosity simulation to a single porosity model of a single matrix block.
- A new facility has been added to the Vertical Equilibrium option to model the residual flow of oil after a contact has swept a cell. (See the "Vertical Equilibrium" chapter of the ECLIPSE Technical Description and the RESIDNUM keyword.)
- Performance of the Vertical Equilibrium option has been improved by adding an option to modify the corner point depths in the vicinity of pinchouts. This avoids convergence problems when there are inactive pinched-out cells with non-zero thickness. (See switch 52 of the OPTIONS keyword.)

Wells & Aquifers

- When a group cannot meet its production target set in keyword GCONPROD, a sequence of remedial actions can be specified with the keyword PRORDER. The action sequence, known as the "Production Rules", can include opening new connections, switching wells to lower THP, changing well VFP tables, applying a fixed amount of artificial lift, as well as opening new wells from the drilling queue (which was previously the only available action).
- Wells can be automatically transferred from one group to another, depending on the value of a chosen quantity such as the well's water cut. (See the WREGROUP keyword.)
- A group may now have up to five drilling rigs and five workover rigs allocated to it with keyword GRUPRIG. Previously, each group was only allowed the use of one rig of each type.
- The options to model partial penetration (keyword COMPVE) and plugbacks (keyword WPLUG) are now available in non-VE runs. Previously they were only available when using the VE option. In non-VE runs, the partial penetration data entered in keyword COMPVE is automatically converted into COMPRP scaled end-points. The end-points are recalculated each time the perforation depths change in a plugback. There are restrictions on using COMPVE and WPLUG in non-VE 'fast' restart runs.
- An economic upper limit on gas-liquid ratio can be applied to a well. Also it is possible to specify different actions for the primary and secondary water cut limits. (See item 12 and 13 of the WECON keyword.)
- Minimum economic limits can now be imposed on injection wells using the keyword WECONINJ. This allows injection wells to be shut or stopped if they are not injecting at a high enough rate.
- ALQ switching can now be triggered by a high gas-liquid ratio. (See item 9 of the WLIFT keyword.)
- A user-defined adjustment can be added to a well's BHP that has been obtained by interpolating the VFP tables. This may be used, for example, to help match the well's flow rate at a given THP. (See the WVFPDP keyword.)
- It is now possible to specify the injected tracer concentration in a well stream to be a linear function of the cumulative fluid injected by the well. This can be useful when modelling H_2S generating processes. (See Environmental Tracers in the ECLIPSE 200 Reference Manual and the WTRACER keyword.)

- A new aquifer type has been added which supplies a constant water flux. This can be useful in ground water studies to model recharge. (See the AQUFLUX keyword.)
- It is now possible to connect aquifers to cell faces adjoining an active cell, allowing connections into the middle of the reservoir. This option is not recommended for general use. (See the AQUCON and AQUANCON keywords.)
- A new chapter "Aquifer Modeling Facilities" in the ECLIPSE Technical Description now describes the Aquifer Modeling Facilities within ECLIPSE.

Output

- Connection quantity keywords in the SUMMARY section may have their connection I,J,K coordinates defaulted (i.e. the well name is followed immediately by a slash). This will cause the data for all the connections in the well to be output.
- Well connection potential rates can be written to the Summary file, using the SUMMARY section keywords COPP, CWPP, CGPP, COPI, CWPI, CGPI. They represent the oil, water and gas flows in the individual connections of production and injection wells when the wells are flowing at their potentials.
- A variety of well counts can be output to the Summary file. They output the number of wells in any given group that are producing, abandoned or unused. Counts can also be made of the number on artificial lift, under group control, controlled by their own surface rate limit, controlled by their own reservoir volume rate limit, controlled by pressure and controlled by their own oil rate limit. Separate counts are available for production wells and injection wells.
- The number of open connections in a well can be output to the Summary file, using the SUMMARY section keyword WMCON.
- Well efficiency factors can be output to the Summary file, using the SUMMARY section keyword WEFF.
- Two new data items reporting the simulator performance can be output to the Summary file: the average number of linear iterations per Newton iteration for a timestep, and a number representing the reason for selecting the timestep length. (See the SUMMARY section keywords NLINEARS and STEPTYPE.)
- Output of the GRUPTREE keyword data (which is printed if the RPTSCHED keyword contains the WELSPECS mnemonic or has switch 14 set) has been extended to include a schematic diagram of the grouping hierarchy. The diagram will not be displayed if the hierarchy has more than 62 levels.
- It is now possible to request that a Save file is written from a run that is itself a fast restart case. (See the LOAD keyword.)

Behavioral changes

These may cause changes to the results of a simulation, or alter the form or content of the output.

- The ordering of connections in horizontal wells has changed. Previously connections were ordered according to depth irrespective of the well's orientation. Now, if any connection is declared to be horizontal in item 13 of the COMPDAT keyword, ECLIPSE will attempt to order the connections in "heel-to-toe" fashion. It takes the heel to be the connection whose I,J coordinates are nearest the wellhead I,J set in WELSPECS items 3 and 4. If more than one connection can be a candidate for the heel, ECLIPSE selects the one with the minimum depth. The rest of the ordering is determined from the location and orientation of the

connections. If ECLIPSE cannot find a unique sensible ordering (e.g. with a multi-branched well), it will issue a warning message and resort to ordering the connections by depth. If the new ordering is not satisfactory, the user can override it by using the 53rd switch in the OPTIONS keyword, which forces ECLIPSE either to use depth ordering or to leave them in the order in which they were input by the user. ECLIPSE uses the connection ordering for the wellbore head calculation, and also to determine the order in which connection flows are printed in the well reports.

Note Wells that use the ECLIPSE 200 Wellbore Friction option have always had a "heel-to-toe" connection ordering, according to the length down the tubing that is entered in keyword WFRICTN. Thus there is no change for these wells.

- A group may now have up to five drilling rigs and five workover rigs allocated to it with keyword GRUPRIG. Thus if a second rig is allocated to a group, it will now be added to the group's available rigs instead of displacing its original rig.
- The flash transformation option (see keyword COMPFLSH) has been modified for production well connections whose grid blocks contain oil above the bubble point pressure. Above the bubble point, the ratio $B_o(\text{flash})/B_o(\text{diff})$ is held constant instead of the ratio $(B_o(\text{flash})-1)/(B_o(\text{diff})-1)$. The new treatment preserves the value of the undersaturated oil compressibility. The previous treatment may be restored by setting the 42nd OPTIONS keyword switch to a negative number.
- The limit on the amount of lift gas that can be supplied to a group, set in keyword GLIFTLIM, is now applied to the sum over wells of the amount of lift gas used by the well multiplied by its efficiency factor. Back compatibility with 95A (which did not take efficiency factors into account) can be obtained by setting the 49th switch of the OPTIONS keyword.
- The first record (basic data) of the production and injection well VFP tables has been extended. The production well table has 3 additional items, identifying the fixed pressure, the artificial lift quantity and the unit convention. (The VFPi program can output tables of either BHP at fixed values of THP or THP at fixed values of BHP, but ECLIPSE only accepts the former type.) The injection well table has 2 additional items identifying the fixed pressure and the unit convention. Tables with the pre-96A format can still be accepted, and ECLIPSE will apply suitable defaults for the new items. (See keywords VFPPROD and VFPINJ.)
- When production well VFP tables are read in, they are now checked to ensure that lines of BHP vs. flow rate at adjacent THP values do not cross. If a crossing is detected (i.e. if the BHP increases with decreasing THP) a warning message will be printed. It may cause well convergence problems or hunting between control modes. However, crossings may also occur if some BHP values are set artificially high to indicate conditions where the flow is supersonic or exceeds specified erosion velocity limits. These are legitimate, and serve to prevent the well from operating at these conditions. The VFP table check can be thresholded (to report only crossover BHPs below a given value), or even turned off entirely, with the keyword VFPCHK.
- When a group is abandoned because its oil rate or gas rate falls below its economic limit (keyword GECON) then **no** testing for re-opening (see keyword WTEST) will be done on **any** of the wells in this group. This is to avoid the situation of a well that was previously shut or stopped for physical reasons being repeatedly tested for opening in a group that has since been abandoned.

- The SWATINIT option has been modified, such that if the input SWATINIT value is less than or equal to the connate water saturation then the P_c table is not scaled. This prevents large PCW values from being generated, but still honours the input water saturation distribution. Note that high PCW values can still occur if the input saturation is greater than the connate saturation; however this can be controlled by using the new PPCWMAX keyword.
- The Dynamic disking option is no longer available. The IDYNAM switch in the DISKING keyword in RUNSPEC can still be used to control the disking of the grid data. (See the DISKING keyword in the RUNSPEC section.)
- The EPSDEBUG output has been modified to account for the connate water in the gas-oil relative permeability output.

New keywords

AQUFLUX Specifies a constant flux aquifer.

DIFFMX-, DIFFMR-
Diffusivity multipliers for X- and R- grid block faces.

DIFFMY-, DIFFMTH-
Diffusivity multipliers for Y- and Theta- grid block faces.

DIFFMZ- Diffusivity multipliers for Z- grid block faces.

DPKRMOD Modifies the oil relative permeability in dual porosity runs.

IMPORT Imports binary grid files.

INTPC Invokes the dual porosity integrated P_c option.

IPCW, IPCG
Allows vertical scaling of imbibition capillary pressure curves.

MULTX-, MULTR-
Transmissibility multipliers for X- and R- grid block faces.

MULTY-, MULTTHT-
Transmissibility multipliers for Y- and Theta- grid block faces.

MULTZ- Transmissibility multipliers for Z- grid block faces.

PPCWMAX Limits the calculated PCW values when the SWATINIT option is used.

PRORDER Sets the production rule order for maintaining group flow targets.

PVCDO Inputs tables of dead oil PVT functions with constant compressibility.

RESIDNUM Specifies the relative permeability table numbers for the VE residual flow option.

VFPCHK Sets a threshold BHP for VFP table checks, or turns them off.

WECONINJ Sets a minimum economic rate for injection wells.

WREGROUP Allows wells to be automatically switched between groups as certain conditions are violated.

WVFPDP Adds a user-defined adjustment to a well's bottom hole pressure obtained by interpolating the VFP tables.

New RUNSPEC section keywords

MEMORY Allocates the required memory at the start of a run.

GRIDOPTS Allows various additional GRID section facilities.

New SUMMARY section keywords

COPP, CWPP, CGPP
Connection oil, water and gas production potentials.

COPI, CWPI, CGPI
Connection oil, water and gas injection potentials.

(F,G)MW(P,I)(U,A)

Number of producers or injectors that are currently unused or abandoned in the field or a group.

`(F,G)MW(P,I)T`

Total number of producers or injectors in the field or a group.

`(F,G)MW(P,I)G`

Number of producers or injectors on group control.

`(F,G)MW(P,I)S`

Number of producers or injectors controlled by their own surface rate limits.

`(F,G)MW(P,I)V`

Number of producers or injectors controlled by their own reservoir volume rate limits.

`(F,G)MW(P,I)P`

Number of producers/injectors controlled by pressure (BHP or THP).

`(F,G)MWPL`

Number of producers on artificial lift (with ALQ > 0.0).

`(F,G)MWPO`

Number of producers controlled by their own oil rate limit.

`(F,R)PRGZ`

Field or region average of grid block P/Z (for gas phase), weighted according to the gas content of each block at surface conditions.

`GMWPR` Number of production wells currently flowing in a group.

`GMWIN` Number of injection wells currently flowing in a group.

`NLINEARS` Average number of linear iterations per Newton iteration in a timestep.

`STEPTYPE` A number representing the reason for selecting the timestep length.

`WEFF` Well efficiency factors.

`WMCON` The number of connections capable of flowing in a given well.

Altered keywords

`AQUANCON, AQUCON`

New item 11 to allow aquifer connections to cell faces adjoining active cells.

`COMPVE` The keyword can be entered in non-VE runs.

`FAULTS` A fault may be specified with a face ‘X-’, ‘Y-’ or ‘Z-’.

`GCONPRI, GCONPROD, GCONSALE, GECON`

The ‘`PLUG`’ option is now available in non-VE runs.

`GCONSUMP` New item 4, for use with the ECLIPSE 200 Extended Network Option only (see The Network Option in the ECLIPSE 200 manual), to define which network node should be assigned the group’s gas consumption or import.

`GLIFTLIM` The maximum lift capacity in item 2 now refers to the sum of the wells’ lift gas rates multiplied by their efficiency factors (unless the 49th flag of the `OPTIONS` keyword is set).

`GRUPRIG` Any group may now have up to five workover and drilling rigs. New item 4 to state whether the rig(s) should be added to the group or removed from it.

`LOAD` A save file can be requested from a fast restart run.

`MULTFLT` A fault name root may be specified in item 1.

`OPTIONS`

- Control 17 — This is no longer used; it has been replaced by the new options in the `PINCH` keyword.
- Control 42 — A negative value restores the previous flash transformation treatment of undersaturated oil.
- Control 47 — Prevents well connection pressures falling below atmospheric.
- Control 48 — For a dual porosity, single permeability run, inactivate any matrix cells whose corresponding fracture cells are killed by the `MINPV` or `MINPVV` threshold pore volumes.
- Control 49 — Neglects the well efficiency factors when comparing group lift gas rates with the `GLIFTLIM` maximum lift capacity.
- Control 50 — Applies the 9-point scheme only in the areal dimensions.
- Control 52 — Controls alternative treatment of corner point depths for Vertical Equilibrium cases.
- Control 53 — Controls the ordering of connections in horizontal wells.

`PINCH` New items 4 and 5 governing calculation of non-neighbor connection transmissibilities across a pinchout.

`RPTREGS` Control 26 outputs residual flow region numbers (`RESIDNUM`)

`THPRESFT` A fault name root may be specified in item 1.

VEDEBUG	Local grid name may be specified as item 8.
VFPINJ	New items 4 and 5 in record 1, identifying the fixed pressure and the unit convention.
VFPPROD	New items 6, 7 and 8 in record 1, identifying the fixed pressure, the artificial lift quantity and the unit convention.
WECON	New item 12 setting the workover procedure when exceeding the secondary water cut.
	New item 13 setting an economic limits to the gas-liquid ratio.
	The 'PLUG' option is now available in non-VE runs.
WLIFT	New item 9 to enable ALQ switching on high gas-liquid ratio.
WPLUG	The keyword can now be used in non-VE runs.
WTRACER	New item 4 to set cumulative tracer factor.

Altered RUNSPEC section keywords

DISKING	The dynamic disking option is no longer available. The IDYNAM switch just controls the disking of the grid data.

ECLIPSE 100: developments in 95A

This section summarizes the changes to ECLIPSE between the 94A release and the 95A release.

New facilities

RUNSPEC & general data

- The RUNSPEC section now recognizes a set of keywords as an alternative to the fixed record structure. Additionally, if the well and group dimensions are not specified when using the new form of RUNSPEC, they will be determined automatically by a pre-scan of the data file, except in a restart run when they will be taken from the restart file.
- Maximum and minimum limits my be placed on arrays in the GRID, EDIT and PROPS section. Values lying outside these limits will be replaced by the appropriate limiting value. (See keywords MINVALUE, MAXVALUE.)

Grid data

- It is now possible to modify individual (ZCORN) depths within a grid block using the ADDZCORN keyword. In addition, a new keyword EQLZCORN is available to set the corner point depths.
- An option has been added to the COORDSYS keyword to permit the formation of fault NNCs laterally between different reservoirs in order to account for 'bending' (piecewise linear) coordinate lines near faults.
- For a radial grid, an incomplete set of DRV values may be specified, and the remaining radii will be filled in by a geometric series. (See keywords DRV and OUTRAD.)
- The LINKPERM keyword may be used to indicate that grid block permeabilities should apply to cell faces rather than cell centres, so that transmissibilities are calculated directly from these "linking permeabilities" instead of being a function of the permeabilities of the two adjacent grid blocks. Alternatively, the keyword PERMAVE provides additional options for the permeability averaging used in the transmissibility calculation. (See also the chapter "Transmissibility Calculations" in the ECLIPSE Technical Description.)
- The PINCH keyword may now also be used to supply the maximum thickness of the empty gap between adjacent layers, beyond which the transmissibility is set to zero. Also, the strict enforcement of the pinchout thickness can be controlled when cells are inactive due to MINPV or MINPVV. Switch 17 in the OPTIONS keyword can be used to control the treatment of MULTZ at pinchouts.
- It is now possible for cells to be pinched-out horizontally. The appropriate NNCs are formed when the keyword PINCHXY is employed.
- The facility to allow dual porosity only in part of the reservoir has been added. For a dual porosity, single permeability model, regions where the conventional single porosity model applies can be selected via the keyword DPNUM.
- The diffusivities calculated by ECLIPSE for non-neighbor connections may now be edited using the EDITNNC keyword.
- Keywords MULTX/Y/Z, MULTPV and DIFFMX/Y/Z may now be applied in the EDIT section.

- The MULTFLT keyword is now accepted in the EDIT section.
- The threshold pressure facility has been extended to allow the engineer to specify a threshold pressure associated with a named fault. (See the THPRESFT keyword.)
- It is now possible to modify the pore volume and transmissibilities during the simulation by using the MULTX/Y/Z/PV keywords in the SCHEDULE Section. (Note that modifying the pore volume will still conserve material and hence cause the pressure to change.) The BOX keyword can be used to modify a limited region of the model.

Properties

- Oil relative permeabilities may now be entered as a function of both water saturation and gas saturation, in the form of a 2 dimensional table, using the keyword SOF32D.
- The Killough wetting phase model is now available for oil wet systems. (See the EHYSTR keyword and the "Hysteresis" chapter in the ECLIPSE Technical Description.)
- It is now possible to use the end point scaling option with the hysteresis model. The imbibition end points are input using a set of keywords analogous to usual end point arrays (SWL SWCR etc.) but prefixed with a 'I'. For example, the imbibition critical water is input using ISWCR. Imbibition end points can be input as a function of depth by using the IMPTVD and IMKRVD keywords.
- It is now possible to specify a separate Todd-Longstaff mixing parameter for the effective density calculation, rather than using the mixing parameter specified for the effective viscosity calculation. (See keyword TLMIXPAR.)
- A temperature option has been added to allow the modelling of cold water injection. An energy equation is solved at the end of each timestep to generate a temperature distribution. The oil and water viscosities can vary as a function of temperature. (See the "Temperature option" chapter in the ECLIPSE Technical Description.)
- It is possible to associate a 'unit' to a passive tracer. This is used for reporting purposes only. See the TRACER keyword.
- The SWATINIT facility to scale water capillary pressure curves from an input initial water saturation can now be used in gas water systems.

Well model

- Groups can be designated to maintain the average pressure in selected fluid-in-place regions at a specified target value. Pressure maintenance is achieved by automatically adjusting the reservoir fluid volume rate target of either the group's producers or its injectors of a particular phase. (See keyword GPMAINT.)
- Individual groups may optionally have production guide rates calculated from the guide rate formula specified with keyword GUIDERAT. (See item 10 of keyword GCONPROD.)
- Drilling rigs can be allocated to groups, to control the drilling of new wells from the drilling queue (keyword QDRILL) or under automatic opening (e.g. 'AUTO' in item 2 of WCONPROD). See keyword GRUPRIG for more details. Previously there was effectively only one drilling rig, which covered the whole field. Now each group may have its own drilling rig, so wells may be drilled simultaneously in different groups.
- The option to perform a re-tubing or lift switching operation on a well with a low flow rate or a high water cut has been extended to allow the well to switch to a lower THP limit. This can be regarded as switching the well into a lower pressure separator to increase its productivity. The secondary THP limit is set in Item 8 of keyword WLIFT.

- A secondary water cut limit may be applied to a well, to give it a “second life” after it can no longer produce within the original water cut limit. The connections closed in previous workovers are reopened, and these will be closed again when the well exceeds the secondary water cut limit. This limit is set in Item 11 of keyword WECON.

Output

- The RPTxxxx keywords may now optionally be supplied with a list of mnemonics, instead of a list of integers, to determine the output to the print file.
- It is now possible to output the relative permeability end points that are calculated from ENKRVD keyword data. (See keyword RPTSOL.)
- A new option is available for black oil runs which associates region and field oil production with the physical production mechanism, e.g. rock compaction and water influx. The breakdown of the oil production can be written to the Print file or output to the Summary file. (See the “Recovery Mechanisms” chapter in the ECLIPSE Technical Description.)
- It is now possible to output a representative potential where the pressure is corrected to the datum depth, using the water density below the initial WOC, the oil density between the original contacts and the gas density above the initial GOC. (See the “Potential Calculations” chapter in the ECLIPSE Technical Description.)
- It is now possible to output to the Summary file various block pressure averages around the wells (see Summary section keywords WBP, WBP4, WBP5 and WBP9). These averages are useful for comparing with the measured buildup pressure of the well. A choice of weighting schemes can be applied in the calculation (see keyword WPAVE).
- It is now possible to output to the summary file the group and field historical rates and totals. These are the sum, over groups and the field, of well flows (oil, water, gas and liquid) with the flows of all history matching wells replaced with their specified historical rates. Similarly for the cumulative totals and water cut, gas-oil ratio, oil-gas ratio and water-gas ratio. In brief, all the keywords of the form ‘W***H’ now have ‘F***H’ and ‘G***H’ versions.
- The oil, water and gas production and injection rates and cumulative totals within specified fluid-in-place regions can now be written to the Summary file. These are the flows through all production and injection well connections situated within the particular fluid-in-place region. The corresponding summary section keywords take the form R(O,W,G)(P,I)(R,T). The user can obtain, for example, the production and injection rates within individual layers, if the FIP regions (or a FIP region family) are arranged into layers.
- The following new quantities can now be output to the Summary file:
  - The timestep length

    Keyword TIMESTEP
  - The elapsed time per timestep

    Keyword TELAPTS
  - The elapsed time per day of simulation

    Keyword TELAPDAY
  - Grid block water and gas phase pressures

    Keywords BWPR, BGPR
  - Oil recovery mechanism

Keywords (F,R)OR(M,F)(R,W,G,S,F,E,X,Y)

- Initial contact corrected potential

  Keywords FPPC, RPPC, BPPC

- Pressure averages around wells

  Keywords WBP, WBP4, WBP5, WBP9

- The VFP table number currently used by a well

  Keyword WMVFP

- Field and group historical production

  Keywords F***H, G***H

- Region production and injection rates

  Keywords R(O,W,G)(P,I)(R,T).

- Rock compaction data can now be output to the restart files. (See the mnemonic ROCKC in the RPTRST keyword.)
- A facility has been added to allow output of the scaled relative permeability curves to the debug file. The output is controlled with a user defined box, and may be output in GRAF user file form which can be read directly into the GRAF program. (See the EPSDEBUG keyword.)
- It is now possible to change the formfeed character in the print file, e.g. for a laserjet printer. (See the FORMFEED keyword.)
- A new facility has been implemented to output solution data of grid blocks containing well connections to a new (RFT) file. It is possible to write out the pressures and saturations of grid blocks containing well connections prior to each well opening, or at user defined times. This new file can be read into the GRAF program and the resulting plots compared with measured RFT data. (See the WRFT keyword and the “File Handling in ECLIPSE” chapter in the ECLIPSE Technical Description.)

Calculation control & restarts

- A material balance correction technique can be used to reduce the accumulated material balance error. (See the MATCORR keyword.)
- A simple file based system has been implemented to allow ECLIPSE to read SCHEDULE data constructed interactively while the simulation is running. This provides a means for the engineer to control the course of the simulation after examining the results it has already output. (See the READDATA keyword.)
- It is now possible to write all the restart data (including the time dependent data) to the SAVE file, so that restarts can be generated from the SAVE file alone. The RESTART file could then be used purely for graphics output. (See the RPTRST and RESTART keywords.)

Changes to the behavior of existing facilities

(These may cause changes to the results of a simulation, or alter the form or content of the output.)

- The format of the TLMIXPAR keyword has been modified to allow the user to input a separate mixing parameter for the effective density calculation. The second item, which was previously a character string ('DEN' or 'NODEN'), is now a real number denoting the Todd-Longstaff density mixing parameter. Pre-95A data sets having the 'DEN'/'NODEN' flag set must have their TLMIXPAR keyword data modified, or they will report an error. 'DEN' corresponds to defaulting the density mixing parameter, and 'NODEN' corresponds to setting the second mixing parameter to 0.0. No change need be made if the 'DEN'/'NODEN' flag was defaulted.
- The treatment of the oil density term at the oil water contact has been modified. The old method can still be requested by setting the 44th control of the OPTIONS keyword.
- OPTIONS keyword switch 19, governing interblock flow calculations for Vertical Equilibrium, now defaults to 1 in block centred geometry cases. This allows the full cell face area for flow in the block centred case. The results of VE runs with block centred geometry will therefore be different from pre-95A simulations, if the OPTIONS switch was defaulted.
- If both the connection transmissibility factor and the Kh value are supplied in items 8 and 10 of keyword COMPDAT, the denominator term involving the pressure equivalent radius will be calculated to be consistent with these values instead of being calculated from the grid block geometry. This term will affect calculations involving the well P.I. or a D-factor. Pre-95A functionality can be restored by setting OPTIONS keyword switch 45 > 0. (See item 8 of the COMPDAT keyword for more details.)
- The 12th switch of the OPTIONS keyword can no longer be used to control transition zone end point scaling. The TZONE keyword should be used instead.
- The 13th switch of the OPTIONS keyword can no longer be used to request the alternative relative permeability scaling option. The SCALECRS keyword should be used instead.
- Non-neighbor connections are now allowed as the default unless explicitly switched off (see keyword NONNC in RUNSPEC).
- An error message will be generated if both the rock and water compressibilities are zero.
- If the vertical end point scaling facility is used, an extra data check is performed. A warning is issued for any cell where the relative permeability at the critical saturation of the displacing phase exceeds the maximum relative permeability, for example if KRWR exceeds KRW.
- A warning message will be issued if a large negative skin factor extends the effective wellbore radius outside the grid block, or beyond half the pressure equivalent radius. In these circumstances, it may be preferable to change the grid block data near the well to model the physical cause of the negative skin (e.g. enhance the permeability out to a certain distance from the well).
- A warning message will be issued if a well is completed in a grid block having zero x- and y-permeabilities or a zero z-permeability.
- For consistency with other programs, array mnemonics on the Restart file which contain a hyphen (-) have been modified to contain an underscore (_). For example OIL-POTN now becomes OIL_POTN. ECLIPSE still recognizes the original keywords, however, so that pre-95A Restart files may still be read.

New keywords

DPNUM	Delineates dual porosity regions.
EPSDEBUG	Outputs scaled relative permeability curves to the Debug file.
EQLZCORN	Allows the engineer to reset corner point depths.
FORMFEED	Sets the formfeed character in the Print file.
GPMAINT	Designates groups to maintain fluid-in-place region pressure.
IKRG, IKRGR, IKRW, IKRWR, IKRO, IKRORG, IKRORW	Imbibition end point relative permeabilities.
IMPTVD, IMKRVD	Imbibition end points versus depth.
ISWL, ISWLPC, ISWCR, ISWU, ISGL, ISGLPC, ISGCR, ISGU, ISOWCR, ISOGCR	Imbibition end point scaling data.
LINKPERM	Treats grid block permeabilities as “linking permeabilities” applying to cell faces.
MATCORR	Initiates the material balance correction option.
MAXVALUE, MINVALUE	Applies limits to arrays in the GRID, EDIT and PROPS sections.
OILVISCT	Inputs tables of oil viscosity as a function of temperature.
PERMAVE	Allows options for permeability averaging in the transmissibility calculation.
PINCHXY	Generates horizontal pinchout NNCs.
READDATA	Reads files of SCHEDULE data that may be constructed externally during the course of the run.
RTEMP	Sets the initial reservoir temperature.
SCALECRS	Requests the alternative end-point scaling method, replacing the action of switch 13 of the OPTIONS keyword.
SOF32D	Inputs 2D tables of oil relative permeabilities.
SPECHEAT	Inputs tables of fluid specific heat data.
SPECROCK	Inputs tables of rock specific heat data.
THCONR	Sets the rock thermal conductivity in each grid block.
THPRESFT	Sets up a threshold pressure for a fault.
VISCREF	Sets reference conditions for the viscosity vs. temperature tables
WATVISCT	Inputs tables of water viscosity as a function of temperature.
WPAVE	Sets weighting factors for the calculation of well block average pressures.
WRFT	Requests that RFT output data is generated.
WTEMP	Sets the temperature of water injected by wells.

New SUMMARY section keywords

TIMESTEP Timestep length

TELAPTS Elapsed time per timestep

TELAPDAY Elapsed time per day of simulation

BWPR, BGPR

Grid block water and gas phase pressures

(F,R)OR(M,F)(R,W,G,S,F,E,X,Y)

Oil recovery mechanism

FPPC, RPPC, BPPC

Initial contact corrected potential

WBP, WBP4, WBP5, WBP9

Pressure averages around wells

WMVFP VFP table number currently used by a well

(F,G)(O,W,G,L)(P,I)(R,T)H,

(F,G)(WCT,GOR,OGR,WGR)H

Field and group historical production and injection rates, totals and ratios

R(O,W,G)(P,I)(R,T)

Region production and injection rates and totals.

Altered keywords

ADDZCORN Can now be used to modify individual corner depths within a grid block.

BOX Can now be used in the SCHEDULE section in conjunction with the MULTX/Y/Z/PV keywords.

COMPDAT If the connection transmissibility factor is supplied, either Kh or the denominator term will be calculated from it.

COORDSYS New items 5 and 6 to specify lateral communication between different reservoirs.

DIFFMX, DIFFMY, DIFFMZ

Can now be used in the EDIT section.

DRV An incomplete set is now allowed if OUTRAD has been specified.

EDITNNC New item 14 to edit non-neighbor connection diffusivities.

EHYSTR The Killough wetting phase model can now be applied in oil-wet cases.

GCONPROD New options 'POTN' and 'FORM' for item 10.

GRUPRIG New item 3 to allocate drilling rigs to groups.

MULTFLT Can now be used in the EDIT section.

MULTX, MULTY, MULTZ, MULTPV, MULTR, MULTHT

Can now be used in the EDIT and SCHEDULE Sections.

OPTIONS Control 12 is not now used

Control 14 is not now used

	Control 17	determines treatment of MULTZ across pinchouts
	Control 19	defaults to 1 in block centred cases
	Control 44	determines the treatment of oil density at the oil-water contact.
	Control 45	restores pre-95A functionality when connection factor is supplied in COMPDAT item 8
	Control 46	determines the application of MINPV to local grids in the ECLIPSE 200 LGR option.
PINCH	New items 2 and 3 control additional options for pinchouts and gaps between layers.	
RESTART	Two new data items to allow fast restarts from time dependent data stored on the SAVE file.	
RPTGRID, RPTREGS, RPTPROPS, RPTSOL, RPTSCHED, RPTRST		
	Now accepts lists of mnemonics as an alternative to integers.	
RPTRST	Control 14	outputs modified pore volumes and transmissibilities
	Control 15	stops output of total derivatives when using the history matching option
	Control 16	requests a “graphics only” restart file
	Control 17	requests restart data to be written to the SAVE file
	Control 18	governs frequency of writing restart data to the SAVE file
	Control 19	requests output of data required by some programs that can interface with ECLIPSE
	Control 20	outputs rock compaction data.
RPTSCHED	Control 60	outputs modified pore volumes and transmissibilities
	Controls 63-64	output data from the ECLIPSE 200 foam option
	Controls 65-66	output data from the temperature option
	Control 67	outputs the initial contact corrected potential
	Controls 68-70	output data from the ECLIPSE 200 foam option
	Control 71	oil recovery mechanism data.
RPTSOL	Control 15	outputs the rock mass for various ECLIPSE 200 options
	Controls 60-62	output processed K_r data from the ENKRVD keyword
	Controls 63-64	output data from the ECLIPSE 200 foam option
	Controls 65-66	output data from the temperature option
	Control 67	outputs the initial contact corrected potential
	Controls 68-70	output data from the ECLIPSE 200 foam option
	Control 71	oil recovery mechanism data.
RUNSPEC	Dimensions and switches are now entered using keywords.	
	The old fixed record structure will still be recognized, for back compatibility with pre-95A data sets.	

`TLMIXPAR` Format changed to allow the input of a separate mixing parameter for the effective density calculation, which is now item 2 in place of the '`DEN`'/'`NODEN`' flag.

Pre-95A data sets will need to be modified if the '`DEN`'/'`NODEN`' flag has been used. '`DEN`' corresponds to defaulting the density mixing parameter and '`NODEN`' corresponds to setting the second mixing parameter to 0.0.

`TRACER` Optional item 3 to specify units for reporting purposes.

`WECON` New Item 11 to set the secondary water cut limit.

`WLIFT` New Item 8 to set the secondary THP limit.

ECLIPSE 100: developments in 94A

This section summarizes the changes to ECLIPSE between the 93A release and the 94A release.

New facilities

Grid data

- It is now possible to make simple modifications to cell corner point depths, using the ADDZCORN keyword.
- When the grid is divided into separate reservoirs with the COORDSYS keyword, each with its own set of coordinate lines, by default there is no transmissibility between adjacent reservoirs. There is now an option in the COORDSYS keyword to allow Z-direction transmissibilities to be calculated between adjacent reservoirs. This enables grids to be set up with 'bending' coordinate lines.

Properties & initial conditions

- The "Compressed Vertical Equilibrium" option has been re-written in an improved form. The compressed vertical equilibrium option allows a model to be run assuming the reservoir fluids are in vertical equilibrium across the entire reservoir section. This is in contrast to the 3D vertical equilibrium option, in which fluids are assumed to be in vertical equilibrium within each individual cell. The 94A option uses extended internal tables to describe the variation in properties, including saturation table end-points, up each column of original cells. Efficiency is gained by reducing the problem to 2D in the regions of the field where vertical equilibrium is assumed to hold across the complete thickness. (See the "Compressed vertical equilibrium option" chapter in the ECLIPSE Technical Description, and the keywords COLLAPSE, CRITPERM and VEDEBUG.)
- There is a facility to input the minimum oil saturation used in the STONE1 3-phase relative permeability model, either as a function of water saturation or as a function of gas saturation. (See the SOMWAT and SOMGAS keywords.)
- The STONE I and II 3-phase relative permeability models may now be used in conjunction with the hysteresis option.
- It is now possible to use the ADD and MULTIPLY keywords in the PROPS section, to modify keyword data arrays over grid cells.
- It is now possible to scale the connate saturation of the capillary pressure curve without scaling the oil relative permeability curves. (See the SWLPC and SGLPC keywords.)
- The capillary pressure curves can now be scaled to honour a maximum capillary pressure entered on a grid block by grid block basis. This facility can be employed in two special options:
  - The scaling can be used in conjunction with the surface tension option to implement a J-function description of the capillary pressures.
  - The scaling can be used to honour an input water distribution. (See the PCW, PCG, JFUNC and SWATINIT keywords and the "Saturation Table Scaling" chapter in the ECLIPSE Technical Description.)

- It is now possible to input the rock compressibility data such that the table numbers correspond to the saturation region numbers (SATNUM) rather than the PVT region numbers (PVTNUM). (See the ROCK keyword and switch 43 of the OPTIONS keyword.)
- The API Tracking facility has been extended to allow different groups of oil PVT tables to be used in different PVT property (PVTNUM) regions. Previously, oil PVT properties could not vary with PVT region number when using the API Tracking facility. (See the APIGROUP keyword.)
- In API tracking runs, the restriction that all oil PVT tables must be input using the same set of Rs nodes has been removed. Tables can now be entered with different sets of Rs nodes, except that the upper and lower Rs nodes must be the same in all tables. ECLIPSE internally maps the input data on to a common set of Rs nodes (which includes all the input nodes). The upper and lower Rs nodes must be the same in each table to prevent extrapolation of the input data during this process. (See the "API Tracking" chapter in the ECLIPSE Technical Description.)
- When using the Todd and Longstaff miscible option, it is now possible to model the transition between miscibility and immiscibility as a function of pressure. (See the PMISC keyword.)
- When using the miscible option, it is now possible to modify the default straight line relative permeability curves for miscible gas displacing oil. (See the MSFN keyword.)
- When using the miscible option, the Todd-Longstaff mixing calculation may now be applied to the viscosity only, leaving the densities to take the pure component values. (See the TLMIXPAR keyword).
- A new facility has been added to the Molecular Diffusion option to allow the diffusion of gas and oil components between the gas phase and the oil phase. (See the DIFFC keyword and the "Molecular Diffusion" chapter in the ECLIPSE Technical Description.)
- The brine option now allows the brine surface density to vary with salt concentration. (See the BDENSITY keyword).
- The initial vaporized oil distribution in an equilibration can now be specified as a table of dew point pressure vs. depth, using the PDVD keyword as an alternative to RVVD.
- In runs with enumerated initial conditions, it is now possible to input the initial bubble point and dew point pressures instead of the initial Rs and Rv distributions. (See the PBUB and PDEW keywords.)
- An option has been provided to limit the application of the DRSDT limit to cells containing free gas. This can prevent non-physical behavior in undersaturated oils with a varying Rs value. (See the DRSDT keyword.)

Aquifers

- It is now possible to modify the analytic aquifer influx coefficient by using a multiplier. (See the AQUANCON keyword.)
- There is a choice of two options for calculating the transmissibilities between a numerical aquifer and the reservoir grid blocks. (See the AQUCON keyword).

Well model

- Known production and injection flows from other reservoirs or regions outside the simulation grid can be incorporated into the simulation model via "satellite groups". These are groups that contain no wells or subordinate groups, and whose production and injection rates are set periodically by the user. (See keywords GSATPROD and GSATINJE.)

- When running non-compressed Vertical Equilibrium cases, there is an option to set the connection centre depth (at which the drawdown is calculated) to the mid-point of the top and bottom depths of the perforations set in keyword COMPVE.

  By default, the connection depth is equal to the grid block centre depth. The block pressure is corrected from the cell centre depth to the well connection centre depth using the hydrostatic gradient of the phases between the two depths. This method gives a better representation of the wellbore pressure - depth profile for partially penetrating wells completed in thick grid blocks, and reduces the tendency for spurious crossflow to occur. It is also consistent with the calculation of well inflow in the Compressed Vertical Equilibrium option. The option is activated with switch 41 in the OPTIONS keyword.
- When well rate targets have to be reset for each well on a regular basis (e.g. when simulating production history), a new "well list" facility allows the data to be entered in a more compact form than is possible using the WELTARG keyword (which requires one line of data per well). One or more lists of wells can be set up with the WLISTNAM keyword, and target rates can be reset with keyword WLISTARG for all the wells on each list in a single data record.
- A flash transformation facility has been added, to transform well flows to flash liberation conditions, when the oil PVT tables are derived from differential liberation data. (See the COMPFLSH keyword.)
- A cubic spline procedure can be requested for interpolating the ALQ (Artificial Lift Quantity) values in the VFP tables. This is primarily intended to be useful for the ECLIPSE 200 Gas Lift Optimization facility, by making the gradient of the wells' performance with reference to the lift gas injection rate vary more smoothly over the table. (See keyword VFPTABL.)

Output

- It is now possible to output the oil, water and gas relative permeability values in the grid blocks to the print, restart and summary files. (See keywords RPTSOL, RPTSCHED and RPTRST, and the Summary section mnemonics of the form BOKR, BWKRX etc.)
- The bubble point and dew point pressure distributions can be output to the print file. (See the RPTSOL and RPTSCHED keywords.)
- The pressure of a Fetkovich aquifer can be output to the Summary file using the Summary Section mnemonic AAQP.
- For History matching Wells (declared with keyword WCONHIST), the well cumulative production history totals of oil, water, gas and liquid can now be written to the Summary File, using the Summary Section mnemonics WOPTH, WWPTH, WGPTH, WLPTH. The engineer may then compare their values directly with the wells' actual cumulative production totals. Also, the measured values of the bottom hole pressure and tubing head pressure can optionally be entered in the WCONHIST keyword. These can be written to the Summary File using the Summary Section mnemonics WBHPH and WTHPH, for comparison with the calculated values.
- To aid the history matching of injection wells (whose calculated flows may differ from the observed flows if they exceed a BHP or THP limit), their observed flow rates and cumulative totals can be written to the Summary File using the Summary Section Mnemonics WOIRH, WWIRH, WGIRH, WOITH, WWITH, WGITH. (See keyword WCONHIST for advice on history matching injectors.)
- The current number of flowing production wells and injection wells in the field can be written to the Summary File with the mnemonics FMWPR and FMWIN.

- Drainage regions can be written to the Restart file, for graphical display purposes. (See the RPTRST keyword.)

Changes to the behavior of existing facilities

(These may cause changes to the results of a simulation, or alter the form or content of the output.)

- The behavior of the RSCONSTT and RVCONSTT keywords has been modified when a table is defaulted. The data for the defaulted table is now copied from the last table rather than table 1.
- The calculation of the default threshold pressures (see keyword THPRES) has been modified to ignore potential differences in cells where a phase is not present. This prevents the calculation of unrealistically large threshold pressures due to the oil potential difference in a water leg.
- Well efficiency factors exceeding 1.0 may now be specified in keyword WEFAC; ECLIPSE no longer regards this as an error.
- Well D-factors may be entered with keyword WDFAC, instead of Item 12 of keyword COMPDAT. If COMPDAT Item 12 is defaulted, the connection D-factors will retain their previous values, instead of being set to zero. These changes are made to clarify the difference between a well D-factor and a connection D-factor. Connections may be shut using COMPDAT without having to respecify the well's D-factor.
- A warning message is now printed at the end of the time step if a group control rate target is exceeded. This would result if the flow from wells declared unavailable for group control is more than that required to achieve the group's target rate.
- The influx from analytic aquifers is now reported separately in the fluid-in-place reports, rather than being subtracted from the "outflow through wells" total.
- An initial restart file will be generated from a restart run if it is requested by either the RPTSOL or RPTRST keywords. This is useful in ECLIPSE 200 flux runs when the base case does not dump flux data.
- The region average DZ value will be output if the region reports are requested using the RPTSOL keyword.
- When output arrays are written to the print file, the maximum and minimum values and their locations are printed at the top of the page. If an array consists of only one value, the full array is not output.

New keywords

ADDZCORN	Adds a constant to the cell corner point depths (ZCORN array).
APIGROUP	Allows the grouping of PVT tables for the API tracking option.
BDENSITY	Brine surface density as a function of salt concentration.
COLLAPSE	Allows user control over which cells are to be collapsed when using the compressed VE option.
COMPFLSH	Flash transformation ratios for well connections.
CRITPERM	Inputs a critical permeability to control which cells are collapsed when using the compressed VE option.
GSATINJE	Injection rate data for satellite groups.
GSATPROD	Production rate data for satellite groups.
JFUNC	Activates the Leverett J-function option.
MSFN	Miscible saturation function tables.
PBUB	Initial bubble point pressure in each cell.
PCG, PCW	Scales the maximum gas and water capillary pressures.
PDEW	Initial dew point pressure in each cell.
PDVD	Dew point pressure vs. depth tables for equilibration.
PMISC	Pressure dependent miscibility tables.
SGLPC	Scales the connate gas and water saturation for the capillary SWLPC pressure curves without scaling the oil relative permeability.
SOMGAS, SOMWAT	Tables of minimum oil saturation used in the STONE I 3-phase relative permeability model.
SWATINIT	Initial water saturations for capillary pressure scaling.
VEDEBUG	Controls debug output for the vertical equilibrium options.
VFPTABL	Requests cubic spline interpolation for ALQ in the VFP tables.
WDFAC	Sets well D-factors.
WLISTARG	Resets operating targets/limits for lists of wells.
WLISTNAM	Sets up lists of wells, for use in keyword WLISTARG.

New summary section mnemonics

WBHPH, WTHPH	Well BHP and THP history (for History Matching Wells)
WOPTH, WWPTH, WGPTH, WLPTH	Well cumulative production history totals (for History Matching Wells)
WOIRH, WWIRH, WGIRH	Well injection rate history (target values in WCONINJE or WELTARG)
WOITH, WWITH, WGITH	Well cumulative injection history totals (cumulative target rates)

FMWPR, FMWIN

Current number of flowing production and injection wells in the field

AAQP Fetkovich aquifer pressure

BOKR, BWKR, BGKR, BOKRX, BOKRX-, etc.

Block relative permeabilities and their directional counterparts.

Altered keywords

ADD	Can now be used in the PROPS section.	
AQUANCON	New item 10 to input an influx coefficient multiplier.	
AQUCON	New item 10 to choose the transmissibility calculation option.	
COMPDAT	Defaulting item 12 now causes the connection's D-factor to retain its previous value, instead of setting it to zero.	
COORDSYS	New item 4 to request calculation of the Z-transmissibilities between adjacent reservoirs.	
DIFFC	New items 7 & 8 to input cross phase diffusion coefficients.	
DRSDT	New item 2 to apply the DRSDT limit only to cells containing free gas.	
MULTIPLY	Can now be used in the PROPS section.	
OPTIONS	Control 41:	Switches on the modified well inflow calculation in vertical equilibrium cases.
	Control 42:	Average over N timesteps the block Bo & Rs used in the COMPFLSH transformation.
	Control 43:	Associates the ROCK data with the saturation region numbers.
ROCK	Will be associated with the saturation region number if OPTIONS switch 43 is set.	
RPTRST	Control 1:	Now outputs an initial restart file from a restart run.
	Control 10:	Outputs drainage regions.
	Control 11:	Output oil relative permeabilities.
	Control 12:	Output water relative permeabilities.
	Control 13:	Output gas relative permeabilities.
RPTSCHED	Control 55:	Outputs the bubble point pressure.
	Control 56:	Outputs the dew point pressure.
	Control 57:	Outputs oil relative permeabilities.
	Control 58:	Outputs water relative permeabilities.
	Control 59:	Outputs gas relative permeabilities.
RPTSOL	Control 7:	Now outputs a restart file from a restart run.
	Control 8:	Region reports now include the average DZ values.
	Control 30:	Outputs scaled max water Pcs generated from SWATINIT.
	Control 55:	Outputs the initial bubble point pressure.
	Control 56:	Outputs the initial dew point pressure.
	Control 57:	Outputs oil relative permeabilities.

Control 58: Outputs water relative permeabilities.

Control 59: Outputs gas relative permeabilities.

`RSCONSTT`, `RVCONSTT`

Defaulted table will default to the last input table.

`RUNSPEC` `QVEO2D` flag in Record 10

Activates the improved Compressed Vertical Equilibrium option.

`QQCOAL` flag in Record 10

New flag to activate the ECLIPSE 200 Coal Bed Methane model.

`STONE1`, `STONE2`

Can now be used in runs with the hysteresis option activated.

`TLMIXPAR` New item 2 to prevent the mixing calculation being applied to the density.

`WCONHIST` New items 9 and 10 to enter measured values of THP and BHP.

`WEFAC` Efficiency factors exceeding 1.0 are now allowed.

ECLIPSE 100: developments in 93A

This section summarizes the changes to ECLIPSE between the 92A release and the 93A release.

New facilities

Grid data

- It is now possible to copy grid data from one part of the field to another by using the `COPYBOX` keyword.
- A new keyword `MULTPV` has been added to the `GRID` section to modify the pore volumes calculated by ECLIPSE. This keyword is analogous to the `MULTX`, `MULTY` and `MULTZ` keywords used to modify the calculated transmissibilities. Use of the keyword is recommended in preference to using the `PORV` keyword in the `EDIT` section. The `MULTPV` modifications will be carried through to any local grid refinements automatically.
- A new facility has been added to allow easy modification of the transmissibility across a predefined fault. (See the `FAULTS` and `MULTFLT` keywords.)
- It is now possible to modify the maximum distance two active cells can be separated by and still generate a pinch-out connection. This threshold distance is input using the new keyword `PINCH`.
- The map origin from the GRID program can be input and subsequently output to the `GRID` file. (See keyword `MAPAXES`.)

Properties & equilibration

- When using the End Point Scaling option, it is now possible to scale the relative permeability tables at the critical saturation of the displacing phase. (See the `KRW/KRWR`, `KRO/KRORW/KRORG`, `KRG/KRGR` and `ENKRVD` sets of keywords.)
- In VE runs it is now possible to specify the relative fractions of rock curve and VE data used for the relative permeabilities and the capillary pressures independently. Previously, the same fraction had to apply for both the relative permeability and the capillary pressure. (See the new keywords `VEFRACP` and `VEFRACPV` and the original keywords `VEFRAC` and `VEFRACV`.)
- The initial dissolved gas distribution in an equilibration can now be specified as a table of bubble point pressure vs. depth, using the `PBVD` keyword as an alternative to `RSVD`.

Well model

- A group control target or upper limit can be applied to the reservoir fluid volume production rate. (This option has previously been available only with group control by prioritization; it is now available also with guide rate control.) In addition, a target or limiting “Production Balancing Fraction” can be specified. This limits the group’s reservoir fluid volume production rate to the specified fraction (or multiple) of the group’s reservoir fluid volume injection rate. (See keywords `GCONPROD` and `GCONPRI`.) Wells and groups can be given reservoir fluid volume guide rates.

- Group production rate targets or limits can be re-set individually with the new keyword GRUPTARG, without the bother of specifying all the other quantities required by the group control keywords GCONPROD or GCONPRI. GRUPTARG has the same relationship to these keywords as WELTARG has to WCONPROD.
- Wells can be re-assigned to a different group during the simulation, by repeating their WELSPECS keyword data with the new group name. Also, in a multi-level grouping hierarchy, groups can be re-assigned to different "fathers", and new groups can be inserted into the hierarchy, by using the GRUPTREE keyword to declare any changes to the "father-son" relationship.
- There is an option to cut back the target production rate of a group by a specified fraction each time it exceeds a water cut, GOR, GLR or WGR limit (see keyword GCUTBACK). This is the equivalent of the WCUTBACK keyword facility for wells applied to group production rates.
- When well guide rates are calculated from a general formula (see keyword GUIDERAT), there is an option to prevent guide rates from increasing. This option decreases the tendency for guide rates to oscillate due to rate-dependent coning phenomena.
- Wellbore storage effects in well tests can be modelled by giving the wellbore a finite volume. (See keyword WBOREVOL.)
- Wells can be closed automatically if their THP exceeds a specified maximum design limit (see keyword WTHPMAX). A high THP could occur if the GOR increases, for example due to gas coning. Wells closed for this reason may be periodically tested for reopening, if requested with keyword WTEST, to see if their THP has fallen enough to operate safely again.
- If a well is unable to produce at its THP limit, there is an option to perform a workover to reduce its water cut (see keyword WORKTHP).
- Automatic retubing or artificial lift switching can now be triggered by high water cut as well as low flow rate (see keyword WLIFT).
- When the well cycling facility is being used (e.g. for a WAG simulation), there is an option to control the time steps to coincide with the well's on/off cycling times (see keyword WCYCLE).
- There is an option to stop the run if a History Matching Well changes to BHP control (see keyword WHISTCTL). The BHP limit for History Matching Wells can now be set with keyword WELTARG.

Output

- Multiple sets of fluid-in-place regions can be specified, to enable reporting of results from different groupings of grid blocks. Results from the multiple fluid-in-place regions can be output to the print file and the Summary file. (See the FIP keyword, and NMFIPR in RUNSPEC record 16.)
- It is possible to output the COORD and ZCORN data in keyword form. When used with the BOUNDARY keyword, this enables smaller grids representing part of the field to be easily output. (See keyword RPTGRID.)
- It is now possible to output fluid viscosities and phase reservoir densities to the print file, the Restart file (for graphical display) and the Summary files. See the RPTSOL and RPTSCHED keywords to output array data to the print file, RPTRST for output to the Restart file, and the new Summary section keywords for the Summary file output (F*VIS, R*VIS, B*VIS and F*DEN, R*DEN, B*DEN).

- The net production from a fluid-in-place region can be written to the Summary file, using the `Summary` section mnemonics ROP, RWP and RGP. This is the total production rate through wells, minus any injection, and minus any influx from analytic aquifers. It is the quantity reported as "outflow through wells" in the fluid-in-place regions report.
- The control mode of each group under guide rate control for production, water injection and gas injection can be written to the Summary file, using the `Summary` Section mnemonics F/GMCTP, F/GMCTW, F/GMCTG (see the `Summary` section). If `RUNSUM` is specified, the control mode numbers are printed as character strings, unless Lotus output format has been requested.
- The production rates and cumulative totals (from Field, Groups, Wells and Connections) of free gas, solution gas, free oil and vaporized oil at grid block conditions can be written to the Summary file. The `Summary` section mnemonics are (F/G/W)(O/G)(PR/PT)(F/S) and C(O/G)(FR/PT)(F/S).
- The CPU time, elapsed time, and the maximum changes in pressure and saturation can be written to the Summary file. (See the `Summary` section mnemonics TCPU, TCPUTS, TCPUDAY, ELAPSED, MAXDPR, MAXDSO, MAXDSW, and MAXDSG.)
- It is now possible to modify the maximum number of quantities written to the Summary file (see `NSUMMX` in record 26 of the `RUNSPEC` section). Previously, this limit was fixed with the installation.

Timestep control & restarts

- The timestep size can be controlled to limit the expected maximum pressure change, and to chop a timestep if a maximum tolerable pressure change has been exceeded. This facility is useful when ECLIPSE is used to model well tests. (See the `TUNING` keyword.)
- The preparation of `Schedule` section data for a restart run can be made easier by entering the keyword `SKIPREST` at the beginning of the `Schedule` section. It instructs ECLIPSE to skip the subsequent keywords in the `Schedule` section of a restart run until the restart time has been reached. This saves you from having to delete the keywords yourself from a copy of the base run's `Schedule` section.

Changes to the behavior of existing facilities

(These may cause changes to the results of a simulation, or alter the form or content of the output.)

- The form of the `ENKRVD` keyword has been modified. The keyword now expects 8 columns instead of 4. The keyword data in data sets made for previous versions of ECLIPSE must be modified by adding 4 default values (4*) to the end of each line.
- Addition of the `DATE` keyword in the `Summary` section now implies that the date (rather than the time) will be written in the left hand column of the Run Summary output generated by the `RUNSUM` keyword.
- If inter-block flows are requested to be output to the Restart file, these will also include the flows in any non-neighbor connections (see `RPTRST` keyword item 2). Also, if an extended `GRID` file is requested, this will now include any non-neighbor connection data (see the `GRIDFILE` keyword).

- The well control mode numbers written to the Summary File (see Summary Section mnemonic `WMCTL`) are made negative if the well is under group control. For example, a well flowing at a user-defined liquid rate target will have a control mode number 4, but a well producing its share of a group's liquid rate target will have a control mode number -4. Also, two new well control mode numbers have been added: 11 = GOR Penalty control, 12 = drawdown control (previously these were output respectively as oil rate and liquid or gas rate control). If `RUNSUM` is specified to request a printed Run Summary output, the control mode numbers are replaced by character strings, unless Lotus output format has been requested.
- The generation of pinchout connections has been modified in runs which employ the `MINPV` or `MINPVV` keywords. Pinchout connections will only be generated if all the inactive cells between two active cells (in the Z direction) have either a combined thickness below the threshold set in the `PINCH` keyword or have a pore volume that falls below the `MINPV` criteria.
- The compressed vertical equilibrium model is not available in the 93A version. An improved version of the facility is planned for the 94A release.

New keywords

`COPYBOX`	Copies grid data from one box to another.
`FAULTS`	Defines fault trajectories, for subsequent editing of properties.
`FIP`	Inputs multiple sets of fluid-in-place regions.
`GCUTBACK`	Sets upper limits for group rate cut-back option.
`GRUPTARG`	Re-sets individual group production rate targets or limits.
`KRGR`, `KRORW`, `KRORG`, `KRWR`	Scales the relative permeabilities at the critical saturation of the displacing phase.
`MAPAXES`	Allows input of the pre-processor map origin.
`MULTFLT`	Modifies the transmissibility across faults defined using the `FAULTS` keyword.
`MULTPV`	Inputs pore volume multipliers.
`PBVD`	Inputs tables of initial bubble point pressure vs. depth.
`PINCH`	Sets the pinch-out threshold distance.
`SKIPREST`	Makes ECLIPSE skip the subsequent keywords in the `Schedule` section of a restart run until the restart time is reached.
`VEFRACP`	Controls the fraction of rock capillary pressure curve to VE pseudo capillary pressure.
`VEFRACPV`	Allows input of `VEFRACP` values on a cell-by-cell basis.
`WBOREVOL`	Sets wellbore volume for modelling wellbore storage in well tests.
`WORKTHP`	Performs a workover if well is unable to produce at its THP limit.
`WTHPMAX`	Sets maximum THP design limit data for wells.

New summary section mnemonics

- FMCTP, FMCTW, FMCTG

  Field's control mode for production, water injection and gas injection

- GMCTP, GMCTW, GMCTG

  Group control modes for production, water injection and gas injection
- FOPRF, GOPRF, WOPRF, COFRF

  Free oil production rates
- FOPRS, GOPRS, WOPRS, COFRS

  Vapourised oil production rates
- FOPTF, GOPTF, WOPTF, COPTF

  Free oil production totals
- FOPTS, GOPTS, WOPTS, COPTS

  Vapourised oil production totals
- FGPRF, GGPRF, WGPRF, CGFRF

  Free gas production rates
- FGPRS, GGPRS, WGPRS, CGFRS

  Solution gas production rates
- FGPTF, GGPTF, WGPTF, CGPTF

  Free gas production totals
- FGPTS, GGPTS, WGPTS, CGPTS

  Solution gas production totals
- FOVIS, ROVIS, BOVIS

  Field, region and grid block oil viscosity
- FWVIS, RWVIS, BWVIS

  Field, region and grid block water viscosity
- FGVIS, RGVIS, BGVIS

  Field, region and grid block gas viscosity
- FODEN, RODEN, BODEN

  Field, region and grid block oil phase reservoir density
- FWDEN, RWDEN, BWDEN

  Field, region and grid block water phase reservoir density
- FGDEN, RGDEN, BGDEN

  Field, region and grid block gas phase reservoir density
- MAXDPR, MAXDSO, MAXDSW, MAXDSG

  Maximum change in pressure and saturation
- ROP, RWP, RGP

  Net oil, water and gas production from a region, minus injection, and minus any water influx from analytic aquifers
- ELAPSED

  Elapsed time
- TCPU

CPU time

- TCPUDAY

  CPU time per day

- TCPUTS

  CPU time per timestep.

Altered keywords

DATE	Now causes dates to be written in the first column in the Run Summaries, instead of time.
ENKRVD	Now requires 8 columns of data rather than 4. The extra data enables scaling of the k_r at the critical saturation of the displacing phase.
GCONPRI	New item 11, sets an upper limit production balancing fraction.
GCONPROD	Additional control modes 'RESV' and 'PRBL' allowed in item 2.
	Additional guide rates for 'RES' and 'INJV' allowed in item 10.
	New item 14 sets target/limit reservoir fluid volume rate.
	New item 15 sets target/limit production balancing fraction.
GRIDFILE	Outputs non-neighbor connection data to the extended GRID file.
GRUPTREE	The keyword can now be used more than once in the simulation, to declare changes in the grouping hierarchy.
GUIDERAT	Additional option 'RES' for nominated phase in item 2.
	New item 9 to prevent calculated guide rates from increasing.
OPTIONS	Control 17 is now redundant (see the PINCH keyword).
	Control 18 is now redundant (see keywords VEFRAC, VEFRACV, VEFRACP, VEFRACPV).
	New control 40 to ignore THP limits when calculating well potentials.
RPTGRID	Control 19 and 20: New facility to output COORD and ZCORN keywords,
	Control 41: outputs pore volume multipliers,
	Control 42: outputs fault data.
RPTRST	Outputs non-neighbor connection flows also if control 2 is set.
	Outputs fluid viscosities if control 8 is set.
	Outputs phase reservoir densities if control 9 is set.
RPTSCHED & RPTSOL	
	If control 8 is set >2, fluid-in-place reports are and printed for all sets of fluid-in-place regions defined with FIP.
	Controls 48 to 50 allow output of fluid viscosities.
	Controls 51 to 53 allow output of phase reservoir densities.
RUNSPEC	Record 4: MFSEGS, the maximum number of records of fault data. Record 10: QVEO2D, not used for this release.
	Record 16: NMFIPR, the maximum number of sets of fluid-in-place regions.

Record 26: NSUMMX, the maximum number of Summary file quantities.

SUMMARY section

R* Keywords in the summary section can now be compounded with the name of the set of fluid-in-place regions.

TUNING Two new controls TRGDPR and XXXDPR added in record 3.

WCONHIST No longer sets the BHP limit to atmospheric pressure every time, so this limit can now be set with WELTARG.

WCYCLE New item 6, to control time step to coincide with cycling times.

WELSPECS The group name in item 2 will no longer be ignored after the first time a well is declared; the well can be re-assigned to a different group if required.

WGRUPCON A 'RES' guide rate phase is allowed for producers, as well as injectors.

WHISTCTL New item 2, to stop the run if a History Matching Well changes to BHP control.

WLIFT New item 7, setting water cut value to trigger lift switching.

WTEST Item 3 can now be up to 4 characters, containing a 'D' if you want the well to be tested for reopening after being closed for exceeding its maximum THP design limit.

ECLIPSE 100: developments in 92A

This section summarizes the changes to ECLIPSE between the 91A release and the 92A release.

New facilities

Grid block and fluid properties

- A facility has been added to allow modifications to be made to the grid data on a region-by-region basis. The regions are specified using the FLUXNUM keyword, and modifications to the GRID data can then be made using the EQUALREG, ADDREG and MULTIREG keywords. In addition the FLUXNUM array can be copied into the REGIONS arrays such as FIPNUM and SATNUM, if you want the grid block FIPNUM numbers for example to equal their FLUXNUM numbers. The grid data can be modified in this way independently of whether or not you are using the ECLIPSE 200 Flux Option, but you will need this ECLIPSE 200 option if you want to run reduced models of individual flux regions. (See the FLUXNUM, EQUALREG, ADDREG, MULTIREG and COPY keywords.)
- It is now possible to use the COORD and ZCORN keywords in conjunction with the BOX keyword. This is a useful facility when two or more models are combined into a single large model. Old models in which the COORD or ZCORN keywords are specified after a BOX keyword that has not been terminated with ENDBOX will now give an error message. This can be corrected by adding an ENDBOX keyword, or by moving the corner point data before the first BOX keyword. It is also possible to specify COORD data separately for each reservoir (if NUMRES>1). (See the COORD, ZCORN and RESVNUM keywords.)
- An alternative shape factor (SIGMAGD) can now be specified in three phase dual porosity runs. The transmissibility derived from the new shape factor is used in matrix cells in which the dominant flow process is oil-gas gravity drainage. Hence the two shape factors allow modelling of the different time constants (resulting from different flow paths) of cells under ‘imbibition’ and ‘gravity drainage’. (See SIGMAGD and SIGMAGDV keywords and the “Dual Porosity” chapter in the ECLIPSE Technical Description.)
- An option has been added to calculate the oil gravity terms using the region average oil reservoir density rather than the local density. (See control 38 in the OPTIONS keyword.)
- It is now possible to default saturation tables entered using SWFN, SGFN SOF2 and SOF3 keywords, provided the table is not the first. The subsequent tables default to the previous table.
- It is now possible to use the Killough wetting phase hysteresis model in both 3 phase and 3 component miscible runs. The wetting phase hysteresis model is applied to the water phase. (See the “Hysteresis” chapter in the ECLIPSE Technical Description.)

Tracers

- A new option has been added to the tracer tracking facility to reduce the numerical dispersion. The option employs a higher order space discretization and a limiting function to prevent unphysical oscillations. (See the “Tracer Tracking” chapter in the ECLIPSE Technical Description, and the TRACTVD keyword.)
- It is now possible to restart a case with more passive tracers present than were present in the base run. This also allows tracer cases to be run from non-tracer base runs.

Well model

- The engineer can supply coefficients for a general formula to calculate production well guide rates for group control. This can enable, for example, wells with high water cut or GOR to produce a progressively smaller share of the group's rate target. (See keyword GUIDERAT.)
- Pressure-dependent injectivity multipliers, that vary with the BHP, can be applied to injection wells to take approximate account of hydraulic fracturing by injected water. (See keyword WINJMULT.)
- The GOR Penalty Option provides a means of controlling the oil production rate target of individual wells on a monthly basis, as a function of their previous month's average GOR. It is specifically intended to model a production regulation imposed by regulatory bodies in certain countries. (See keyword WGORPEN.)
- The minimum economic limit checks in keyword WECON can optionally be applied to the wells' production potentials instead of their actual flow rates. They will not then be triggered when the well is cut back under group control. The option is activated with Item 10 in keyword WECON.

Output

- It is now possible to modify the orientation and layout of the printed grid arrays in the print file. (See the BOUNDARY keyword.)
- The initial fluids in place reports (activated with item 8 of keyword RPTSOL) now also show the average porosities and permeabilities of the grid blocks within the regions.
- Analytic aquifer - grid block connection data can be written to the print file by setting the 12th switch in RPTSOL to 2 or more (1 will just print data for the aquifers themselves, as before).
- The average saturations, R_s and R_v values over the field and individual reporting regions can now be written to the summary file.
- It is now possible to write out well, group and field tracer concentrations to the summary file. (See the SUMMARY section.)
- A number indicating a well's mode of control can be written to the summary file, using the mnemonic WMCTL.
- The run summaries generated by using the RUNSUM keyword can be written to a separate file, rather than appended to the print file. In addition the file can be output in a format suitable for importing directly into a spreadsheet program such as **Lotus 123**. (See the keywords SEPARATE and **Lotus**.)

Efficiency

- A new option has been added to improve the efficiency of runs in which a number of independent reservoir units are connected only by the production system. In these cases, the systems of linear equations for each reservoir are independent and may require different numbers of iterations. In addition the solution of the linear equations may be run in parallel on appropriate computers. (See keywords ISOLNUM and RPTISOL, and the "Independent Reservoir Regions" chapter in the ECLIPSE Technical Description.)

Changes to treatment of existing facilities

- The calculation of DX and DY from a corner point grid has been modified to account for grid cells which have been rotated relative to the X-direction. This change will not modify the cell pore volume or the transmissibilities. However the DX and DY values are used to calculate the well connection factor (via the Peaceman radius) and hence the results of existing models may be slightly modified.
- The way that ECLIPSE handles non-neighbor connections has been optimized so that the number of these connections is minimized. Where these connections lie between cells in the same line, i.e. in three dimensions from (x_1, y_1, z_1) to (x_2, y_2, z_2) for the cases
  - $x_1 = x_2$, $y_1 = y_2$ different z, or
  - $x_1 = x_2$, $z_1 = z_2$ different y, or
  - $y_1 = y_2$, $z_1 = z_2$ different x

  these non-neighbor connections can be treated more efficiently by adding the transmissibility to that between the cells and deleting them from the list of non-neighbor connections. The list of non-neighbor connections reported by RPTGRID is split into two parts, with the deleted NNCs being listed first, followed by the full set of active NNCs. The output of non-neighbor connections is controlled by switch 24 of keyword RPTGRID.
- As a result of the previous modification, the treatment of pinch-out connections is also modified, and ECLIPSE first reports them as distinct pinch-out connections before applying the above criteria and adding the calculated transmissibility to that (if any) between the cells that were joined by the pinch-out. The elimination of these connections generally results in a reduction in the required cpu time. A new keyword PINCHOUT in the GRID section gives an alternative method of selecting the generation of pinch-outs and can be used in place of OPTIONS switch 17. As stated in the previous modification, the list of deleted NNCs are reported first; and since pinch-out connections are treated as NNCs which are first generated then deleted, this is where they are reported.
- The transmissibility multiplier in the AQUCON keyword now defaults to 1.0 instead of zero.
- The method of setting up the linear equations for IMPES runs has been modified to improve the performance of the linear solution. This will speed up IMPES runs where the linear solution is difficult.
- If a well is in danger of being shut-in prematurely under THP control due to feedback effects with the implicit values of the water and gas fractions being used to interpolate the VFP table, the well will automatically switch to using the explicit values of the water and gas fractions to interpolate the tables. The user can also designate specific wells to use explicit values. (See keyword WVFPEXP for more information.)
- The output of dual porosity data to the print file has been modified. The printing of grid data now distinguishes between the matrix and fracture cells. The dual-porosity connections can be printed independently rather than with all the non-neighbor connections. (See the RPTGRID keyword.)
- The messages warning that VFP tables have been extrapolated have been improved, and now give information about the quantity and table last extrapolated. As before, the extrapolation warnings will only be given if they have been requested with the keyword EXTRAPMS.

New keywords

ADDREG	Adds a constant to a named GRID array for a given flux region.
EQUALREG	Sets a named GRID section array for a given flux region.
FLUXNUM	Defines the flux regions.
GUIDERAT	Sets coefficients for a general formula for well guide rates.
ISOLNUM	Identifies the extent of each independent reservoir region.
LOTUS	Requests run summaries be written in a **Lotus** format.
MULTIREG	Multiplies a named GRID array by a constant for a flux region.
PINCHOUT	Allows the generation of pinch-out connections where possible.
RESVNUM	Requests COORD input for a given reservoir.
RPTISOL	Generates a grid array of isolated reservoir numbers.
SEPARATE	Requests run summaries be written to a separate file.
SIGMAGD, SIGMAGDV	Specifies a separate shape factor for oil-gas cells undergoing gravity drainage.
TRACTVD	Requests the flux limiting scheme for tracer tracking.
WGORPEN	Sets data for the GOR Penalty Option.
WINJMULT	Specifies pressure-dependent injectivity multipliers.
WVFPEXP	Designates wells to use explicit WCT & GOR for VFP table lookup.

New summary section mnemonics

- WMCTL

  Well Mode of Control.
- FTIC, FTPC, GTIC, GTPC, WTIC, WTPC

  Tracer concentrations.
- FOSAT, FWSAT, FGSAT, FNSAT, FRS, FRV

  Field averaged solution.
- ROSAT, RWSAT, RGSAT, RNSAT, RRS, RRV

  Region averaged solution.

Altered keywords

AQUCON	Transmissibility multiplier (item 9) now defaults to 1.0.
BOUNDARY	Now allows different orientation and layout of printed grid data.
COORD, ZCORN	Can now be used with the BOX keyword.
COPY	Can now copy from FLUXNUM in the REGIONS section.
EHYSTR	Now allows wetting phase hysteresis in 3 phase runs.
OPTIONS	New options 38 and 39.

`RPTGRID` New switches 36,37,39, controlling output of flux region numbers, independent reservoir numbers, and dual porosity connections.

`RPTSOL` Setting switch 12 > 1 prints analytic aquifer connection data.

`RUNSPEC` Record 16: `NTFREG` the maximum number of flux regions for the ECLIPSE 200 Flux option, may now also be used for editing data in ECLIPSE 100. `NRFREG` sets the maximum number of independent reservoir units in the model.

Record 20: the redundant logical flag `QIMCOL` has been replaced by `QEXGOP` which enables the extended compressor option in the ECLIPSE 200 Gas Field Operations model.

`SWFN`, `SGFN`, `SOF2`, `SOF3`

Defaulted tables are now allowed.

`TBLK`,`TVDP`

Can now be specified in restart runs to initialize new tracers.

`WECON` New item 10 to apply minimum rate limits to the well's potential.

ECLIPSE 100: developments in 91A

This section summarizes the changes to ECLIPSE between the 90A release and the 91A release.

New facilities

Grid block & fluid properties

- Alternative keywords are available for entering saturation functions and oil PVT data:
  - Saturation function tables can be input with keywords `SWOF` and either `SGOF` or `SLGOF`, instead of `SOFN`, `SWFN` and `SOF3` or `SOF2`. The new set of keywords allow the user to enter the oil relative permeabilities in the same tables as the water and gas relative permeabilities, instead of in a separate table vs. oil saturation.
  - Keyword `PVCO` offers a simpler alternative to `PVTO` for entering live oil PVT data, if undersaturated oil with a particular R_s value has a compressibility and viscosibility that are independent of the pressure. The user enters these quantities instead of a table of undersaturated oil properties vs. pressure. Keyword `PMAX` must be used to enter the maximum expected pressure in the run.
- It is possible to assign a constant R_s or R_v value to each PVT region using the `RSCONSTT` and `RVCONSTT` keywords. This extends the constant R_s or R_v facility (see keywords `RSCONST`, `RVCONST`) to allow different regions of the reservoir to have different values of R_s or R_v. There must be no transmissibility between the PVT regions, to prevent the fluids mixing (otherwise the R_s or R_v values would change).
- A molecular diffusion option has been implemented.

  The diffusion option allows for diffusion of gas and oil components in both the liquid and vapor phases. Hence for example gas will diffuse from a region of low R_v to a region of high R_v, and vapourised oil will diffuse in the opposite direction. The diffusion process can be a significant recovery mechanism in some dual porosity reservoirs. See the “Molecular Diffusion” chapter in the ECLIPSE Technical Description.
- A brine option has been implemented to model the transport of waters with different salinities within a reservoir. A conservation equation for mass of salt within the water phase is solved at the end of each timestep. The salt concentration modifies the density and viscosity of the water phase for the next time step. See the “Brine Tracking” chapter in the ECLIPSE Technical Description.
- It is now possible to specify separate saturation function tables for the matrix-fracture connections using the `KRNUMMF` and `IMBNUMMF` in dual porosity runs.
- An option has been added to allow a problem to be equilibrated using drainage saturation curves but subsequently to run using the imbibition curves. (See the `EHYSTR` keyword).
- When using the End Point Scaling option it is now possible to scale the relative permeability values independently of the saturation values. See keywords `KRW`, `KRO`, `KRG` and `ENKRVD`.

- It is now possible to modify the saturation and PVT region numbers during the simulation. This has previously been possible by restarting a run with a modified regions section; but if many changes are required it is more convenient to respecify SATNUM (for example) within the SCHEDULE section. This procedure is **not** recommended in general. See the keywords SATNUM, KRNUM, IMBNUM, ROCKNUM and MISCNUM.
- A minimum pore volume can be specified, which a cell must have in order to be included in the simulation. This defaults to 1E-6, which was the pore volume criterion for active cells in previous versions. The value can be set for the entire grid using the MINPV keyword, or for sections of the grid using the MINPVV keyword.
- It is now possible to specify negative depths, i.e. the grid can be above or straddle the zero depth reference line. ECLIPSE detects the presence of negative depths via negative values in the TOPS or ZCORN keywords, and then removes the restrictions on positivity for equilibration depths and datum depths. The action of negative values in the WELSPECS keyword is altered so that they are treated as the actual depth, but default values can still be obtained by using the usual '1*' notation. All tables of functional variation against depth are subject to the usual constraints, i.e. depths must be monotonically increasing.
- A new option has been added for specifying inter-block flows when the Vertical Equilibrium option is used, in conjunction with a grid entered using block centred geometry. It is activated by setting the 19th OPTIONS switch to 2. (See the OPTIONS keyword and the "Vertical Equilibrium" chapter in the ECLIPSE Technical Description.)

Well model

- There is a facility to account for changing separator conditions in black oil runs. Keyword SEPVALS is used firstly to define the initial separator conditions, and subsequently to change them. When the separator conditions have been changed, ECLIPSE transforms the oil and gas flow rates of the wells and groups that feed into the separator. Groups are assigned to separators with keyword GSEPCOND.
- Extensions to the well drawdown control facility (keyword WELDRAW). This facility can now be applied to gas wells; the user can select the phase (liquid or gas) for which ECLIPSE converts the drawdown limit into a rate limit. Also, the user can request that drawdown limits should be taken into account when calculating well production potentials.
- A constant user-defined relative permeability of the injected phase can be specified for injection well connections with keyword COMPINJK. This enables the injectivity to be calculated more accurately in cases where, for example, gas is injected into a large grid block initially containing oil.
- Scaling well P.I.s.

  Well connection transmissibility factors can be multiplied by a specified value, using keyword WPIMULT.
- Over-riding control mode for history matching wells.

  Keyword WHISTCTL can be used to specify a control mode for history matching wells, which will over-ride the mode set in subsequent WCONHIST keywords.

Output

- Oil recovery efficiencies are calculated for the field and each report region. There are six quantities calculated:
  - (OIP(initial) - OIP(now)) / OIP(initial)
  - oil production from wells / OIP(initial)
  - (OIP(initial) - OIP(now)) / initial mobile oil (w.r.t. water)
  - oil production from wells / initial mobile oil (w.r.t. water)
  - (OIP(initial) - OIP(now)) / initial mobile oil (w.r.t. gas)
  - oil production from wells / initial mobile oil (w.r.t. gas)

  The two mobile oil values allow for different critical saturations of oil-water and oil-gas. When directional scaling is used, the **lowest** critical saturation is chosen, i.e.

  `SOWCR` = Min(`SOWCRX`, `SOWCRX-`, `SOWCRY`, `SOWCRY-`, `SOWCRZ`, `SOWCRZ-`).

  `SOGCR` is chosen similarly. The mobile oil values are therefore the maximum possible.

  The recovery efficiencies are printed in the fluid-in-place reports, and can be written to the summary file with the keywords:

  `FOE`, `FOEW`, `FOEIW`, `FOEIG`, `FOEWW`, `FOEWG`,

  `ROE`, `ROEW`, `ROEIW`, `ROEIG`, `ROEWW`, `ROEWG`.
- The number of non-linear iterations taken for each time step can be written to the summary file (see the `NEWTON` keyword).
- Extra information can be written to the summary spec file to allow the GRAF program to monitor an ECLIPSE simulation while it is running. (see the `MONITOR` keyword).
- It is now possible to output the water and gas phase pressures to the restart file to be displayed in GRAF (see the `RPTRST` keyword).
- Inactive cell information can be written to the `GRID` file (see the `GRIDFILE` keyword).

Timestep control

- The next timestep can be restricted to a specified size, using the `NEXTSTEP` keyword. This is a simpler alternative to using the `TUNING` keyword to control this value.
- Throughput ratios are now used as a new timestepping control to speed up IMPES runs. This may cause slightly different results to be obtained from 91A unless `THRUPT` is set to a high value in keyword `TUNING`.
- `OPTIONS` switch 32 can be used to limit the number of consecutive times a single timestep can be chopped due to a well initially on rate control switching to pressure control and dying in the same timestep. The default allows only one chop. Chops can be prevented by supplying a negative value.

Changes to treatment of existing facilities

- In the `WCONHIST` keyword, defaults to item 7 (VFP table number) and item 8 (ALQ value) now mean “no change from well’s previous value” instead of “zero”. This saves having to specify these items continually if THP values have to be calculated in a history match.

- The handling of fault-generated non-neighbor connections has been modified to use the appropriate relative permeability table when the directional relative permeability option is being used. This may result in some different results compared to previous versions of ECLIPSE.
- A reduced set of Grid data is carried during the simulation, which eliminates the reading/writing to disk of the full Grid data set. For large models this should improve the elapsed CPU times. It imposes a small overhead in storage requirements, but the previous disking procedure can be enabled by setting `IDYNAM` = 1 in `RUNSPEC` record 30.
- The tracer tracking disking option is now by default off. If tracer disking is required, for example in runs with many tracers, then the disking can be used by setting `QTDISK` to true in `RUNSPEC` record 16.
- Cells in 2-phase runs that cross the saturation pressure (see keywords `RSCONST` or `RVCONST`) are now listed on the print file prior to the run stopping.
- Improvements have been made in the property table look-up routines which can lead to faster execution on vector machines. A reduction in the required cpu time of 20-30% is possible for large 3-phase simulations.

New keywords

COMPINJK	Constant user-defined injection well relative permeabilities.
DIFFC	Specifies the diffusion data, component molecular weights and diffusion coefficients for the molecular diffusion option.
DIFFDP	Requests the diffusion calculation for matrix-fracture flow only in dual porosity runs.
DIFFMX, DIFFMY, DIFFMZ, DIFFMR, DIFFMTHT, DIFFMMF	Specifies multipliers on the calculated diffusivities.
DIFFX, DIFFY, DIFFZ, DIFFR, DIFFTHT	Over-writes the calculated diffusivities in the EDIT section.
ENKRVD	Specifies the depth variation of the end-point rel-perm values.
GRIDFILE	Controls output to the GRID file.
GSEPCOND	Assigns separators to groups, for changing separator conditions.
IMBNUMMF	Specifies the matrix-fracture saturation tables for imbibition processes, if the hysteresis option is active.
KRG	Specifies the end-point gas relative permeability for a block.
KRNUMMF	Specifies the matrix-fracture saturation table number.
KRO	Specifies the end-point oil relative permeability for a block.
KRW	Specifies the end-point water relative permeability for a block.
MINPV	Specifies a minimum pore volume that a cell must have to be included in the calculation.
MINPVV	Specifies a MINPV for each cell.
MONITOR	Requests information required by GRAF for the run-time monitoring option to be written to the summary file.
NEWTON	Requests output of the number of non-linear iterations to the summary file.
NEXTSTEP	Restricts the size of the next timestep.
PMAX	Maximum pressure expected during the simulation.
PVCO	Tables of saturated oil properties and compressibilities.
PVTWSALT	Specifies the water PVT data in the presence of salt.
RSCONSTT	Sets a constant R_s value for each PVT table region.
RVCONSTT	Sets a constant R_v value for each PVT table region.
SALT	Specifies the initial salt concentration for enumerated starts.
SALTVD	Specifies the initial salt conc vs. depth for equilibrated starts.
SEPVALS	Separator test results at original and new conditions.
SGOF	Tables of gas/oil saturation functions v gas saturation.
SLGOF	Tables of gas/oil saturation functions v liquid saturation.
SWOF	Tables of water/oil saturation functions v water saturation.
WHISTCTL	Sets over-riding control mode for history matching wells.
WPIMULT	Multiplies well connection factors by a specified value.
WSALT	Specifies the salt concentration in a water injector.

New summary section mnemonics for oil recovery efficiencies

- FOE, FOEW, FOEIW, FOEIG, FOEWW, FOEWG,

  ROE, ROEW, ROEIW, ROEIG, ROEWW, ROEWG

  Oil recovery efficiencies.

Altered keywords

AQUCT, AQUFET & AQUFETP

The initial aquifer salt concentration can be specified for runs using the Brine option.

ACTNUM

This keyword now works with the currently defined box and will allow previously input values to be overwritten. In dual porosity cases information is only required for the matrix **or** the fracture cells. ECLIPSE will automatically copy a set of values for the matrix cells into the fracture cells and vice versa. This is done to ensure consistency between matrix and fracture cells controlled via this keyword. The selection of an active matrix cell coupled to an inactive fracture cell should be done through the physical grid data (for example. Porosity). ECLIPSE will disregard data if more values than expected from the current BOX limits are supplied.

DEBUG

New switches 34 and 35.

EHYSTR

New option added to equilibrate with a drainage curve and simulate with an imbibition curve.

OPTIONS

Control 19 includes a second option for VE cases.

Control 29 is now not used.

Control 31 redefines average pressure output.

Control 32 limits timestep chops due to wells dying.

RPTGRID

New switches 27 to 33 for diffusivities.

New switch 34 to output K/D values (ECLIPSE 200 surfactant model).

New switch 35 to output a list of cells having less than the minimum pore volume to be included in the calculation.

RPTPROPS

New switch 18 to output diffusion coefficients

New switch 19 to output surfactant properties (ECLIPSE 200 surfactant).

RPTREGS

New switches 22 and 23 to output matrix-fracture saturation function regions.

New switch 24 to output surfactant regions (ECLIPSE 200 surfactant model).

RPTRST

New switch 7 to output water and gas phase pressures.

RPTSCHED

Items 35 and 38 control the output of the salt concentration and salt fluid in place reports for the Brine option.

RPTSOL

Items 29, 35 and 38 control the output of the initial salt concentration and initial salt fluid in place reports for the Brine option.

RUNSPEC	Record 6: QBRINE now controls the brine option as well as the salt-sensitive polymer option. QSFACT controls the Surfactant option (ECLIPSE 200). Record 10: new item QMDIFF set T for the diffusion option. Record 16: new item QTDISK controlling tracer disking. Record 18: new item LSTACK, sets the length of the stack of previous search directions for local grid simulations. (ECLIPSE 200 Local Grid Refinement only).
TUNING	New item 9: THRUPT provides an extra timestepping control, primarily intended for IMPES runs.
WCONHIST	Defaults to items 7 and 8 now mean “no change from previous value” instead of “zero”.
WELDRAW	New items 3 and 4 added, to select the rate control phase and to request that drawdown limits should be taken into account when calculating well production potentials.

ECLIPSE 100: developments in 90A

This section summarizes the changes to ECLIPSE between the 88/09 release and the 90A release.

New facilities

- Lumping of well connections into completions for simultaneous closure in automatic workovers. Keyword COMPLUMP can be used to lump well connections together into completions. When the well is worked over (e.g. after exceeding a water cut limit set in keyword WECON), the worst-offending completion is identified (by summing the production rates of its connections) and all the connections belonging to the completion will be shut together.
- Automatic plug-backs and calculation of partial penetration skin in VE runs. In runs using the Vertical Equilibrium option, wells may be perforated over just part of the thickness of a grid block (see keyword COMPVE). There is now an option to re-calculate the skin factor and transmissibility factor for a well connection, taking account of partial penetration. There is also an option to plug back the perforations from the top or bottom when a flow or economic limit is violated (see keywords WPLUG, WECON, GECON, GCONPROD, GCONPRI and GCONSALE).
- Gas imports. A group (or the field) can import gas at a specified rate from outside. Imported gas will be taken into account when performing gas re-injection and when calculating the sales gas rate. The gas import rate is set with keyword GCONSUMP.
- Special gas producers for sales gas control. Wells can be declared to be special gas producers (see keyword WGASPROD). When the sales gas rate falls below its minimum limit (see keyword GCONSALE), the drilling queue is scanned and a special gas producer will be opened preferentially to other producers in the drilling queue. When a special gas producer is opened, its gas production rate is stepped up by user-defined increments when necessary to meet the sales gas requirements. By designating some high-GOR wells as special gas producers, sufficient gas can be produced to meet the sales gas requirements independently of any oil production target.
- Additions to the periodic testing option for closed wells. Wells that are closed automatically during the simulation can be tested periodically to see if they can be re-opened (see keyword WTEST). This option has been extended in two ways. Firstly, the user can set an upper limit to the number of times a given well can be tested. Secondly, when a well is re-opened after a successful test, its efficiency factor can be ramped up steadily over a given interval of time, to avoid convergence problems when a high-rate well is suddenly turned on.
- Automatic cycling of wells on and off. There is a facility to cycle wells on and off automatically for specified intervals of time (see keyword WCYCLE). The on-period and off-period can be set independently for each well. When a cycling well is turned on, its efficiency factor can optionally be ramped up over a set interval of time, and there is an option to restrict the time step length.
- Well cut-back reversal limits. Lower limits on well water cut, gas-oil ratio, gas-liquid ratio and water-gas ratio can be set to reverse a previous rate cut-back process (see keyword WCUTBACK).

- History Matching Wells. Production wells can be placed under a special type of control to aid the process of history matching. Their observed oil, water and gas rates are entered with the keyword WCONHIST. The well can be made to produce at a rate which matches one of the phases, or at the equivalent liquid or reservoir volume rate of the observed production. The latter option is useful for making the well produce the correct amount of total fluid from the reservoir before the mobility ratios are fully matched. Thus the rate of pressure decline should be approximately correct. The observed rates and production ratios can be written to the summary file, for graphical comparison with the calculated rates.
- Calculating reservoir fluid volume rates at local conditions. A well's reservoir fluid volume rate is based on the volume that the produced or injected mixture would occupy at the average hydrocarbon conditions (pressure, R_s, R_v) in the reservoir. There is now an option to base the calculation on the average hydrocarbon conditions in the fluids-in-place region in which the well is completed. This option is useful if the field is divided into zones of significantly different pressures. The fluids-in-place region is set in item 13 of the WELSPECS keyword.
- An alternative restart facility has been introduced to save computer time when performing a 'simple' restart. The processed base case data is stored in a SAVE file, and includes all the data required for the run excluding the time-dependent solution, which is stored as usual in the restart file. Computer time is saved, as computed data such as pore volumes and transmissibilities do not need to be recalculated. The existing restart facility has been retained because this allows greater flexibility to change the input data on a restart. (See the SAVE, LOAD and RESTART keywords.)
- Reservoir drainage regions may be written to the debug file by setting the 25th OPTIONS control greater than zero. These are useful for the ECLIPSE 200 flux boundary option. Flows are minimal between different drainage regions. Thus if the drainage regions are reasonably stable over a period of time, they can be used with the flux boundary option to save computing time in sensitivity or history matching runs.
- A facility has been added to allow a model to be run assuming the reservoir fluids are in vertical equilibrium across the entire reservoir section. This is in contrast to the 3D vertical equilibrium option in which a single grid cell is assumed to be in vertical equilibrium. A column of grid cells is represented by a single cell covering the full reservoir section. Variations in rock properties for the different layers are preserved in the 'compressed cell'. Efficiency is gained by reducing the problem to 2D and only solving for one set of solution variables at each X, Y grid block. The option is initiated by setting QVEO2D = T in the 10th record of the RUNSPEC section, for example:

```
RUNSPEC
.
.
= NRPVT NPPVT NTPVT NTROCC QROCKC QRCREV QROCKH QVEO2D
   20    20     1     1      F      F      F      T      /
```

- A 9-point non-orthogonal pressure correction option has been added. This uses a finite-volume based discretization scheme, which introduces extra pressure derivative terms into the Jacobian, to represent the effects of cell geometry and pressure on fluid flows. Compared with the standard 5-point scheme, this gives improved accuracy for distorted grids, but increases storage and run time. The option is selected by setting QOPT9P = T in the 30th record of the RUNSPEC section, e.g.:

```
RUNSPEC
.
.
= QSOLVE NSTACK QFMTOU QFMTIN QUNOUT QUNINP NGDISK IDYNAM QOPT9P
     T     50     F      F      F      F      0      0      T   /
```

- The miscible flood model is now available as an ECLIPSE 100 option. In previous releases of ECLIPSE the miscible flood model has been an ECLIPSE 200 feature. The model is a implementation of the Todd and Longstaff empirical model described in [Ref. 24]. (See the "Miscible Flood Modelling" chapter in the ECLIPSE Technical Description.)
- A new option has been added to allow capillary pressures to vary with oil pressure. The user supplies tables of oil/water and oil/gas surface tension as a function of oil pressure. The capillary pressure curves are supplied as usual (in SWFN, SGFN) and are assumed to apply at a specified reference pressure. At other pressures, the capillary pressure is adjusted in proportion to the surface tension variation. See the STOG and STOW keywords for more details.

Modifications to existing facilities

- The two strategies for group control - prioritization and guide rates - can now both be applied in the same run. Groups to be controlled by prioritization should have their rate limits set with keyword GCONPRI, while groups using the guide rate method have their rate limits set with keyword GCONPROD. A group can be changed from one method to the other simply by re-specifying its rate limits with the appropriate keyword. The mixing of the two methods is subject to the restriction that the guide rate method cannot be used by a group subordinate to a group using the prioritization method. Wells from the drilling queue that are not subordinate to a prioritization group will be opened when necessary to meet the target rate of a group under guide rate control. See keyword PRIORITY for more details.
- The minimum rate limits for wells in keywords WECON and WLIFT are not applied to a well while it has its rate cut back by the prioritization option to meet a group rate limit. This is to prevent the well being prematurely closed or put on artificial lift when it is producing at a low rate so as not to exceed a group's production rate limit.
- The identity of the "top-up" phase for group injection control can be changed during the run, if required. Setting a non-default value for Items 5 or 7 in keyword GCONINJE for a particular injected phase, or selecting NETV in Item 10, defines that phase as the "top-up" phase. There can only be one "top-up" phase, as the injection targets for that phase can only be determined after the injection rates for any other phases have been calculated. It is now possible to change the "top-up" phase during the simulation. But at any given time there must be only one "top-up" phase, so different groups cannot have different "top-up" phases.

- When a well is opened, during its first time step the wellbore head terms (the hydrostatic pressure difference between the connections and the bottom hole reference depth) were based on a very approximate estimate of the wellbore contents. Now the initial wellbore head terms are based on a more accurate estimate of the fluid mixture in the wellbore. This may result in a small change in the connection flow rates during a well's first open time step, when compared with earlier versions of the program. The change may be more noticeable for wells under THP control, especially if there is a large difference between the bottom hole reference depth and the VFP table's bottom hole datum depth.
- The VFP table number for a well can now be reset with the `WELTARG` keyword.
- The default BHP limit for injection wells has been changed from 1E20 to 1E5 psia (which should still be high enough not to be exceeded in realistic simulations). The change will make the calculation more robust if the well is temporarily switched to BHP control for any reason.
- The method of specifying optional data to be written to the restart file has been improved. The following optional information can be written to the restart file in addition to the minimum required for a restart:
  - Inter-block flows
  - Fluids-in-place
  - Phase potentials
  - Bubble point and Dew point pressures.

  Output of these parameters can be independently controlled using the `RPTRST` keyword. This keyword can also be used to apply greater control on the timing of restart file dumps.
- The automatic scaling of critical end-points has been extended to 3-phase cases with the flexibility to apply the option to each phase independently (see the `TZONE` keyword).
- Default relative permeability and capillary pressure values can be specified in the `SWFN` and `SGFN` keywords. If a value is defaulted the table is linearly interpolated to supply the missing value.
- The `OPTIONS` keyword has been extended to limit the number of 'Cell overlap' messages printed, to provide a drainage regions facility and to stop the run when a non-converged timestep is too small to be chopped.
- Memory requirements have been reduced for data checking runs by saving the solver workspace which is not required if `QSOLVE` = F. This facility is useful if data is prepared on a machine with limited memory. The data can then be transferred to the larger machine for the full simulation run.
- When using capillary pressure hysteresis, an option has been provided to chop the saturations when large saturation changes alter the hysteresis state. This option can improve non-linear convergence in some cases; the greatest effect is seen in dual porosity runs. (See the `OPTIONS` keyword).
- For enumerated initial solutions, the initial pressure can be specified as a pressure v depth table. This is an alternative to specifying the pressure on a block-by-block basis. (See the `PRVD` keyword).
- The summary information written to the summary files can be reduced by requesting that the summary data is only written to file at report times. (See `RPTONLY` keyword).
- The date can now be output to the summary files (See `DATE` keyword).
- A check is now carried out on the undersaturated oil PVT data supplied in the `PVTO` keyword to prevent the undersaturated lines intersecting. The lines should not intersect, and if they do ECLIPSE will experience convergence problems.

- An option has been added to suppress the use of disking when the Tracer tracking option is used. This option will have the effect of speeding up the run (reducing the elapsed time rather than the CPU time), at the expense of using more memory. (See `OPTIONS` keyword.)
- The end-point scaling option can now be used in conjunction with Stone's method (1 and 2) for calculating 3 phase relative permeability.

New keywords

`COMPLUMP`	Lumps connections together into completions.
`LOAD`	Initiates a run from a previous `SAVE` file.
`MISCNUM`	Specification of the miscible region numbers.
`PRVD`	Sets the initial pressure as a function of depth.
`RPTRST`	Controls output to the restart file.
`SAVE`	Requests that a `SAVE` file is written.
`SDENSITY`	Input of the surface density of the solvent in the miscible model.
`SORWMIS`	Tables of residual oil saturation vs. water saturation for the miscible model.
`STOG`	Specification of oil/gas surface tension vs. pressure.
`STOW`	Specification of oil/water surface tension vs. pressure.
`TLMIXPAR`	Input of the Todd-Longstaff mixing parameter for miscible flood.
`TZONE`	Controls the automatic scaling of critical end-points in a transition zone (replaces `OPTIONS` switch 12).
`WCONHIST`	Observed flow rates for History Matching Wells.
`WCYCLE`	Instructions for automatically cycling wells on and off.
`WGASPROD`	Declares production wells to be special gas producers.
`WPLUG`	Sets plug-back lengths for wells subject to automatic plug-backs. model.

New summary section keywords:

- Field, group and well **liquid** production rates and totals

  (`FLPR`, `FLPT`, `GLPR`, `GLPT`, `WLPR`, `WLPT`).
- Observed flow rates and production ratios of History Matching Wells

  (`WOPRH`, `WWPRH`, `WGPRH`, `WLPRH`, `WWCTH`, `WGORH`, `WOGRH`, `WWGRH`).

`DATE`	outputs the date to the summary file.
`RPTONLY`	reduces frequency of summary output.

Altered keywords

`COMPVE`	New items 10 - 13, governing re-calculation of skin factor and connection transmissibility factor to take account of partial penetration.
`GCONPRI`	Extra option 'PLUG' in items 3, 5, 7 and 9 for automatic plug-back.
`GCONPROD`	Extra option 'PLUG' in items 7, 11, 12 and 13 for automatic plug-back.

GCONSALE	Extra option ‘PLUG’ in item 5 for automatic plug-back. Extended sequence of actions if sales gas rate falls below lower limit set in item 4.
GCONSUMP	Extra item 3 specifying gas import rate.
GECON	Extra option ‘PLUG’ in item 7 for automatic plug-back.
OPTIONS	Control 21 redefines Liquid as Oil + Gas.
	Control 22 limits the number of well hunting messages allowed.
	Control 23 to limit ‘Cell overlap’ messages.
	Control 24 for overburden pressure in the rock compaction option.
	Control 25 to write drainage regions to the debug file.
	Control 26 to help prevent negative gas saturations.
	Control 27 to stop the run when a timestep is too small to chop.
	Control 28 initiates capillary pressure hysteresis chops.
	Control 29 suppresses tracer disking.
	Control 30 to impose binary chop on well iterations.
QDRILL	Can now be used if some groups use the prioritization option. Drilling queue wells will still be opened as required under guide rate groups.
RUNSPEC	Record 6: QMISCL - set .T. to use the miscible flood option (formerly an ECLIPSE 200 option).
	Record 10: new items QVEO2D and QSURFT to activate Vertical Equilibrium compression to 2 dimensions and pressure-dependent capillary pressure tables.
	Record 12: QTWOPT, NSMISC and NTMISC - miscible flood option (formerly an ECLIPSE 200 option).
	Record 20: the redundant quantity NWCOLC (item 2) has been replaced by NWFRIC, the maximum number of wells having wellbore friction. This is an ECLIPSE 200 facility; see the chapter on ECLIPSE Features for more details. A warning message will be printed if the value of NWFRIC is set > 0 in runs with no friction wells, as this increases the storage requirements.
	Record 30: new item QOPT9P to activate 9-point scheme.
SGFN	Default values are now allowed.
SWFN	Default values are now allowed.
WCUTBACK	New items 10 to 13, setting cut-back reversal limits.
WECON	Extra option ‘PLUG’ in item 7 for automatic plug-back.
WELSPECS	New item 13 setting the fluids-in-place region number for calculating the well’s reservoir fluid volume rate.
WELTARG	The VFP table number can now be reset with this keyword.
WTEST	New items 4 and 5 to set the maximum number of tests and the start-up time.

ECLIPSE 100: developments in 88/09

This section summarizes the changes to ECLIPSE between the 88/09 release and the 87/11 release.

New facilities

- Pinchout connections are now automatically generated in the z-direction if one or more inactive layers have zero thickness (See the `OPTIONS` keyword).
- Connection transmissibility factors can now be calculated for horizontal wells. The direction in which the well penetrates the grid block is specified in the `COMPDAT` keyword, and the appropriate components of permeability and block dimensions are substituted into the Peaceman formula. It is still assumed that the well completely penetrates a rectangular grid block through its centre, perpendicularly to two of its faces. This option is useful for horizontal wells.
- The Rock Compaction Option has been extended to allow hysteresis. A two dimension table of pore volume and transmissibility multipliers can be specified using the `ROCKTABH` keyword. In addition an overburden pressure can be specified as a function of depth. This pressure is subtracted from the pore pressure when interpolating the compaction multipliers (see `OVERBURD` keyword).
- An edit facility has been added to modify the non-neighbor connections generated from faults in the grid (See the `EDITNNC` keyword).

Modifications to existing facilities

- `SIGMA` and `DZMTRX` parameters can now be entered on a cell-by-cell basis, in dual porosity runs. See keywords `SIGMAV` and `DZMTRXV`. The interpretation of `DZMTRX` in dual porosity runs has been altered to give better agreement with observations. Older versions of ECLIPSE will give different results from 8809 unless `DZMTRX` is reduced by a factor of 2.
- The `VEFRAC` parameter can now be entered on a cell-by-cell basis in V.E. runs. See keyword `VEFRACV`.
- The `VEFRAC` parameter default now applies only to the relative permeabilities and not to the capillary pressure. An option has been added to apply `VEFRAC` to the capillary pressure (see `OPTIONS` keyword).
- The well economic limit checks are bypassed for wells subordinate to groups with a zero rate target. This prevents wells being shut-in permanently when their platform is temporarily shut down.
- For wells that produce mainly gas and mainly oil at different stages of their life, closure or workover due to high water cut can be prevented whilst the well is a gas producer by setting Item 5 negative in keyword `WECON`.

- When using the Vertical Equilibrium Option (VE), the calculation of flow across a face has been modified to account for the position of the adjacent cell. In addition when non-neighbor connections are generated from geological faults, the relative geometry of the two cells connected by the non-neighbor connection is accounted for in the VE flow. An option to use the original method has been retained (`OPTIONS` keyword).
- When using the VE option and entering non-neighbor connections explicitly, it is now possible to specify the flowing face for the non-neighbor connection (See the `NNC` keyword)
- New controls have been added to the `TUNING` keyword to ensure that the non-linear equations are converged to within user specified pressure and saturation tolerances. This option is not recommended for general use.
- The `NEWTRAN` method of calculating the transmissibilities has been made the default method for grids specified using the `COORD` and `ZCORN` keyword.
- A problem has been corrected that affects Impes cases using directional relative permeabilities.

New keywords

`SIGMAV`	Specifies `SIGMA` by cell.
`DZMTRXV`	Specifies `DZMTRX` by cell.
`VEFRACV`	Specifies `VEFRAC` by cell.
`ROCKTABH`	Specifies Hystereic rock compaction data.
`OVERBURD`	Specifies the rock overburden pressure as a function of depth for the rock compaction option.
`EDITNNC`	Keyword to modify a non-neighbor connection.

Altered keywords

`COMPDAT`	New item 13 specifies direction in which well penetrates block.
`OPTIONS`	New switches 16 - 20.
`WECON`	Item 5 can be set negative to prevent closure or workover due to high water cut whilst the well is a gas producer.
`RPTSOL`	New switch 41.
`RPTSCHED`	New switch 41. Altered switches 11,12,15 (3 = shortform only).
`TUNING`	New controls `DDPLIM` and `DDSLIM` to ensure that the non-linear equations are converged within user specified pressure and saturation tolerances.

ECLIPSE 100: developments in 87/11

This section summarizes the changes to ECLIPSE between the 87/1 release and the 87/11 release.

New facilities

- Increased speed on both vector and scalar machines achieved by increased vectorisation and a workcount reduction in nested factorization.
- Memory management. ECLIPSE 8711 requires less memory per active cell than ECLIPSE 871. In addition, new disking options have been introduced to allow large studies to be performed on machines with limited real memory. See the "Disking Options" chapter in the ECLIPSE Technical Description.
- Prioritization of wells to meet group production targets. This is an alternative to the existing method of apportioning group production targets among the wells in proportion to their guide rates. In the prioritization option, wells are turned on in decreasing order of priority, the well with the highest priority flowing first, until a group's production rate limit is exceeded. The well that exceeds the group's rate limit is cut back to meet the limit, and the wells with priority lower than this do not flow. The priority number for each well can be calculated from their potentials using a general equation (keyword `PRIORITY`) or can be specified by the user (keyword `WELPRI`). The group rate limits are set using keyword `GCONPRI`.
- A maximum allowable drawdown can be specified for each production well, using the keyword `WELDRAW`. This is converted into a maximum value for the well's liquid production rate at each time step, using the current values of the liquid mobility in the well's connections.
- Multiple workover rigs. Workover rigs can be allocated to individual groups, using the keyword `GRUPRIG`. The time taken for each well to be worked over is set as before with keyword `WORKLIM`, and the appropriate rig is made unavailable for other workovers for this duration. A workover on a particular well will be postponed if its workover rig is occupied until the end of the time step. If the keyword `GRUPRIG` is omitted, there is assumed to be only one rig for the whole field, as in earlier versions of the program.
- A flag can be set with the keyword `WDRILRES` that prevents a well in the drilling queue or on automatic opening from being opened if it is completed in the same grid block as a well that is already open.
- A minimum oil saturation can be set with the keyword `WELSOMIN`. A connection will not be opened automatically in a workover, or opened when its well is opened automatically or from the drilling queue, if the connecting grid block has an oil saturation that is less than this.
- A summary of the parameters set in `RUNSPEC` section can be printed using `RPTRUNSP` keyword.

Modifications to existing facilities

- The 'End Run' flag for wells (keyword WECON) and groups (keyword GECON) will not be active until the well or group has produced some fluid. The flag can therefore be set before the wells are opened without immediately stopping the run.
- If a well has an oil or gas production rate less than the minimum economic limit set in keyword WECON, but still has some unopened connections left in its opening queue which meet the minimum oil saturation requirement (if imposed with keyword WELSOMIN), one of these connections will be opened automatically as soon as a workover rig is available, instead of the well being shut-in.

New keywords

GCONPRI	Sets group and field rate limits for prioritization option.
GRUPRIG	Assigns workover rigs to groups.
PRIORITY	Activates well prioritization option and sets priority coefficients.
RPTRUNSP	Prints parameters set in RUNSPEC section.
WDRILRES	Sets flag to stop wells being automatically opened if another open well is completed in the same grid block.
WELDRAW	Sets maximum allowable drawdown for wells.
WELPRI	Sets priority numbers for wells.
WELSOMIN	Sets minimum oil saturation for automatic opening of a completion.

New summary section keywords

- (F,G,W)(O,W,G)P(P,I)

  (Field, Group, Well) (Oil, Water, Gas) Potential (Production, Injection).
- CPR, CWCT, CGOR, COGR, CWGR

  pressure, water cut, gas-oil ratio, oil-gas ratio, water-gas ratio at connections.

Altered keywords

OPTIONS	New actions for switches 3 and 15.
RPTPROPS	New actions for switches 1-6.
RPTSCHED	New switch 39 for printing TUNING parameters.

ECLIPSE 100: developments in 87/1

This section summarizes the changes to ECLIPSE between the 86/1 release and the 87/1 release.

New Facilities

- Well opening on closure of another well.

  The user can nominate a well to be opened when another specified well is shut automatically (e.g. after violating an economic limit). This facility can be used, for example, to open an injector when a producer closes after becoming uneconomic. If the injector has the same completions as the producer, the effect will be equivalent to converting the producer into an injector, although in this case they are regarded as being two separate wells. The next well to open is nominated in keyword `WECON`.

- Maximum rate of drilling new wells.

  The time taken for individual wells to be drilled can be set in keyword `WDRILTIM`. No well can be opened automatically (see 'Automatic Opening of New Wells') or opened from the drilling queue while another well is being drilled. This limitation does not affect wells that are opened 'manually' (e.g. with keyword `WELOPEN` or `WCONPROD`).

- Maximum number of open wells in a group.

  A limit on the number of open wells subordinate to each group can be set in keyword `GECON`. No well can be opened automatically (see 'Automatic Opening of New Wells') or opened from the drilling queue while its group or any superior group has a full complement of open wells. This limitation does not prevent wells from being opened 'manually' (e.g. with keyword `WELOPEN` or `WCONPROD`).

- Automatic opening of new wells.

  An extra choice '`AUTO`' has been added to the `OPEN/SHUT/STOP` flag in keywords `WCONPROD`, `WCONINJE`, `WCONINJ` and `WELOPEN`. Wells placed in this state are initially `SHUT`, but will be opened automatically as soon as the restrictions described in 2 and 3 above will allow. They will be opened in sequence in the order that they are first specified in keyword `WELSPECS`, provided there is room for another open well in their groups, at a rate limited by their individual drilling times. This facility is completely separate from the drilling queue (keyword `QDRILL`), which opens new wells only when needed to maintain group flow targets.

- Temporary closure of wells during drilling and workovers.

  A flag can be set in keyword `WDRILTIM` to treat wells as shut while they are being drilled automatically (see 'Automatic Opening of New Wells') or worked over. If this flag is set, the well is effectively shut until the end of the drilling operation by reducing its efficiency factor (see keyword `WEFAC`) in proportion to the fraction of the time step left after drilling. (If however the flag is not set, the drilling time only acts to limit the number of wells that can be drilled in a given time step.) The well will also be treated as temporarily closed during automatic workovers, for the duration set in keyword `WORKLIM`. The well can also be given a 'compartment number' in keyword `WDRILTIM`. When a well is being automatically drilled or worked over, all the wells sharing the same compartment number will be temporarily shut for the duration of the operation, by reducing their efficiency factors.

- Automatic opening of well connections.

An extra choice 'AUTO' has been added to the OPEN/SHUT flag for well connections in keywords COMPDAT and WELOPEN. Connections placed in this state are initially SHUT, but will be opened automatically whenever the well is subject to an automatic workover (see keywords WECON, GECON and GCONPROD). A new connection will be opened at each workover, in the order that they are first specified in keyword COMPDAT.

- Periodic testing of closed wells.

  Wells that have been closed due to economic limits or inability to operate at current reservoir conditions can be tested periodically to see if they can flow again. The testing instructions are input with keyword WTEST.

- Carter-Tracy aquifers.

  This is a new analytic aquifer option, providing an alternative to Fetkovich aquifers. The new keywords associated with Carter-Tracy aquifers are AQUCT, AQUANCON and AQUTAB.

- Additional options for Hysteresis.

  There is now a choice two models for relative permeability hysteresis in the non-wetting phase: Carlson's model and Killough's model. Previously only Carlson's model was available. There is also an option of using Killough's wetting phase hysteresis model in two-phase runs. The options are selected using the keyword EHYSTR.

- Gas Pseudo-Pressure option for well inflow.

  An inflow equation that is equivalent to the Gas Pseudo-Pressure equation can be selected to model the flow of dry gas between the well completions and the completed grid blocks. This is more accurate than the back-pressure equation (which is essentially the Russell-Goodrich equation) in cases where the drawdown is large enough to span a significantly non-linear segment of the graph of 1/(gas viscosity * FVF) vs. pressure. The option is selected in Item 8 of keyword WELSPECS, and is not available in gas condensate runs.

- Specification of well P. I. values.

  The steady-state productivity or injectivity index of a well can be set to a specific value. The well's connection transmissibility factors are adjusted proportionately to make the calculated P.I. value equal the specified value, at the current reservoir conditions. P. I. values are set with keyword WELPI.

Modifications to existing facilities

- Automatic lift curve switching.

  If both a VFP table number and an artificial lift quantity value are entered in keyword WLIFT for a well, the switching will now take place in two stages. The first time the rate falls below the set limit, the VFP table number will be changed (a re-tubing operation). When the rate falls below the limit a second time, artificial lift will be turned on. Constraints on the total group and field artificial lift capacity can also be set with keyword GLIFTLIM, to limit the number of wells automatically switched to artificial lift.

- Well cut-back limits.

  The well cut-back process (see keyword WCUTBACK) can now be reversed if the well's grid block pressure violates a specified limit (Item 9).

- Hysteresis.

By default, turnaround curves are not used in capillary pressure hysteresis - a cell being on a bounding drainage or imbibition curve or a scanning curve. This avoids the third reversal discontinuity which is inherent in the Killough model, and which can cause convergence problems. The old turnaround curve option may still be obtained by using the 6th argument of the OPTIONS keyword.

- Saturation table scaling.

  The relative permeability scaling option which preserves the relative permeability values at three saturation nodes has been extended to operate in both two and three phase problems. In addition, the same scaling procedure now maps the scaled critical saturation values on to the critical, rather than irreducible, saturation node in the relative permeability tables. Further details are contained in the "Saturation Table Scaling" chapter in the ECLIPSE Technical Description.

- Fetkovich aquifers can now be connected to more than one face of the reservoir, by using the new keywords AQUFETP and AQUANCON instead of AQUFET.

- Directional relative permeabilities.

  Defaulted directional saturation table numbers (entered using the directional forms of the keywords KRNUM and IMBNUM) are now set equal to the table number entered using the SATNUM keyword in each grid cell. If hysteresis is active, the default table number for the directional forms of the IMBNUM keyword is the value entered using the IMBNUM keyword in each cell.

- Table extrapolation warnings.

  The warning messages for extrapolation of PVT and VFP tables are now given only if requested, by using the new keyword EXTRAPMS. A separate warning message will be given if PVTO data is looked up at an R_s value greater than the largest value in the table, as this can produce a negative compressibility fluid. The message gives the pressure table number, the R_s value, and the highest R_s value in the table. This message is also controlled by EXTRAPMS.

- The treatment of multiple reservoirs (NUMRES>1 in RUNSPEC) has been changed slightly: cells with different reservoir numbers (keyword COORDSYS) will have zero mutual transmissibility values. Multiple reservoirs are thus independent except for possible connection through wells or group controls.

- DEBUG keyword switches that control calculation options (instead of requesting debug output) have been moved to the OPTIONS keyword.

New keywords

AQANTRC	Specifies concentration of water phase tracers in analytic aquifers.
AQUANCON	Specifies connection data for analytic aquifers.
AQUCT	Specifies property data for Carter-Tracy aquifers.
AQUFETP	Specifies property data for Fetkovich aquifers.
AQUTAB	Provides influence function table for Carter-Tracy aquifers.
EXTRAPMS	Requests that warnings should be given for table extrapolation.
GLIFTLIM	Sets limits for maximum group and field artificial lift capacity.
OPTIONS	Activates special options within particular program facilities.
WDRILTIM	Limits well drilling rate, and closes wells during drilling.
WELPI	Sets well productivity/injectivity index values.
WTEST	Provides instructions for periodic testing of closed wells.

New summary section keywords

- FAQR, AAQR, FAQT and AAQT for aquifer influx rates and totals.
- FOGR, GOGR and WOGR for producing oil-gas ratios.
- FFPO, FFPW, FFPG, RPPO, RPPW, RPPG for field- and region-averaged. potentials for oil, water and gas.
- BPPO, BPPW, BPPG for grid-block potentials of each phase.

Altered keywords

AQFETTRC	Keyword is now obsolete, it has been renamed AQANTRC.
COMPDAT	Additional option 'AUTO' in Item 6 for automatic opening of new connections during workovers.
DEBUG	Items previously used to activate special options within particular program facilities have been removed. The OPTIONS keyword should now be used to activate special options.
EHYSTR	New Items 2 and 3, governing the new relative permeability hysteresis options.
GECON	New Item 9 limiting the number of open wells within the group.
RUNSPEC	Record 26: this has been extended to include table dimensions for Carter-Tracy influence functions.
WCONINJ	Additional option 'AUTO' in Item 3 for automatic opening of well.
WCONINJE	Additional option 'AUTO' in Item 3 for automatic opening of well.
WCONPROD	Additional option 'AUTO' in Item 2 for automatic opening of well.
WCUTBACK	New limit (Item 9) included to reverse cut-back process.
WECON	New Item 9 nominating another well to be opened when the current well is closed.
WELOPEN	Additional option 'AUTO' in Item 2 for automatic opening of wells or connections.

`WELSPECS`	Choices in Item 8 extended to include Gas Pseudo-Pressure option.
`WLIFT`	Re-tubing and artificial lift switching are now performed as a two-stage process.

ECLIPSE 100: developments in 86/1

This section summarizes the changes to ECLIPSE between the 85/7 release and the 86/1 release.

New facilities

- Automatic radial gridding.

  If, in a radial run, the `INRAD` and `OUTRAD` keywords are used, ECLIPSE will set up `DR` values with a logarithmic distribution, using the correct number of cells in the radial direction.

- `RUNSUM` keyword in the summary section.

  This enables neat tabulated output of the data in the summary files to be written at the end of the normal print output file. This is the same output as is produced by GRAF under the run summaries option, but produced by ECLIPSE itself.

- Group injection control.

  Target and limiting injection rates can be set for groups and the field, with the keyword `GCONINJE`. These can be defined in terms of a re-injection or voidage replacement fraction, if required. If a target cannot be met because of insufficient injection potential, the drilling queue will be scanned for a suitable injector to open.

- Multi-level group control.

  Groups can be arranged in a multi-level tree structure, using the keyword `GRUPTREE`. Thus a group (e.g. a platform) can have other groups subordinate to it. Group production and injection controls can be applied at any level.

- Sales gas production control.

  The sales gas production is defined as the total gas production minus the gas consumption (see keyword `GCONSUMP`) and minus the gas injection. The sales gas production from any group or the field can be controlled by having the surplus gas re-injected. Sales gas targets and limits are set with the keyword `GCONSALE`.

- Well rate cut-back limit on block pressure.

  The set of limits for automatic well rate cut-back (see keyword `WCUTBACK`) has been extended to include a limiting pressure for any grid block with an open connection to the well. This is treated as a lower limit for producers and an upper limit for injectors. The well rate will be cut back if a well block violates this limit.

- Control of numerical diffusion in tracer tracking runs.

  The effects of numerical diffusion on the smearing of tracer interfaces is most noticeable whenever tracer moves within single phase regions of the grid (for example gas injection into a gas cap, water injection into an aquifer). Control of tracer numerical diffusion is activated for single phase tracers by the `QTDISP` keyword in the `RUNSPEC` section.

- `ALL` keyword in the summary section.

  This requests a full set of field, group and well data be written to the summary files. See `SUMMARY` section for details.

Modifications to existing facilities

- Group production control.

  The treatment of defaulted group guide rates has been modified. If the group guide rates are defaulted in keyword GCONPROD, the field target is apportioned directly between the wells in proportion to their guide rates or production potentials. To restore the former treatment (i.e. to apportion the field target between the groups in proportion to their potential sums), set the group guide rates to zero. Also, different actions can be selected for each of the four rate limits in keyword GCONPROD.

- Implicit collective control.

  The Jacobian derivatives for the interaction between wells under group control are no longer calculated. Accordingly the flag QIMCOL in the RUNSPEC section is set internally to FALSE. The well target rates are updated in the first NUPCOL Newton iterations of each time step, to meet the group production/injection targets. During any subsequent iterations the well target rates will remain constant, to allow the Newton iterations to converge faster. During these subsequent iterations the group rates may drift slightly from their targets. But the drift can be limited to within a specified tolerance fraction by using the keyword GCONTOL.

- Enhanced relative permeability scaling.

  In oil-water runs the existing end point scaling option transforms the water relative permeability curve between the critical water saturation and the maximum water saturation values. An option is now available to include the residual oil saturation in water, $(1 - S_{owcr})$, in the scaling calculation. The new facility is available for two phase combinations. See the "Saturation Table Scaling" chapter in the ECLIPSE Technical Description and keyword SCALELIM for further details.

- Initial gravity drainage/imbibition quiescence in dual porosity models.

  Quiescent initial conditions can be produced when the QGRAID flag in the RUNSPEC section is set TRUE in dual porosity/permeability runs. The engineer can impose initial quiescence in the matrix/fracture flow by activating switch 35 in the DEBUG keyword.

- Oil immobilization in Vertical Equilibrium.

  The vertical equilibrium model has previously assumed that, in a three phase case, the phases always occur in the order oil-water-gas. It is, however, possible for a gas-water contact to rise above a band of oil, if this oil is immobilized - i.e. at residual saturation. This process can occur, for example, where a thin oil layer is displaced upwards by invading water. A band of residual oil remains from the original base of the oil layer. Once the oil has moved sufficiently, this residual oil can account for all the oil in the grid block. A band of immobilized oil then exists, and a gas water contact can rise completely above this. Details of the four possible configurations of a three-phase VE cell are given in the "Vertical Equilibrium" chapter in the ECLIPSE Technical Description. ECLIPSE previously modelled only the first configuration.

New keywords

GCONINJE	Sets targets and limits for group injection control.
GCONSALE	Sets targets and limits for sales gas control facility.
GCONSUMP	Sets group gas consumption rates.
GCONTOL	Sets tolerances for group control targets.
GRUPTREE	Sets up tree structure for multi-level group control.
INIT	Requests initial file be written for use by GRAF and PSEUDO.
OUTRAD	Sets outer radius for logarithmic calculation of DR values.
RUNSUM	Prints tabular output of summary file data at end of print file.
SCALELIM	Sets saturation scaling limits in the enhanced relative permeability scaling option.
WCONINJE	Sets controls and limits for injection wells; a simpler alternative to WCONINJ.

New summary section keywords

- FGSR, GGSR, FGST, GGST

  for gas sales rates and totals.
- ALL

  generates basic selection of field, group and well data.

Altered keywords

DEBUG	New debug switches to apply the enhanced relative permeability scaling model in two phase runs, and to produce quiescent initial conditions in gravity drainage/imbibition calculations in dual porosity/permeability systems.
GCONPROD	Items 11 - 13 added to specify different actions on violating particular maximum rate limits.
MESSAGES	Default print limits for severity 1 and 2 messages changed from 1000 to 5000.
QDRILL	The drilling queue can now contain injection wells.
RUNSPEC	Record 16: QTDISP applies numerical diffusion control in the tracer tracking model. Record 20: QIMCOL = T option is no longer available, as the well interaction derivatives for group control are not calculated. NUPCOL has its default value changed from 4 to 3.
WGRUPCON	Guide rate definition options supplied for injection wells.
WCONINJ	A simpler alternative to this keyword, WCONINJE, is available to set control data for injection wells. This omits the re-injection and voidage replacement targets for individual wells, but re-injection and voidage replacement can be performed with more flexibility at the group/field level using keyword GCONINJE. The WCONINJ keyword is still available, however, but it should not be used for injectors under group control. A warning message is now printed if the BHP limit is defaulted.

`WCONPROD` Default BHP value changed from zero to atmospheric pressure. A warning message is now printed if the BHP limit is defaulted.

`WCUTBACK` Grid block pressure limit included as additional limit option.

ECLIPSE 100: developments in 85/7

This section summarizes the changes to ECLIPSE between the 84/11 release and the 85/7 release.

New facilities

- API tracking.

  This option can be used to model the mixing of different types of oil, having different API values and PVT properties. See the “API Tracking” chapter in the ECLIPSE Technical Description.

- Tracer Tracking.

  Fluid movement can be tracked with the aid of passive tracers (which do not affect the fluid properties). See the “Tracer Tracking” chapter in the ECLIPSE Technical Description.

- Saturation table endpoint scaling.

  The saturation table endpoints can be scaled independently for each grid block, or as a function of depth. See the “Saturation Table Scaling” chapter in the ECLIPSE Technical Description.

- A new check on total compressibility has been implemented. See the “Total Compressibility Checks” chapter in the ECLIPSE Technical Description.

- Initial fluid mobility correction.

  This option allows the correct initial mobile quantities of each phase to be determined. See the “Saturation Table Scaling” chapter in the ECLIPSE Technical Description.

- An IMPES solution method has been implemented, as an optional alternative to the fully implicit method. See keyword `IMPES` and the “Fully Implicit & IMPES Methods” chapter in the ECLIPSE Technical Description.

- `DPGRID`.

  This is a new grid section keyword which enables the grid data for a dual porosity run to be entered for the first `NDIVIZ`/2 layers (the matrix cells), the fracture values being copied from these. This makes the specification of faulted dual porosity runs easier, and avoids spurious cell overlap messages.

- Gravity imbibition in dual porosity runs.

  The exchange of fluid between matrix and fracture cells due to gravitational forces may now be modelled. See the “Dual Porosity” chapter in the ECLIPSE Technical Description.

- An alternative method can be selected for calculating grid block transmissibilities. See keyword `OLDTRANR` and the “Transmissibility Calculations” chapter in the ECLIPSE Technical Description.

- There is a choice of alternative methods for calculating 3-phase oil relative permeabilities. See keywords `STONE1` and `STONE2` and the “Saturation Functions” chapter in the ECLIPSE Technical Description.

- Economic limits for production well connections.

Individual well connections can be closed if their water cut, GOR or WGR exceeds its economic limit. See keyword CECON.

- Downtime for wells.

  Efficiency factors can be defined for wells, to take account of regular downtime. See keyword WEFAC.

- Automatic lift switching.

  Artificial lift can be switched on automatically when the well flow rate falls to a specified value, by changing either the VFP table number or the artificial lift quantity. See keyword WLIFT.

- Workover rate limitation.

  A limit can be imposed on the rate at which well workovers can be performed. See keyword WORKLIM.

- Optional output of well production/injection totals by layer, according to 9th switch in RPTSCHED keyword.
- The volumes of each phase in the regions defined by the initial oil, water and gas zones can be produced at simulation report times in VE runs. These fluids-in-place reports are produced using item 21 in the RPTSOL and RPTSCHED keywords.
- Optional output of unified restart files and unified summary files. See RUNSPEC section and the "File Handling in ECLIPSE" chapter in the ECLIPSE Technical Description.

Modifications to existing facilities

- The dual porosity option now multiplies the fracture permeabilities rather than the transmissibilities by the fracture porosities. This does not affect the fracture-to-fracture transmissibilities, but ensures that the PI for the wells and circle completion non-neighbor connections are calculated with the fracture porosity factor. As an alternative, the NODPPM keyword may be used to avoid the fracture porosity multiplier entirely, the user entering net bulk fracture permeabilities directly.
- An improved treatment has been provided for overlapping transition zones in the equilibration calculation. If the sum of the water and gas saturations exceeds 1.0, the saturations are recalculated from the gas-water capillary pressure.
- The default equilibration option (Item 9 in keyword EQUIL) has been changed to 0 in runs using the Vertical Equilibrium option.
- The tilted block equilibration option (Item 9 > 0 in keyword EQUIL) has been modified for runs using the NEWTRAN keyword. The tilt on the top and bottom faces of each cell is now determined from the corner point depths, instead of the depths of the neighboring cells' top and bottom faces. This prevents the spurious tilting of cells adjacent to displacement faults.
- In the flat or tilted block equilibration options, R_s is reset to the saturated RS value at the grid block pressure if the block contains any free gas, instead of the average R_s value of all the levels. This prevents pressure transients resulting from the immediate solution of free gas into under-saturated oil. The resetting of R_s can be prevented by giving the 8th switch in the DEBUG keyword a negative value.
- In the Material Balance Reports (RPTSCHED switch 8) for a restart run, the initial fluids in place and cumulative flow now refer to time 0.0 days, and not the latest restart time. The grid block region numbers must not be changed during a restart.

- If the well PVT table number (`WELSPECS` item 11) is defaulted, it will be set to the PVT region number of the well's lowest completed grid block, instead of region 1.
- The process of opening wells automatically from the drilling queue has been modified. If the field target rate can no longer be met, the first well in the queue not belonging to a group under group control will be opened. This avoids the problem of drilling queue wells being left un-opened if they are added to the queue after their groups have dropped out of field control.
- The initial pressure in Fetkovich aquifers can now be defaulted. This will cause the pressure to be calculated automatically, so as to place the aquifer as nearly as possible in equilibrium with the reservoir.
- The names of formatted output files (restart, summary etc.) have been changed, to distinguish them from unformatted ones.

New keywords

APIVD	Sets initial oil API vs. depth.
AQFETTRC	Sets tracer concentrations for water in Fetkovich aquifers.
CECON	Imposes economic limits on production well connections.
COLUMNS	Sets left and right margins for reading input data file.
DPGRID	Customizes the grid for dual porosity problems.
DZMTRX	Sets parameter for gravity imbibition treatment in dual porosity.
ENDNUM	Defines endpoint vs. depth table number regions for ENPTVD tables.
ENPTVD	Sets saturation table endpoints vs. depth, for endpoint scaling.
GRAVITY	Sets fluid gravities, as alternative to DENSITY keyword.
IMPES	Selects IMPES solution method as alternative to fully implicit.
IMPLICIT	Restores use of the fully implicit solution method.
MESSAGES	Resets print and stop limits for messages of any severity.
NODPPM	Avoids dual porosity permeability multiplier for fracture cells.
NOWARN	Suppresses ECLIPSE warning messages.
OILAPI	Sets initial oil API values in each grid block.
OLDTRANR	Selects alternative method of calculating block transmissibilities.
SGCR	
SGL	
SGU	
SOGCR	Family of keywords to set saturation endpoints in each grid block,
SOWCR	for endpoint scaling option.
SWCR	
SWL	
SWU	
STONE1 & STONE2	Selects alternative methods of calculating 3-phase oil relative permeabilities.
TBLK	Sets initial tracer concentrations in each grid block.
TNUM	Defines regions for tracer concentration vs. depth tables.
TRACER	Defines tracer names.
TVDP	Provides tables of initial tracer concentration vs. depth.
WEFAC	Sets well efficiency factors, to account for downtime.
WLIFT	Sets instructions for automatically switching on artificial lift.
WORKLIM	Limits the rate at which workovers can be performed.
WTRACER	Sets tracer concentrations for fluids injected through wells.

Altered keywords

AQUFET	The initial pressure (item 2) can be defaulted.
DEBUG	Debug switches for API, tracer and endpoint scaling calculations.
RESTART	Allows specification of report number on a unified file.
RPTPROPS	New switches for output of tracer and endpoint scaling data.
RPTREGS	New switches for output of tracer and endpoint table regions.
RPTSCHED	New output switches for API and tracer tracking.
RPTSOL	Output switches for API, tracer and endpoint scaling quantities.
RUNSPEC	Record 6: extended for API and tracer tracking options.
	Record 12: extended for endpoint saturation scaling option.
	Record 14: extended for mobile fluid in place and tracer tracking calculations.
	Record 16: includes a switch for the gravity imbibition option in dual porosity runs.
	Record 30: includes switches for output of unified restart and summary files and input from a unified restart file. NSTACK default changed from 4 to 10.
SUMMARY	Extra SUMMARY section keywords for API and tracer tracking output.
TUNING	Extra quantity to control max. timestep increase after a chop.

ECLIPSE 100: developments in 84/11

This section summarizes the changes to ECLIPSE between the 84/9 release and the 84/11 release.

New facilities

- Hysteresis of relative permeability and capillary pressure curves (see the “Hysteresis” chapter in the ECLIPSE Technical Description).
- Rock Compaction option - pressure dependent multipliers for pore volume and transmissibility (see keyword ROCKTAB).

Modifications to existing facilities

- DRSDT now has a default value of infinity (1.0E20) instead of zero. The values of DRSDT and DRVDT will have no effect in runs containing both dissolved gas and vaporized oil.
- The total compressibility check has been removed. A new compressibility check will be provided in a future version of ECLIPSE.

New keywords

COMPIMB	Sets imbibition table number for hysteresis in well connections.
EHYSTR	Sets curvature parameter for capillary pressure hysteresis.
IMBNUM	Sets imbibition table region numbers for hysteresis.
ROCKNUM	Sets rock compaction table region numbers.
ROCKTAB	Supplies tables of rock compaction data.

Altered keywords

DEBUG	Extra switch (16) for hysteresis calculation debug output.
DRSDT	Default value now infinity; has no effect on runs containing both dissolved gas and vaporized oil.
DRVDT	Has no effect on runs containing both dissolved gas and vaporized oil.
RPTREGS	New switches to govern output of imbibition and rock compaction table number regions.
RUNSPEC	Record 10: extended for rock compaction option, adding NTROCC, QROCKC and QRCREV. Record 12: extended for hysteresis option, adding QHYSTR.

ECLIPSE 100: developments in 84/9

This section summarizes the changes to ECLIPSE between the November 1983 version and the September 1984 version.

New facilities

- Vertical Equilibrium option (see the “Vertical Equilibrium” chapter in the ECLIPSE Technical Description).
- Extensions to well model: improved group/field production control, tolerance fraction for economic limits, automatic rate cut-back, calculation of well potential (see the “Well Modelling Facilities” chapter in the ECLIPSE Technical Description).
- Extensions to equilibration facility: quiescence and threshold pressures (see the “Equilibration” chapter in the ECLIPSE Technical Description).
- The Pair Method to accelerate non-linear equation convergence.
- New control on oil/gas mass transfer (see keyword `VAPPARS`).
- Memory management for solver stack (see `RUNSPEC` section).
- Output file for PSEUDO package (see keyword `PSEUDOS`).
- Option for formatted restart and summary files (see `RUNSPEC` section).
- Extra output options, including phase pressures and depth corrected pressures (see keywords `RPTSOL` and `RPTSCHED`).
- New alternatives to existing keywords (see keywords `DATES` and `DZNET`).

Modifications to existing facilities

- The average field and region pressures are now hydrocarbon volume weighted averages. The original pore volume weighted averages could be significantly affected by large aquifer zones. The change in weighting will also affect well voidage rate calculations, as the reservoir fluid volumes are calculated at the average field pressure.
- The `NEWTRAN` option may now be used for radial grids.

New keywords

COMPRP	Sets saturation scaling data for well connections in non-VE runs.
COMPVE	Resets top and bottom depths for well connections in VE runs.
DATES	Sets dates of reports; alternative to TSTEP.
DATUM	Sets datum depths for output of depth-corrected pressures.
DZNET	Sets grid block net thicknesses; alternative to NTG.
GCONPROD	Sets flow targets and limits for group and field control.
GECON	Sets economic limits for groups and field.
PSEUDOS	Requests output of data for the PSEUDO package.
THPRES	Sets threshold pressures for flow between equilibration regions.
VAPPARS	Controls vaporization of oil in under-saturated gas.
VEFRAC	Sets the weighting factor between VE functions and rock curves.
WCUTBACK	Sets automatic rate cut-back limits for production wells.
WELOPEN	Opens and closes wells and well connections.
WGRUPCON	Sets well guide rates for group control.
WLIMTOL	Sets tolerance fraction for well, group and field economic limits.

Altered keywords

DEBUG	Extra switches for new facilities.
EQUIL	Item 9: N=0 gives special VE equilibration in VE runs.
NEWTRAN	May now be used for radial grids.
RPTSCHED	Extra output switches for phase pressures and VE quantities.
RPTSOL	Extra output switches for phase pressures and VE quantities.
RUNSPEC	Record 12: extra switch QVEOPT for Vertical Equilibrium (VE) option.
	Record 14: extra switches QUIESC, QTHPRS, QREVTH for quiescence and threshold pressure options.
	Record 16: extra switch QPAIR for activating Pair Method.
	Record 30: NSTACK=0 stores solver stack on scratch disk file; extra switches QFMTOU, QFMTIN for formatted restart file.
SOF3	Defaults can be inserted in columns 2 and 3 for interpolation.
WCONPROD	Item 3: extra control mode ‘GRUP’ for wells under group control; abbreviation ‘G’ no-longer uniquely refers to ‘GRAT’.
WELSPECS	Item 5: this now defaults to depth of top-most connection in well.
	Item 6: re-named ‘preferred phase’; relevant for group control.
	Item 7: if set negative, well potential will be printed in reports.

Obsolete keywords

`GROUPCON` Group control facility substantially revised; use `GCONPROD` for group/field control data and `GECON` for group/field economic limits.

`WCONGRUP` Use `WCONPROD` to set operating limits for group-controlled wells, and `WGRUPCON` to set their guide rates (unless they are defaulted).

ECLIPSE 200: developments in 99A

This section summarizes the new developments to the ECLIPSE 200 options between the 98A release and the 99A release. See the respective chapters of this manual for more detailed explanations.

Flux boundary conditions

- There is an option to use pressure and saturation boundary conditions, instead of defined flux boundary conditions, from the full field run.

Gas field operations model

- Gas import rates and cumulative totals can now be reported to the Summary file.

Gas lift optimization

- When group production rules are used in combination with gas lift optimization, the user can now dictate which production rule actions should take precedence over gas lift optimization, and which should only be performed subsequent to a gas lift optimization calculation.

Local grid refinement

- It is now possible to specify tuning data for individual local grids.
- The restriction that the global grid and all local grids should have the same dimensionality when using the tracer option has been removed (e.g. 3D local grids can be used in a 2D global model).

Multi-segment wells

- The frictional pressure drop multiplier can be defined to vary with the water-oil ratio and gas-oil ratio of the mixture flowing through the segment.
- Specific segments can be defined to represent a ‘labyrinth’ flow control device, using a built-in correlation to model the pressure loss.
- A down-hole separator facility can separate water or free gas from the mixture flowing through a specified segment, and send it along a lateral branch to reinject into the formation.
- The Miscible Flood Model may now be used with Multi-segment Wells.
- A more robust iteration sequence for solving multi-segment wells can be invoked.
- Information on segment tracer flows, oil API and pressure loss multiplier can be written to the Summary File.
- Information on the phase flow velocities and holdup fractions is now written to the RFT file.

The network option

- The extended network model now has a facility for defining efficiency factors at the nodes.

Parallel options

- The Parallel Options now work with the Network Option and Multi-segment Wells.
- The Parallel LGR option has been removed.
- The Parallel executable `local.exe` has been removed and the parallel (domain decomposition) option now runs multiple copies of the ECLIPSE executable.

Reservoir coupling

- The `LICENSES` keyword can be entered in the master's `RUNSPEC` section to instruct the master immediately to check out FLEX*lm* licenses for specified ECLIPSE 200 options. Licenses taken out by the master can be shared with all the slaves running on the same host.

The surfactant model

- It is now possible to use the Surfactant Model with the end point scaling option. If the end point scaling option is active it is also possible to scale the critical oil to water saturation when fully miscible conditions are achieved (post surfactant flood).

ECLIPSE 200: developments in 98A

Coal Bed Methane Model development history

New developments for 97A release

This section summarizes the changes to the Coal Bed Methane Model between the 96A release and the 97A release.

New facilities

- It is now possible to specify a separate diffusion coefficient for the solvent component. See the `DIFFCOAL` keyword.

Altered keywords

`DIFFCOAL`	New item 3 allows input of a different diffusion coefficient for the solvent component.

New developments for 95A release

This section summarizes the changes to the Coalbed Methane model between the 94A release and the 95A release.

New facilities

- A new relative adsorption model has been added that simulates the de-adsorption process based on the gas and solvent partial pressures. See the `COALPP` keyword.
- The methane production rates and cumulative totals can be output to the Summary file. See the Summary section.

New keywords

`COALPP` Partial pressure relative adsorption model.

New `SUMMARY` section keywords:

`FMPR GMPR WMPR`

Methane Production Rate

`FMPT GMPT WMPT`

Methane Production Total

`FMIR GMIR WMIR`

Methane Injection Rate

`FMIT GMIT WMIT`

Methane Injection Total.

Altered keywords

`RUNSPEC` With the new keyword format, use keywords `COAL`, `DUALPORO`, `SOLVENT`.

Environmental Tracers development history

New developments for 96A release

This section summarizes the changes to the Environmental Tracer option between the 95A release and the 96A release.

New facilities

- It is now possible to specify the injected tracer concentration in a well stream to be a linear function of the cumulative fluid injected by the well. This can by useful when modelling H_2S generating processes.

Altered keywords

`WTRACER` New item 4 to set cumulative tracer factor.

New developments for 95a release

This section summarizes the changes to the Environmental Tracer option between the 94A release and the 95A release.

New facilities

- A tracer diffusion model has been added. This facility can be used to model molecular diffusion of a contaminant within, for example, a ground water flow. See the `TRDIF` keyword.
- If the adsorption model is employed, there are now two options for describing the initial concentration. The tracer can either be assumed to partition between the fluid and the rock, or extra adsorbed tracer can be added to the rock to be in equilibrium with the fluid concentration. See the `TRROCK` keyword.

- It is now possible to use the standard tracer solver method, as an alternative to the "Cascade" numerical diffusion control algorithm. See item 5 in the RUNSPEC section keyword TRACERS.
- Unit names can be ascribed to tracers, for reporting purposes. See the TRACER keyword.
- The rock mass can now be output to the Print file, using the mnemonic 'ROCKMASS' (15th control) in the RPTSOL keyword.

New keywords

TRDIF	Specifies the tracer diffusion coefficient.

Altered keywords

RUNSPEC	With the new keyword format, use keyword TRACERS.
TRACER	New item 3 to specify units for reporting purposes.
TRROCK	New item 3 to specify the initialization model for adsorption.
RPTSOL	'ROCKMASS' mnemonic (15th control) to output the rock mass.

Flux Boundary Conditions development history

New developments for 98A release

This section summarizes the changes to the Flux Boundary option between the 97A release and the 98A release.

New facilities

- An OPTIONS keyword switch has been added to suppress certain data checks for local grids adjacent to flux boundaries and also for wells connecting into multiple flux regions in the base full field run. This is acceptable as long as the wells do not actually straddle the reduced flux region in the reduced run, and the local grids are not actually adjacent to the reduced flux region in the reduced run. See the 65th switch in the OPTIONS keyword.

Altered keywords

OPTIONS	The 65th switch suppresses certain error conditions for wells and local grids in a full field run.

New developments for 96A release

This section summarizes the changes to the Flux Option between the 95A release and the 96A release.

Changes to the behavior of existing facilities

- The performance has been improved in USEFLUX runs where only a small proportion of a large number of flux regions are active. When the flux file is read in, only the active regions are stored on the temporary scratch file, hence reducing disk space requirements (for the scratch files) and the time subsequently spent doing I/O.

New developments for 94a release

This section summarizes the changes to the Flux Option between the 93A release and the 94A release.

New facilities

- In two phase runs using either the `RSCONST(T)` or `RVCONST(T)` facility, the FLUX file now includes the flux of the extra phase (gas or oil respectively). This enables the `USEFLUX` run to be set up as a three phase model. If this was attempted in previous versions of the program, the flux of the extra phase was taken to be zero.

Changes to the behavior of existing facilities

- An initial restart file will be generated from a restart run if it is requested by either the RPTSOL or RPTRST keywords. This is useful in the case where a base run does not dump flux data.

Altered keywords

`RPTSOL`	Now outputs an initial restart file from a restart run.
`RPTRST`	

New developments for 92A release

This section summarizes the changes to the Flux Option between the 91A release and the 92A release.

New facilities

- Flux regions, defined by the `FLUXNUM` keyword, can now be used to identify regions for subsequent modification of the grid data. `GRID` Section data for specified flux regions can be modified selectively using the keywords `EQUALREG`, `ADDREG` and `MULTIREG`. In addition, the `FLUXNUM` array can be copied into the `REGIONS` Section arrays such as `FIPNUM` and `SATNUM`, if you want the grid block `FIPNUM` numbers for example to equal their `FLUXNUM` numbers. This facility for data specification is available whether or not you are using ECLIPSE 200 Flux Boundary Conditions.
- Flux region numbers can now be written to the Print file, if requested using the 36th control in the `RPTGRID` keyword.

New keywords

ADDREG	Adds a constant to a named GRID array for a given flux region.
EQUALREG	Sets a named GRID Section array for a given flux region.
MULTIREG	Multiplies a named GRID array by a constant for a flux region.

Altered keywords

COPY	Can now copy from FLUXNUM in the REGIONS Section.
RPTGRID	36th control outputs flux region numbers.

The Foam Model

The Foam Model was a new option in 99A.

Gas Calorific Value Control

Gas Calorific Value Control was a new option in 99A.

Gas Field Operations Model development history

New developments for 98A release

This section summarizes the changes to the Gas Field Operations Model between the 97A release and the 98A release.

New facilities

- New SUMMARY section keywords (F,G)GC(R,T) report the gas consumption rate, and its cumulative total, at and below the field or the specified group.

New developments for 97A release

This section summarizes the changes to the Gas Field Operations Model between the 96A release and the 97A release.

New facilities

- The GRUPTREE keyword can now be used in the Annual Scheduling File, to change the group hierarchy during the contract year (see keyword GASBEGIN).

Altered keywords

GASBEGIN	The GRUPTREE keyword is allowed in the Annual Scheduling File.

New developments for 96a release

This section summarizes the changes to the Gas Field Operations Model between the 95A release and the 96A release.

New facilities

- The Gas Field Operations Model may now be used in combination with Local Grid Refinement.
- An accelerated iteration scheme for calculating the DCQ in the first pass of the contract year is now available. It allows the DCQ to increase again if a previous iteration had decreased its value too much. In the original scheme the DCQ could only decrease at each iteration, so in order to achieve an accurate result the amount by which the DCQ could reduce per iteration had to be strictly limited, requiring several iterations to converge if there was a substantial reduction in DCQ from the previous contract year. The accelerated scheme should converge to the required accuracy in fewer iterations of the first pass. It is activated with item 2 of the `GASFIELD` keyword in the `RUNSPEC` section. See also item 8 of keyword `GASYEAR`. The number of iterations used to converge the DCQ can be written to the Summary File using the `SUMMARY` section keyword `NGOPAS`.
- A new keyword, `GDCQECON` in the `SCHEDULE` section, allows the user to define a minimum economic DCQ value for a contract group. All the producers in this group are shut when the DCQ falls below its minimum economic value. If the group is the FIELD, the run will be terminated.
- Automatic compressors can be turned off by using keyword `COMPOFF`. They will be turned on again when needed. ECLIPSE automatically turns off the compressors when there is a reduction in the Swing or Profile factor. But `COMPOFF` provides a way of turning them off manually when the rate target is set with `GCONPROD` or `GCONPRI` instead of using `GASYEAR`. Note that, when using the new Extended Network Model, compressors must be defined with keyword `NETCOMPA` instead of `GASFCOMP`.

New keywords

`COMPOFF`	This keyword temporarily turns off all automatic compressors.
`GDCQECON`	Sets a minimum economic DCQ for a contract group.
`NETCOMPA`	This keyword must be used to define automatic compressors when the Extended Network Model is employed.

SUMMARY section

`NGOPAS`	Outputs the number of iterations used to converge the DCQ in the first pass of the contract year.

Altered keywords

`GASYEAR`	New item 8 to define the convergence tolerance for the DCQ iterations when using the accelerated scheme.
`GASFCOMP`	This keyword must **not** be used to define automatic compressors when using the new Extended Network Model. Keyword `NETCOMPA` should be used instead.

RUNSPEC section

`GASFIELD`	New item 2 to select the accelerated iteration scheme for the DCQ calculation.

New developments for 94a release

This section summarizes the changes to the Gas Field Operations Model between the 93A release and the 94A release.

New facilities

- If satellite groups (see the ECLIPSE 100 reference manual) are included in the grouping hierarchy, their flows may be updated at any month within the contract year by including the keywords GSATPROD and GSATINJE in the Annual Scheduling File.

Altered keywords

GASBEGIN	The new keywords GSATPROD and GSATINJE are allowed in the Annual Scheduling File.

New developments for 93a release

This section summarizes the changes to the Gas Field Operations Model between the 92A release and the 93A release.

New facilities

- The multiple contract groups option allows selected groups to have their own separate gas supply contracts, instead of just a single contract on the FIELD's production. Each contract group has its own set of swing and profile factors, and ECLIPSE can adjust their DCQs independently to meet their swing requirements. See keywords GSWINGF & GDCQ for more details.
- The new ECLIPSE 100 keyword GRUPTARG can be included in the Annual Scheduling File (see keyword GASBEGIN).
- The group nodal pressures in the network, when operating at delivery capacity, can be written to the Summary File with the mnemonic GPRDC.

New keywords

GDCQ	Sets initial DCQ values for multiple contract groups
GSWINGF	Defines multiple contract groups and sets their swing and profile factors

SUMMARY section

New summary section mnemonics:

GGDCQ	Group Gas DCQ (for multiple contract groups)
GPRDC	Group Pressure at Delivery Capacity

Altered keywords

GASBEGIN	The new keyword GRUPTARG is allowed in the Annual Scheduling File

New developments for 92a release

This section summarizes the changes to the Gas Field Operations Model between the 91A release and the 92A release.

New facilities

- The extended compressor option allows each compressor to have up to five levels of compression. The compressors will be turned on or switched to a higher compression level individually in a predefined order until the field can make its production target. The order in which these events occur is determined by an "action sequence number". See keyword `GASFCOMP` for more details.
- Two more items can be written to the Summary File:
  - Well gas delivery capacity (`WGDC`), i.e. the well's gas production rate when the field is operating at its delivery capacity
  - Group compressor level (`GMCPL`), i.e. the current operating level of the automatic compressor associated with the group.

Changes to treatment of existing facilities

- If a well is opened from the drilling queue (see keyword `QDRILL`) during the first pass of a contract year in order to meet the swing requirement, it will automatically be opened in the same month of the second pass, whether or not it is needed to meet the DCQ × profile target. Before 92A, the well would only be opened in the second pass if it was needed to meet the DCQ × profile target. If you wish to restore the pre-92A treatment, set Item 34 > 0 in the `OPTIONS` keyword.
- The well gas flows at delivery capacity are now printed instead of the well potential rates in the right hand column of the Production Report (if requested by setting the drainage radius negative in Item 7 of keyword `WELSPECS`).
- It is no longer necessary to put the field under guide rate control with keyword `GCONPROD` before using the `GASYEAR` keyword - ECLIPSE will now do this automatically. You should still use `GCONPROD` if there are limits on the field production rate of any other phase apart from gas, or if any groups have rate limits or guide rates. If you wish to use prioritization instead of guide rate control for controlling the field's gas production, you must use the keywords `PRIORITY` and `GCONPRI` as before.

New keywords

SUMMARY section

New summary section mnemonics:

`GMCPL`	Group Multi-level Compressor Level
`WGDC`	Well Gas Delivery Capacity

Altered keywords

`GASFCOMP`	New items 5 and 6 for use with the extended compressor option.
`OPTIONS`	New item 34.
`RUNSPEC`	The redundant logical flag `QIMCOL` in Record 20 has been replaced by `QEXGOP`, which must be set T if the extended compressor option is required.

New developments for 91a release

This section summarizes the changes to the Gas Field Operations Model between the 90A release and the 91A release.

New facilities

- To provide flexibility to handle different forms of contract, there is a choice of interpretations of the swing requirement. The chosen interpretation is selected by entering its associated mnemonic as Item 3 in keyword GASYEAR.
- There is an option to repeat the first pass calculation of a contract year if it requires too large a reduction in DCQ to obey the swing requirement. See Item 5 in the GASYEAR keyword.
- The DCQ may be reduced by a specified fraction at the start of each new contract year, to anticipate part of the overall reduction needed to obey the swing requirement. See Item 6 in the GASYEAR keyword.
- When keyword SWINGFAC is entered, a check is made to see if the profile factors are normalized to give the required Annual Contracted Quantity. A warning will be printed if they are not normalized.
- It is now possible to change well controls and request reports within each contract year. The user can request ECLIPSE to place a set of keywords in the "Annual Scheduling File", by bracketing the set with the keywords GASBEGIN and GASEND. ECLIPSE will cycle through the file every contract year, performing the requested operations at the start of each month specified by the GASMONTH keyword.
- The user can specify, with keyword GASFDELC, whether the well rate limits and any group oil or water rate limits should be retained or removed when calculating the delivery capacity.

Changes to treatment of existing facilities

- The check that the swing factor is not less than the profile factor in any month has been downgraded from an error to a warning.
- Zero values may be entered for some swing and profile factors.

New keywords

GASBEGIN	Denotes start of set of keywords to go into the Annual Scheduling File.
GASEND	Denotes end of set of keywords to go into the Annual Scheduling File.
GASFDELC	Specifies whether the well rate limits, and any group oil and water rate limits, should be retained or removed when calculating delivery capacity.
GASMONTH	Sets month for next set of operations in Annual Scheduling File.

Altered keywords

GASYEAR	Extended choice of options for swing requirement application in Item 3. New items 5, 6 and 7 to control the automatic reduction of DCQ.

Gas Lift Optimization development history

New developments for 98A release

This section summarizes the changes to the Gas Lift Optimization facility between the 97A release and the 98A release.

New facilities

- An upper limit can be imposed on the total gas rate of a group (i.e. produced gas + lift gas). See the GLIFTOPT keyword.

Altered keywords

GLIFTOPT A new Item 3 allows the user to set an upper limit on the total gas rate of a group (i.e. produced gas + lift gas).

New developments for 94a release

This section summarizes the changes to the Gas Lift Optimization facility between the 93A release and the 94A release.

New facilities

- Cubic spline interpolation of ALQ values in the VFP tables.

  There is an option to use a cubic spline procedure to interpolate the ALQ values (i.e. lift gas injection rate) in the VFP tables. The spline procedure has the advantage of making the gradient of well performance with respect to lift gas injection rate vary continuously over the range of the table. The (default) linear interpolation can produce sudden changes of gradient at the tabulated ALQ values, which may bias the optimized allocation of lift gas to these values. The spline interpolation procedure is enabled with the keyword VFPTABL.

- Optimization frequency with a network.

  Gas lift can be optimized in a network in each of the first NUPCOL Newton iterations of the time step, instead of just the first, if the network is also balanced with this frequency (by setting a negative balancing interval in keyword NETBALAN). See Item 4 in keyword LIFTOPT.

New keywords

VFPTABL Enables spline interpolation of ALQ values in the VFP tables.

Altered keywords

LIFTOPT 'YES' in Item 4 is now allowed when there is a network present, provided a negative balancing interval has been set in keyword NETBALAN to request network balancing in each of the first NUPCOL Newton iterations.

New developments for 93a release

This section summarizes the changes to the Gas Lift Optimization facility between the 92A release and the 93A release.

New facilities

- Minimum lift gas injection rates for wells.

A minimum lift gas injection rate can be specified for each well (see keyword `WLIFTOPT`). Provided that there is sufficient lift gas available, each well's lift gas allocation will not be allowed to fall below its specified minimum value, unless the well is unable to flow at that lift gas injection rate. Alternatively, a negative value can be specified for the minimum lift gas injection rate. ECLIPSE will then allocate at least enough lift gas to these wells to keep them flowing, provided that there is enough lift gas available. If there is not enough lift gas for all the wells' minimum requirements, ECLIPSE allocates lift gas to each well in decreasing order of their weighting factors.

If the wells belong to a group (including the FIELD) that can achieve its production rate target without requiring any gas lift, by default no lift gas will be allocated to the wells even if they have a minimum lift gas rate set. However, any wells that are given a weighting factor greater than 1.0 will be allocated their minimum lift gas rate, even if the group/field production rate target could be met without any gas lift.

- Summary file output of incremental gradients.

  The incremental gradients of selected wells (i.e. the increase in oil production rate achieved by allocating an additional increment of gas lift to the well, divided by the increment size) can be written to the Summary File by using the mnemonic `WOGLR` - Well Oil Gas Lift Ratio.

New keywords

SUMMARY section

New Summary Section mnemonic:

`WOGLR`	Well Oil Gas Lift Ratio (i.e. the incremental gradient).

Altered keywords

`WLIFTOPT`	New item 5 sets the wells' minimum lift gas injection rate.

GI Pseudo-Compositional Model development history

New developments for 93A release

This section summarizes the changes to the Gi Model between the 92A release and the 93A release.

New facilities

- It is now possible to run a Gi Model restart from a non-Gi base case.
- Grid block Gi values can be written to the Summary file, using the Summary section mnemonic `BGI`.

New keywords

SUMMARY section

New summary section mnemonic:

BGI	Block Gi value.

Local Grid Refinement & Coarsening development history

New developments for 98A release

This section summarizes the changes to the Local Grid Refinement and Coarsening Option between the 97A release and the 98A release.

New facilities

- The LGR-in-place solution is now compatible with the improved dual permeability solver, which will in general lead to reduced run times for dual permeability models.
- An alternative treatment of transmissibility multipliers is available when using autorefinement. Keywords SMULTX, SMULTY and SMULTZ are applied on the autorefined grid only at boundaries of the base grid.
- The “default” tracer tracking scheme is now available in local grid runs.
- It is now possible to simulate 2-phase tracers in local grid runs (i.e. gas tracers in live oil systems or oil tracers in wet gas systems).
- The Temperature Option is now available in local grid runs.
- It is now possible to avoid the creation of .LGR summary files using keyword NOSUMLGR. This can be useful to save file space. When running LGR-in-place, there is little point dumping summary files for the local timesteps since the global and local timesteps are all synchronized.
- There is now a local grid version of the WPIMULT keyword, as the keyword can now set PI multipliers for individual well connections.
- It is now possible to use MULTX, MULTY, MULTZ and MULTPV within the SCHEDULE section in local grid runs. This may be useful for studies of fracturing etc.
- The enumeration keywords PBUB and PDEW may now be specified between the REFINE/ENDFIN pair in the SOLUTION section.
- There is an option to read only the global data from a restart file containing local grids. See the GETGLOB keyword.
- It is now possible to change the data specified in keyword AMALGAM when doing a restart. Any local grids added a restart may be amalgamated with existing local grids in the base run.

Changes to the behavior of existing facilities

(These may cause changes to the results of a simulation, or alter the form or content of the output.)

- When using the coarsening option in dual porosity runs, the DZMTRXV, LX, LY and LZ arrays will now be upscaled to the coarse cell dimensions when using the gravity drainage (GRAVDR) or viscous displacement (VISCD) options.

New keywords

GETGLOB	Instructs ECLIPSE to read only the global data from a restart file containing local grids.
SMULTX	Alternative X-direction transmissibility multiplier in autorefinements.
SMULTY	Alternative Y-direction transmissibility multiplier in autorefinements.
SMULTZ	Alternative Z-direction transmissibility multiplier in autorefinements.
WPIMULTL	Local grid version of WPIMULT to apply PI multipliers to individual well connections.

SUMMARY section

NOSUMLGR	Avoid the creation of .LGR summary files for the local timesteps.

Altered keywords

WELOPENL	New items 7 and 8 allow the opening/closing of connections belonging to a range of completion numbers.
WELSPECL	New 'GPP' option in Item 9 invokes the Generalized Pseudo-pressure method.
COMPDATL	New item 14 allows the user to specify the pressure equivalent radius used in the calculation of Peaceman's formula.
COMPDATM	New item 15 allows the user to specify the pressure equivalent radius used in the calculation of Peaceman's formula.

RUNSPEC section

AUTOREF	New item 4 to control the behavior of transmissibility multiplier keywords SMULTX and MULTX in autorefined grids.

New developments for 97A release

This section summarizes the changes to the Local Grid Refinement and Coarsening Option between the 96A release and the 97A release.

New facilities

- A fully implicit solution method (LGR in-place) has been implemented for local grids. This is an alternative to the default procedure for local grids in which the local grids are advanced separately using local timestepping. In-place local grids are solved simultaneously with the global grid, and use the same time steps as the global grid. The fully implicit coupling overcomes the stability problems that may arise with traditional local grids due to high flows across the boundary, and it also enables group control targets to be met with greater accuracy. The keyword LGRLOCK can be used to convert a traditional local grid to a fully implicit in-place local grid. The keyword LGRFREE may be used to revert an in-place local grid back to local timestepping.
- A keyword HRFIN has been introduced to make it easier to set up the geometry of radial local grids.

- The principal directions for the linear solver may be specified independently for each local grid, if required, by using the SOLVDIRS keyword (see the ECLIPSE 100 Reference Manual).
- The action of LGROFF may now be controlled by the 56th switch of the OPTIONS keyword. By default, the local grids are not switched off until the global timestep is completed, but just before advancing the local timestep. Thus, pseudo-local calculations are still performed for the global timestep and the wells will be solved using the local grid solutions. If the 56th switch is set > 0, the local grids will be switched off before advancing the global timestep. LGROFF will always be processed before solving the global timestep for any local grids run in-place (see keyword LGRLOCK).

Performance

- The cascade algorithm used for (single phase) tracers in runs containing local grid refinement has been speeded up considerably.

Changes to the behavior of existing facilities

(These may cause changes to the results of a simulation, or alter the form or content of the output.)

- When a depth variation of the saturation table end points is requested using either ENPTVD or ENKRVD or their hysteretic equivalents, the depth variation is honoured for local depths. Previously, the end points were inherited from host cells in the global grid which honoured the depth variation only in the global grid.
- In dual porosity runs, the flows between matrix and fracture cells for refined cells in the global grid are now represented by the sum of matrix-fracture flows for the corresponding refined cells in order to improve convergence.
- When the parameters MXWSIT and MXWPIT controlling the number of iterations for solving local grid wells are specified in TUNINGL, these are honoured for any wells in local grids when solving the global timestep as well as when solving the local timestep. Previously, the values specified in TUNING were used.
- When the top (or bottom) perforation depth in COMPVEL lies outside the grid block, ECLIPSE adjusts it to back to the top (or bottom) corner depth of the block. Now, however, if this occurs for a single connection entered in a record, ECLIPSE will also adjust the depth of the bottom (or top) of the perforations by a similar amount, provided that the perforated length remains entirely within the local grid block. Thus the length of the perforations should be preserved if the depth has to be adjusted due to a mis-match with the grid. This change mirrors that made to the COMPVE keyword (see the ECLIPSE Reference Manual).

New keywords

LGRLOCK	Converts an LGR from local timestepping to an in-place LGR.
LGRFREE	Returns an in-place LGR to local timestepping.
HRFIN	Splitting ratios for radii in radial LGR.

Altered keywords

OPTIONS	New switch 56 modifies the action of the LGROFF keyword.

New Developments for 96A Release

This section summarizes the changes to the Local Grid Refinement and Coarsening Option between the 95A release and the 96A release.

Note The Parallel LGR Option has been incorporated in the Parallel Options ECLIPSE 200 feature.

New facilities

- Local Grid Refinement may now be used in combination with the following facilities:
  - The Gas Field Operations Model
  - Drilling and workover rigs

    (keyword GRUPRIG)
  - Workover frequency constraints

    (keyword WORKLIM)
  - Group artificial lift capacity constraints

    (keyword GLIFTLIM)
  - Well partial penetration model

    (keywords COMPVE, COMPVEL)
  - Saturation scaling for well connections

    (keywords COMPRP, COMPRPL).
- A radial local grid refinement may now be specified in a box of four columns using the keyword RADFIN4.
- An autorefinement facility has been introduced to refine the whole grid as a reverse process to coarsening. This is intended to scale datasets easily for massively parallel computations. See the section on Autorefinement.
- The REFINE keyword can be used in the SCHEDULE section to modify REGIONS data during the simulation.
- For amalgamated local grids, the non-neighbor connections arising between the local grids in an amalgamation, their corresponding transmissibilities, and the corresponding flows for each phase are now written to the GRID, INIT and RESTART files respectively.
- Saturation table end-points within local grids are now written out to the INIT file for post-processing when the end-point scaling option is active.

- The global-local transmissibilities between local grids and the global grid may now be written to the INIT file.
- The maximum material balance correction in the global grid, arising from the difference between the local and global solutions, can now be output to the print file by setting the mnemonic 'SUMMARY' greater than 3 in keyword RPTSCHED.
- The local grid connection quantity keywords in the SUMMARY section, starting with 'LC', may have their connection I, J, K coordinates defaulted to give output for all the local grid connections in the well.
- A local grid name may now be specified in the VEDEBUG keyword.
- When advancing a local grid to the end of the global timestep, the initial solution for the Newton iteration to solve the local grid may be obtained by a temporal interpolation of the local grid pseudo solution, which is used while solving the global grid. This is enabled by the OPTIONS keyword switch 51. It has an effect only if the local grid takes several mini-steps to cover a global timestep. Then in general it will improve the convergence of the linear equations when solving the local grids by providing a better initial guess for the Newton iteration. (This facility was available in later versions of the 95a release.)

Changes to the behavior of existing facilities

(These may cause changes to the results of a simulation, or alter the form or content of the output.)

- Error checks have been added to prevent aquifers connecting with refined cells, which is not allowed and may cause convergence problems.
- When the OLDTRAN keyword is used with a corner point grid, the transmissibilities between the global and local grids are now calculated consistently with the block-centred representation of the grid, instead of using the NEWTRAN method. Also, when NEWTRAN is used with a block-centred grid, the global-local transmissibilities are now calculated using a corner-point representation of global and local grids. This ensures the global and local flows are more consistent and so improves accuracy, with improved convergence in many cases.
- The restriction that transmissibilities between amalgamated local grids were only calculated using a corner point representation of the grid with NEWTRAN has been removed. When the OLDTRAN method is employed, the transmissibilities between local grids are now calculated consistently with the block-centred representation of the grid.
- The treatment of non-neighbor connections within or connecting to local grids has been improved. These may have caused convergence difficulties for the global grid in earlier versions of the code when the LGR was placed in a highly faulted region of the global grid having global non-neighbor connections.
- When the Vertical Equilibrium option is active, ECLIPSE now checks for gaps between adjacent active global and local cells on the top and bottom faces of local grids. If these do not match (due to pinched-out inactive cells on the interface), ECLIPSE will reset corner point depths so that the depths do match. In previous versions, the VE algorithm broke down and caused convergence problems in such cases. See OPTIONS keyword switch 52. (This facility was available in later versions of the 95A release.)
- In runs where the Vertical Equilibrium option is active, an improved treatment of non-neighbor connections has been implemented, whether these occur either within local grids or between the global and local grids. The restriction that local grids should not be placed next to faults when using the VE model has been removed.

- When the pore volumes of any global cells which have been refined are altered in the EDIT section, the ratio of the edited pore volume to the calculated pore volume is applied as a multiplier on MULTPV for the corresponding refined cells. Previously, no account was taken and the global edited pore volume was reset to the sum of the local unedited pore volumes.
- The EDITNNC keyword may not be used to edit any non-neighbor connections to or from refined cells in the global grid. ECLIPSE will ignore any lines of data in EDITNNC relating to such connections. This is to avoid convergence difficulties, because the local grid non-neighbor connections are not altered by using EDITNNC. It is recommended that the multiplier keywords MULTX etc. (and MULTX- etc.) in the GRID section (not in the EDIT section) are used instead, rather than modifying individual connections with EDITNNC.
- When the global timestep is chopped or designated to be re-solved, the convergence of the solution in the global grid host cells has been improved in many cases when there is live oil or dissolved gas present.
- The ordering of connections has been changed in horizontal wells that are not friction wells (i.e. do not use WFRICTN or WFRICTNL in the Wellbore Friction Option). Previously they were ordered according to depth irrespective of the well's orientation. Now, if any connection is declared to be horizontal in item 13 of COMPDATL or item 14 of COMPDATM, ECLIPSE will attempt to order the connections in "heel-to-toe" fashion. See COMPDATL item 13 or COMPDATM item 14 for further details.
- When a local grid is switched off using the LGROFF keyword, its grid solution data written to the Restart file is obtained by interpolation from the global cells containing the LGR. This is simply a cosmetic change and will not alter the results in any way (for either the base or restart runs), but simply provides a more reasonable solution for graphical post-processors such as GRAF when plotting the solution data for local grids once they have been switched off. Previously, local grid solution data written to the restart files was not updated while the LGR was switched off.

New keywords

RADFIN4	Specifies a radial refinement in a box of four columns.
AUTOCOAR	Specifies coarsening within an autorefinement.
COMPVEL	Connection partial penetration data for wells in local grid refinements and local grid amalgamations.
COMPRPL	Connection saturation re-scaling data for wells in local grid refinements and local grid amalgamations.

RUNSPEC Section

AUTOREF	Activates the autorefinement facility.

Altered keywords

OPTIONS	New switch 51 to improve linear convergence behavior.
	New switch 52 to improve convergence for VE cases.
RPTSCHED	Material balance errors for each phase may be written out using mnemonic 'SUMMARY'.
REFINE	Now allowed within the SCHEDULE section.

New developments for 95A release

This section summarizes the changes to the LGR model between the 94A release and the 95A release.

New facilities

- Local Grid Refinement may now be used in combination with the following facilities:
  - API tracking
  - Brine tracking
  - Tracer tracking (except 2 phase tracers)
  - Surfactant model
  - Foam model.

  The local grid refinement option cannot be used with 2 phase tracers; these are tracers defined as either a gas tracer where gas can dissolve in oil or an oil tracer where oil can vaporize into the gas.
- It is now possible to model amalgamations of Cartesian LGRs which are allowed to touch. Wells may now be completed in several local grids, as long as they belong to the same amalgamation. The facility allows the user to refine the grid more efficiently around deviated wells, and also to employ nested refinements. See the keyword AMALGAM.

  Wells in amalgamated local grids must have their connections defined with keyword COMPDATM instead of COMPDATL. Also for these wells it is necessary to use the new keywords COMPLMPL, WELOPENL and WFRICTNL in place of COMPLUMP, WELOPEN and WFRICTN. (COMPLMPL lumps connections into completions; WELOPENL shuts or re-opens well connections; and WFRICTNL permits the modelling of friction wells completed across several local grids if the ECLIPSE 200 Wellbore Friction Option is active.)

- Extra LGRs may now be added at a RESTART. The new LGRs should be specified after the existing LGRs in the GRID section. Adding an LGR is equivalent to specifying an LGR at the start of the base run then turning it on at a later date using the LGRON keyword.

  This facility allows the user to add new LGRs into the grid where new wells are about to be opened. It is not possible to add a new LGR around a well that has already been declared with the keyword WELSPECS.
- The keywords REFINE and ENDFIN may now be used within the EDIT section to edit local grid data arrays. The pair can also be used in the GRID section to supply additional data for LGRs that have already been defined.
- It is possible to supply an incomplete set of values for local grid arrays in the GRID section. Any values for local grid cells that have not been set by the end of the GRID section will be interpolated from the global host cells.
- Wildcards may be used to refer to several local grids in a single record in keywords LGRON and LGROFF.
- REGIONS data for local grids is now output to the INIT file.

Changes to the behavior of existing facilities

(These may cause changes to the results of a simulation, or alter the form or content of the output.)

- For the first local time step following a completed global time step, the initial estimate of the solution has been changed from the previous local solution to the last pseudo solution used to solve the global system. In many cases this will be a better estimate of the final converged solution and hence will reduce the number of Newton iterations required to converge the local time step. The modification will cause small differences in the results, but these should be within the convergence tolerance.
- The formation of pinchout connections arising from combining both the PINCH and MINPV or MINPVV keywords has been corrected for refinements and coarsenings. Pinchout NNCs are now formed according to the same criteria as for the global grid when they either lie within an LGR, or are adjacent to or straddle the boundary of an LGR. Also, the transmissibility between coarse cells has been modified to account for pinchouts internal to the coarsening. This may affect flow rates when pinchouts are required with refinements or coarsenings. Previously, pinchouts were not handled correctly for LGRs or for grid coarsening. (See the keywords PINCH, MINPV and MINPVV in the ECLIPSE 100 Manual.)
- For an LGR straddling a fault, the horizontal NNC transmissibilities between local and global cells on either side of the fault are now calculated.
- When the face of an LGR lies on the boundary between two reservoirs (i.e. coordinate line systems - see keyword NUMRES in the RUNSPEC section), the appropriate local-to-global connections are now formed if communication between the reservoirs is specified in the COORDSYS keyword.
- The REGIONS data for local grid refinements is now output to the Print file for all local grid cells, rather than only those cells for which the appropriate REGIONS array has been set using REFINE/ENDFIN.
- Output of real grid arrays in the global grid now uses the letters R and C instead of the decimal point to distinguish the global cells that correspond to local grid refinements and local grid coarsenings.

- If both the connection transmissibility factor and the Kh value are supplied in items 8 and 10 of keyword COMPDATL, the denominator term involving the pressure equivalent radius will be calculated to be consistent with these values instead of being calculated from the grid block geometry. This term will affect calculations involving the well P.I. or a D-factor. Pre-95A functionality can be restored by setting OPTIONS keyword switch 45 > 0. (See item 8 of the COMPDATL keyword for more details. The same modification applies to keyword COMPDAT.)
- If the global timestep is constrained by the material correction following the local timesteps, the name of the constraining LGR is now printed after the LGRC mnemonic on the timestep summary output.

New keywords

AMALGAM	Specifies local grid amalgamations.
COMPDATM	Specifies connections for wells in amalgamated local grids.
COMPLMPL	Lumps connections in local grids for automatic workovers.
WELOPENL	Shuts or re-opens wells or well connections within local grids.
WFRICTNL	Designates a well in a local grid (or completed in several amalgamated local grids) as a friction well.

Altered keywords

REFINE/ENDFIN	
	The keyword pair may now be used in the GRID and EDIT sections.
COMPDATL	If the connection transmissibility factor is supplied, either Kh or the denominator term will be calculated from it.
LGRON/LGROFF	
	Wildcards may now be used.
OPTIONS	New switch 46 to reproduce the behavior of versions prior to 9413 when MINPV or MINPVV is specified in the global grid but not in local grids.

New developments for 94a release

This section summarizes the changes to the LGR model between the 93A release and the 94A release.

New facilities

- It is now possible to set the initial conditions by enumeration.

  The initial solution values (of PRESSURE, SWAT etc.) will default to the values in the host cell. The solution within each local grid can be specified using the REFINE keyword in the SOLUTION section. (See the REFINE keyword.)
- The maximum number of wells in a Cartesian refinement is now unlimited. This had previously been set to 40.
- The memory requirements for local grid refinement have been reduced.

Changes to the behavior of existing facilities

(These may cause changes to the results of a simulation, or alter the form or content of the output.)

- The default transformation between surface rate and reservoir fluid volume rate for wells in local grids has been modified. Previously the transformation was performed at the average hydrocarbon pressure in the **local** grid. The transformation is now performed at the average hydrocarbon pressure in the **global** grid, which is consistent with the way the transformation is performed in non-LGR runs. Thus the reported reservoir volume rates will be slightly different; and so will the behavior of runs using reservoir volume rate control or voidage replacement. If you wish to have the transformation performed at conditions more local to each well, you could set up a fluid-in-place region around each well and use Item 14 in keyword `WELSPECL` to request that the reservoir conditions for the transformation be taken from these regions instead of the whole field.
- Well D-factors may be entered with keyword `WDFAC`, instead of Item 12 of keyword `COMPDATL`. If `COMPDATL` Item 12 is defaulted, the connection D-factors will retain their previous values, instead of being set to zero. These changes follow those made to the `COMPDAT` keyword (see the ECLIPSE 100 reference manual), and are made to clarify the difference between a well D-factor and a connection D-factor. Connections may be shut using `COMPDATL` without having to re-specify the well's D-factor.
- The calculation of NNC transmissibilities between coarsened cells has been improved when a grid coarsening straddles a fault.

Altered keywords

`COMPDATL`	Defaulting item 12 now causes the connection's D-factor to retain its previous value, rather than setting it to zero.
`REFINE`	The refine keyword is now available in the `SOLUTION` section.

New developments for 93a release

This section summarizes the changes to the LGR model between the 92A release and the 93A release.

New facilities

- The ECLIPSE 200 Polymer and Solvent models may now be used in runs containing local grid refinements.
- The non-neighbor connections that ECLIPSE generates to couple the LGRs to their adjacent global grid cells can be written to the Debug file. The data is written in the form of an `NNC` keyword. This can be useful when converting an ECLIPSE 200 LGR into a traditional 'in-place' refinement. (See the RPTGRIDL keyword.)
- It is now possible to control the mobile end-point correction facility for individually selected LGRs. (See the `QMOBIL` keyword.)

Changes to the behavior of existing facilities

(These may cause changes to the results of a simulation, or alter the form or content of the output.)

- If inter-block flows are requested to be output to the Restart file (with item 2 of the `RPTRST` keyword), these will also include the flows between the LGR cells and their adjacent global grid cells. Also, if an extended Grid file is requested, this will now include the non-neighbor connections coupling the LGRs to the global grid cells.

- If an LGR is placed next to a pinched-out cell, ECLIPSE will now generate the non-neighbor connections to couple the LGR to the global grid cell on the other side of the pinch-out. Previously LGR-to-global cell connections were not handled correctly across a pinchout.

New keywords

QMOBIL	Allows control of the mobile end-point correction within an LGR.

Altered keywords

CARFIN	Up to 40 wells per local grid can be requested in item 11.
GRIDFILE	Outputs global-to-local connection data to the extended Grid file.
RADFIN	Up to 40 wells per local grid can be requested in item 9.
RPTRST	Outputs global-to-local grid block flows if item 2 is set >1.
RPTGRIDL	24th control enables output of global-to-local grid connection data to be written to the DEBUG file.

New developments for 92a release

This section summarizes the changes to the LGR model between the 91A release and the 92A release.

New facilities

- The REFINE keyword can be used in the PROPS and REGIONS sections to initiate data input for individual local grids. This allows, for example, the user to enter grid-type end point scaling data for the local grid system, enabling near-wellbore variations in saturation end point to be modelled more accurately.

Changes to treatment of existing facilities

- The VEFRACVL keyword has been replaced by the REFINE and VEFRACV keywords.
- A full Grid file containing inactive cell, local grid and coarsening data is now written by default in runs containing local grid refinements or coarsenings. This removes the requirement to use the GRIDFILE keyword to display local grid and coarsening data in the GRID and GRAF programs.
- If the OPTIONS keyword switch 37 is set > 0, ECLIPSE will replace the solution state of each local grid cell with that of its global host cell. This may speed the solution in cases where there are state changes happening in local grid cells near the well which are not taking place in the global cell, for example with condensate dropout.

New keywords

REFINE	Initiates data input for a named local grid.

Altered keywords

ENDFIN	Is now also used to end a REFINE block.
GRIDFILE	Now defaults to the full Grid file in LGR/LGC runs.
OPTIONS	New option 37.
VEFIN	Now uses 'YES'/'NO' instead of T/F for the V.E. flag in item 1.
VEFRACVL	Replaced by the VEFRACV keyword within a REFINE block.

New developments for 91a release

This section summarizes the changes to the LGR model between the 90A release and the 91A release.

New facilities

- The local grid systems can be run with Vertical Equilibrium (VE) switched on in selected local grids. The option is only available when VE is switched on in the global grid, but VE or dispersed flow can be specified separately for each local grid (see the VEFIN keyword). The VEFRACV data can be specified for each grid block within each local grid system, using the VEFRACVL keyword.
- There is an option to use a more accurate representation of the geometry of the global grid blocks that contain refined grid systems, when Vertical Equilibrium is switched on in the global grid. This is achieved by increasing the number of "distinct depths" used to represent the global cells. The number of distinct depths can be limited to prevent the initial setup being too expensive. The option is activated with the 2nd item of the VEFIN keyword.
- Keyword WELPI can now be used on wells in local grids, to scale their connection transmissibility factors to give each well a specified steady-state productivity/injectivity index at the current reservoir conditions. In earlier releases, it was not possible to use WELPI on wells in local grids.
- The length of the next timestep in the local grid systems can be restricted to a specified maximum size, using the NEXTSTPL keyword. This is a simpler alternative to using the TUNINGL keyword to control this value.
- It is now possible to get information about the CPU usage in the sections of the program calculating the local grid solutions.

Changes to treatment of existing facilities

- There is a new method for calculating inter-block flows when the Vertical Equilibrium option is used in a global grid entered with block centred geometry. The block centred grid is converted to a smooth corner point system, and this corner point grid is used to calculate the VE relative permeabilities. This method is applied in local grid refinement runs to prevent problems with the flows from the global grid to the local grid. (In non-LGR runs this method is optional, and is activated by setting the 19th OPTIONS switch to 2.)
- An improved method of storage allocation allows a substantial reduction in the total memory requirement of LGR simulations to be achieved. In cases with many LGR regions of greatly varying mesh sizes, a saving of up to 50% in the LGR storage requirements has been achieved.

New keywords

VEFIN	1. Switches on VE in individual local grids. 2. Specifies the maximum number of distinct depths in the host cell.
VEFRACVL	Specifies local grid VEFRAC values on a block by block basis.
NEXTSTPL	Restricts the size of the next timestep in the local grid systems.

Multi-Segment Wells development history

Multi-segment wells was a new option in 98A.

The Network Option development history

New developments for 98A release

This section summarizes the changes to the Network Option between the 97A release and the 98A release.

New facilities

- In both the standard and extended network options it is now possible to equivalence the ALQ to the surface density of either the oil or the gas flowing in the branch, to look up a VFP table that has been prepared to cover a range of surface densities. This facility can be useful, for example, when the network combines oil of different surface densities produced from different parts of the reservoir. See the GRUPNET or BRANPROP keyword as appropriate.
- The extended network now has a facility for removing water at network nodes. This will affect the pressure losses in the downstream branches, but will not affect reported group flows or the operation of group targets or limits. See the NWATREM keyword.

New keywords

NWATREM	Removes water from a node in the Extended Network.

SUMMARY section

GALQ	Outputs the group's (or node's) outflow pipe's ALQ value. No units are given, as there are several possible definitions for this quantity.

Altered keywords

BRANPROP	New item 5 to specify whether the ALQ in the branch should be equivalenced to the surface density of the oil or gas.
GRUPNET	New Item 7 to specify whether the ALQ in the branch should be equivalenced to the surface density of the oil or gas.

New developments for 97A release

This section summarizes the changes to the Network Option between the 96A release and the 97A release.

New facilities

- In the Extended Network Model, it is now possible to control a group's production rate by automatically adjusting the pressure loss across a choke in a designated network branch. The choke increases the pressure in its inlet node to a value that reduces the production of the subordinate wells to meet the group's rate target. This provides a third method of group control (in addition to guide rate control and prioritization), which previously was only available to well-groups via the 'manifold group' facility. The choke method is available only when using the Extended Network Model. See Items 3 & 5 of the NODEPROP keyword.
- Actions initiated by the PRORDER and GDRILPOT keywords (see the ECLIPSE 100 manual) can now be postponed until the network is balanced, by using the keyword WAITBAL. This will prevent, for example, a well being opened from the drilling queue prematurely because an intermediate network balancing iteration has high nodal pressures and consequentially low well productivities.
- The choice as to whether to calculate the Extended Network's branch flows and pressure losses from the 'time-average' well flow rates (well production rates multiplied by their efficiency factors) or the 'peak' well flow rates (no account taken of the wells' efficiency factors) can now be made independently for each well with Item 3 of the WEFAC keyword. This option is available only when using the Extended Network Model.
- It is now permitted, in both the Standard and the Extended network models, to have fixed pressure nodes/groups subordinate to other fixed pressure nodes/groups. Previously this was possible only with the Standard network model.
- The pressure drop along a branch of the network can be written to the Summary File using the SUMMARY section keywords GPRB, GPRBW, GPRBG respectively for the production network, water injection network and gas injection network.

Behavioral changes

These may cause changes to the results of a simulation, or alter the form or content of its output.

- The 33rd switch of the OPTIONS keyword, governing the number of network iterations in which a well might be revived, has increased its default value. See the OPTIONS keyword for more details.
- The 54th OPTIONS switch, governing whether the wells' full flow rates or their time-averaged values should be used to calculate the pressure losses in the Extended Network, has been replaced by Item 3 in keyword WEFAC. The former OPTIONS switch should not be used.

New keywords

WAITBAL	Prevents PRORDER and GDRILPOT actions being performed until the network is balanced.

SUMMARY section

GPRB	Pressure drop along the group's (or node's) outlet branch of the production network.
GPRBG	Pressure drop along the group's inlet branch of the gas injection network.
GPRBW	Pressure drop along the group's inlet branch of the water injection network.

Altered keywords

NODEPROP	Item 3 designates nodes as choke inlets, if they do not correspond to well-groups.
	New item 5 to specify the group to which the choke should respond.
OPTIONS	The 33rd switch has increased its default value.
	The 54th switch has now been replaced by Item 3 of the WEFAC keyword.
	New 57th switch invokes an alternative procedure for handling automatic chokes when a low flow target has been set.
WEFAC	New Item 3 replaces the 54th switch of the OPTIONS keyword.

New developments for 96A release

This section summarizes the changes to the Network Option between the 95A release and the 96A release.

New facilities

- The "Extended Network Model" is available for production networks. It currently has two major enhancements over the "standard" model. Firstly, the surface network may be structured independently of the group control hierarchy (although still conforming to a gathering tree). Secondly, the automatic compressor/pump facilities are more versatile. The Extended Network Model is enabled by including the NETWORK keyword in the RUNSPEC section, and the network is constructed using BRANPROP and NODEPROP instead of GRUPNET. Automatic compressors/pumps are defined with keyword NETCOMPA. See also keywords COMPOFF and NCONSUMP. Currently the Extended Network Model cannot be used with Gas Lift Optimization.
- Individual wells can be isolated from the effects of the network by setting item 2 of keyword WNETDP to a value greater than or equal to 1.0E10. This causes the well's THP limit to remain unchanged by the network balancing calculation, independent of its group's nodal pressure in the network.

- Fixed pressure groups (or fixed pressure nodes, in the Extended Network Model), can have their pressure values automatically adjusted with the aim of keeping their production rates within specified upper and lower limits. (See keyword GNETDP.)
- When using the Extended Network Model, the 54th switch of the OPTIONS keyword can be used to control whether or not well efficiency factors are taken into account in the network branch flows. Setting this to 1 causes the network pressure losses to be calculated at the maximum flow rate rather than the average flow rate.

Behavioral changes

These may cause changes to the results of a simulation, or alter the form or content of the output.

- The messages that the network has converged in 'n' iterations are no longer printed, unless requested by using the mnemonic VFPPROD or NEWTON in keyword RPTSCHED (control 10 or 15).

New keywords

BRANPROP	Defines the Extended Network structure and assigns properties to the branches. This keyword must be used instead of GRUPNET when the Extended Network Model is employed.
COMPOFF	Switches off automatic compressors in the Extended Network Model and the Gasfield Operations Model.
GNETDP	Specifies which fixed pressures are to be changed dynamically and under what circumstances.
NCONSUMP	Specifies gas consumption rate at nodes in the Extended Network.
NETCOMPA	Defines automatic compressors or pumps in the Extended Network Model. This keyword must be used instead of GNETPUMP when the Extended Network Model is employed.
NODEPROP	Assigns properties to nodes in the Extended Network. This keyword must be used after BRANPROP when the Extended Network Model is employed.

RUNSPEC section

NETWORK	Turns on the Extended Network Model facilities, and specifies the maximum number of nodes and branches in the network.

Altered keywords

OPTIONS	new 54th control governs whether or not well efficiency factors are taken into account in the network branch flows.
WNETDP	Setting item 2 to a value greater than or equal to 1.0E10 will cause the well's THP limit to be independent of its group's nodal pressure in the network.

New developments for 94A release

This section summarizes the changes to the Network Option between the 93A release and the 94A release.

New facilities

- Network balancing at each Newton iteration

By setting a negative network balancing interval in Item 1 of keyword NETBALAN, ECLIPSE can be made to balance the network in each of the first NUPCOL Newton iterations of every time step. This should result in a much smaller balancing error at the end of the time step, although the time step may require more Newton iterations to converge.

- Fixing well flow rate instead of THP from the balanced network

  There is an option to have a flow rate limit instead of the THP limit held constant until the next balancing calculation, for wells constrained by the network. In general, this should only be considered if problems arise due to holding the well THP constant. See keyword WNETCTRL.

New keywords

WNETCTRL	Selects well THP or rate limits to be set by the network.

Altered keywords

NETBALAN	A negative balancing interval in Item 1 causes the network to be balanced in each of the first NUPCOL Newton iterations of every time step.

New developments for 93A release

This section summarizes the changes to the Network Option between the 92A release and the 93A release.

New facilities

- Developments to the Injection Network facility.
  - A gas injection network facility is now available. Gas injection networks are constructed with the keyword GNETINJE.
  - An injection VFP table number of 9999 can be entered in item 4 of keyword GNETINJE to indicate that there is no pressure drop along the injection network branch between the group and its father.
  - The group nodal pressures of a gas or water injection network can be written to the Summary File using the SUMMARY Section mnemonics GPRG and GPRW.
- Changing the network during the simulation.

  The GRUPTREE keyword can be used at any time in the simulation to re- allocate groups to different ‘fathers’, and to include new groups in the hierarchy. It is therefore possible, for example, to add a new network node part-way along an existing branch. Firstly, use GRUPTREE to enter any changes to the ‘father-son’ relationships of the grouping hierarchy. Secondly, use GRUPNET (or GNETINJE) to declare the new node as part of the network, and to change the properties of the original branch (e.g. its VFP table number).

New keywords

SUMMARY section

New SUMMARY section mnemonics:

GPRG	Group nodal PRessure in the Gas injection network
GPRW	Group nodal PRessure in the Water injection network

Altered keywords

`GNETINJE` A gas injection network can be selected in item 2.

A VFP table number of 9999 in item 4 indicates a zero pressure drop along the group's network branch to its father.

New developments for 92A release

This section summarizes the changes to the Network Option between the 91A release and the 92A release.

New facilities

- Branches with zero pressure drop.

  A VFP table number of 9999 can be entered in item 3 of keyword `GRUPNET` to indicate that there is no pressure drop along the production network branch between the group and its father. This saves the user from having to construct a special VFP table with BHP equal to THP over a range of flowing conditions.

Changes to treatment of existing facilities

- Faster convergence of network balancing iterations.

  A modification has been made to the network balancing calculation to speed convergence. When production is not constrained by group or field rate limits, the network balancing calculation will generally require fewer iterations to converge. The converged nodal pressures will differ slightly from their values obtained with earlier versions of ECLIPSE, but only within the tolerance specified in item 2 of keyword `NETBALAN`. Thus the well flow rates will not be exactly the same as in earlier versions.

- Dying wells

  The procedure governing the treatment of wells unable to produce against their current THP limits in the network has been modified. It is now as follows:

  During the network balancing calculation (which is performed in the first Newton iteration of the time step), if a well is unable to produce at a THP equal to its group's nodal pressure at a particular network balancing iteration, it will be revived to see if it can flow at the next network balancing iteration. But the revival process is stopped after the 4th network balancing iteration, to allow subsequent iterations to converge. Thus if it cannot flow during or after the 4th network balancing iteration, the well dies. The number of network balancing iterations in which the well can be revived can be modified by the user with switch number 33 in the `OPTIONS` keyword.

  If the well dies during the network balancing calculation, it will not be revived during subsequent Newton iterations of the time step. The network will have been balanced with the well not flowing. The well will remain dead, to prevent it flowing at a THP that does not take account of its flow rate in the network.

  If a well survives the network balancing calculation in the first Newton iteration but is unable to flow during a subsequent Newton iteration of the time step, it will be revived in the next Newton iteration to see if it can flow at the new reservoir conditions. But the revival process is stopped after the 3rd Newton iteration, to allow subsequent iterations to converge. The number of Newton iterations in which the well can be revived can be modified by the user with switch number 15 in the `OPTIONS` keyword.

The changes from the 91A release are the ability to control revival during the network balancing calculation with OPTIONS switch 33, and the possibility of revival during subsequent Newton iterations of a well that survived the network balancing calculation but died in a later Newton iteration. (The 91A treatment of preventing any wells from being revived after the first Newton iteration could give pessimistic results.)

Altered keywords

GRUPNET	A VFP table number of 9999 in item 3 indicates a zero pressure drop along the group's network branch to its father.
OPTIONS	New switch 33 governs well revival during network balancing.

New developments for 91A release

This section summarizes the changes to the Network Option between the 90A release and the 91A release.

New facilities

- Network balancing error reporting and control

  The network is balanced using the grid block conditions existing at the beginning of the time step. By the end of the time step, the grid block conditions and well flow rates may have changed, and the flows in the network branches may no longer be exactly consistent with the nodal pressures. The longer the time step or interval between balancing calculations, the greater will be the balancing error at the end of the time step. ECLIPSE checks the balancing error in each branch of the network at the end of each time step, and prints out the largest error in any branch. The balancing error is defined as the difference between the pressure drop along the branch when the network was last balanced, and the pressure drop calculated using the current flow rates at the end of the time step.

  The engineer can supply a target value and a maximum permitted value for largest branch balancing error, in the NETBALAN keyword. ECLIPSE will attempt to adjust the time step length so as to keep the balancing error at approximately its target value, subject of course to any other constraints on the time step. Time steps limited by the target balancing error can be identified by the mnemonic 'NETW' printed in the time step information line. If the maximum permitted balancing error is exceeded, ECLIPSE will chop the time step and rebalance the network. Since time step chops can severely increase the running time of a simulation, a maximum permitted balancing error should only be used with caution, and its value should be several times larger than the target error. An over-riding minimum step size for balancing error control can also be set, to prevent time step reductions being too severe.

- Well lift gas flow along network branches

  If wells in a production network are subject to gas lift, the flow of lift gas can be included in the network branch pressure drop calculations by setting item 6 in keyword GRUPNET to 'FLO' or 'ALQ' for the pipelines that carry lift gas along with produced fluids. If 'FLO' is selected, the lift gas flows of the subordinate producers will be added to the gas flow rate along the pipeline. The lift gas flow is assumed to be equal to the sum of the ALQ values of the subordinate producers, multiplied by their efficiency factors. If 'ALQ' is selected, the gas flow rate will not be adjusted, but the ALQ value used to look up the pipeline's pressure drop will be set equal to the sum of the ALQ values of the subordinate producers, multiplied by their efficiency factors. If either option is selected, the Artificial Lift Quantity for the wells (WCONPROD item 12) and the pipeline (for the 'ALQ' option) must be defined as the rate of lift gas injection. This definition must be selected in the VFP program if it is used to construct the VFP tables.

Changes to treatment of existing facilities

- Dying wells

  When a well dies because it can no longer operate at current reservoir conditions, it is normally given three chances. For example, if it cannot operate in Newton iteration 1 it will be revived again for iteration 2 to see if it could still operate. After 3 iterations revival ceases and the well is declared dead.

  This approach can lead to an inconsistency when the Network Option is used. If the well dies during the network balancing calculation in iteration 0 but is revived again and can operate in a subsequent iteration of the time step, it will be operating at a THP that does not take account of its flow rate in the network.

  The program has been changed to prevent wells being revived after iteration 0 if the Network option is in use. Thus if a well dies during the network balancing calculation, it will remain dead. This may cause wells to die slightly earlier than in runs using previous versions of the code. If required, the pre-91A treatment of dying wells can be restored by setting the 15th control in the `OPTIONS` keyword to 3.

Altered keywords

`GRUPNET`	New item 6 to set instruction for including lift gas flow along network branches.
`NETBALAN`	New items 6, 7 and 8 to specify time step controls to limit the network balancing error at the end of each time step.

New developments for 90A release

This section summarizes the changes to the Network Option between the 88/09 release and the 90A release.

New facilities

- Water injection networks. A separate network, independent of the production network, can be defined for water injection, with keyword `GNETINJE`.
- Fixed pressure drops can be applied between a well's tubing head and its group's node in the network, with keyword `WNETDP`.
- The nodal pressure of any group in the production network can be written to the summary file, using the mnemonic `GPR` - Group PRessure.

New keywords

`GNETINJE`	Defines network structure for the water injection network.
`WNETDP`	Applies a fixed pressure drop between a well's tubing head and its group's node in the network.

New developments for 88/09 release

This section summarizes the changes to the Network Option between the 87/01 release and the 88/09 release.

New facilities

- Special treatment of production rate control for groups representing sub-sea completion manifolds.

Well-groups can be declared in keyword GRUPNET to be sub-sea completion manifolds. When a group of this type comes under rate control, the THP value of all its wells is adjusted so that the group meets its rate target, provided that it does not fall below the nodal pressure of the group in the network. This allows a group of wells to produce at a target rate while all the wells have the same THP. Groups not declared as sub-sea completion manifolds will in general have their wells operating at different THP values when under group control meeting a group rate target.

Altered keywords

GRUPNET	New Item 5, declaring the group to be a sub-sea completion manifold.
NETBALAN	New Items 4 and 5 setting the convergence tolerance and iteration limit for the calculation of THP for sub-sea completion manifold groups under rate control.

Parallel Options development history

New developments for 98A release

This section summarizes the changes to the Parallel Options between the 97A release and the 98A release.

New facilities

- The improved dual permeability solver is now used for parallel dual permeability runs, which will improve performance in many cases.
- The directory in which the slaves run (and write their output files) can now be specified with the PHOST keyword.
- The user can now specify the name of the local (or slave) executable with the MODULE keyword in the RUNSPEC section. This is useful for installations on specialist parallel machines where an additional executable linked with the machine vendor's optimized version of PVM may have been provided.

New keywords

RUNSPEC section

MODULE	Allows the user to specify the name of the local executable.

Altered keywords

PHOST	New item 3 to specify the directory in which the slave processes should run.
PARAOPTS	Some options controlled by this keyword have been deprecated and some additional options included.

Behavioral changes

These may cause changes to the results of a simulation, or alter the form or content of the output.

- Due to restructuring of the parallel code, output from the parallel version of ECLIPSE is now buffered. As a result, screen output now appears in short bursts rather than at a steady rate.

New developments for 97A release

This section summarizes the changes to Parallel Options between the 96A release and the 97A release.

New facilities

- The parallel domain decomposition code now runs with local grid refinement, except not presently with the ‘in-place’ local grid refinement option.
- The parallel local grid refinement option now allows multiple local grids to be stacked up on the same process.
- Each domain of a parallel domain decomposition can be allocated to a specific processor.
- The `PARAOPTS` options have been extended to provide additional options for optimizing message passing.

New keywords

`IHOST`	Allows individual LGRs to be allocated to a specific local process.
`PHOST`	Allows a local process to be placed on a specific host machine or processor.

Altered keywords

`PARAOPTS`	The keyword data has been divided into two records. There are extra items to allow more user defined tuning of the parallel option.

Altered RUNSPEC section keyword

`PARALLEL`	The '`SHARED`' option in item 2 has been changed to '`SERIAL`', as the shared memory parallel option is no longer supported. Additional 3rd data item to specify the maximum number of processes when multiple LGRs can be stacked up on the same process.

The Polymer Flood Model development history

New developments for 93A release

This section summarizes the changes to ECLIPSE Polymer Model between the 92A release and the 93A release.

New facilities

- The ECLIPSE 200 Local Grid Refinement option can now be used in conjunction with the Polymer Flood Model.
- It is now possible to use the Polymer Flood Model in conjunction with the ECLIPSE 200 Surfactant model. The two models are largely independent; however both components are assumed to affect the water viscosity. The water viscosity is modified by two independent multipliers, and hence the polymer and surfactant are assumed not to interact.

New developments for 92A release

This section summarizes the changes to ECLIPSE Polymer Model between the 91A release and the 92A release.

New facilities

- The effect of shear thinning on the polymer solution can now be modelled. The shear rate is assumed to be proportional to water phase velocity and to be reversible. (See the `PLYSHEAR` keyword and the model description.)

New keywords

`PLYSHEAR` Inputs shear thinning data.

Altered keywords

`RPTGRID` 38th switch outputs geometric data associated with the shear thinning model.

Reservoir Coupling development history

New developments for 98A release

This section summarizes the changes to the Wellbore Friction Option between the 97A release and the 98A release.

Behavioral changes

(These may cause changes to the results of a simulation, or alter the form or content of its output.)

- When new slaves have been started from a restart file, their reported flow rate in the master during the first time step used to be spuriously large in previous releases. (This was because the slaves' flow rates reported in the master are obtained from the change in their cumulative flows over the time step; for a new slave that was a restart the cumulative would, seemingly, change from zero to the value at its restart time in the very first time step.) This behavior has now been eliminated, with the correct change in cumulatives now being used to calculate the reported rates for the new slaves in their first time step.

The Solvent Model development history

New developments for 98A release

This section summarizes the changes to the Solvent model between the 97A release and the 98A release.

New facilities

- The grid block solvent relative permeability can be output to the Print file using the mnemonic `KRN` in the `RPTSOL` and `RPTSCHED` keywords. The inter-block flow rates of solvent can be output using the mnemonic `FLOSOL` in `RPTSOL` and `RPTSCHED`.
- The `SUMMARY` section keyword `BNKR` outputs the grid block solvent relative permeability.

New keywords

SUMMARY section

BNKR	Grid block solvent relative permeability.

Altered keywords

RPTSCHED	New KRN and FLOSOL mnemonics output the grid block solvent relative permeabilities and inter-block flow rates.
RPTSOL	New KRN and FLOSOL mnemonics output the grid block solvent relative permeabilities and inter-block flow rates.

New developments for 96A release

This section summarizes the changes to the Solvent model between the 95A release and the 96A release.

New facilities

- The Solvent Model may now be used in combination with the end-point scaling option.

New developments for 95A release

This section summarizes the changes to the Solvent model between the 94A release and the 95A release.

New facilities

- It is now possible to specify a separate Todd-Longstaff mixing parameter for the effective density calculation, rather than using the mixing parameter specified for the effective viscosity calculation. This has necessitated a change to the format of the TLMIXPAR keyword.

Altered keywords

TLMIXPAR	Format changed to allow the input of a separate mixing parameter for the effective density calculation, which is now item 2 in place of the 'DEN'/'NODEN' flag. Pre-95A datasets will need to be modified if the 'DEN'/'NODEN' flag has been used. 'DEN' corresponds to defaulting the density mixing parameter and 'NODEN' corresponds to setting the second mixing parameter to 0.0.

New developments for 94A release

This section summarizes the changes to ECLIPSE Solvent Model between the 93A release and the 94A release.

New facilities

- It is now possible to modify the default straight line relative permeability curves for miscible solvent displacing oil. (See the MSFN keyword.)
- The Todd-Longstaff mixing calculation may now be applied to the viscosity only, leaving the densities to take the pure component values. (See the TLMIXPAR keyword.)

Changes to the behavior of existing facilities

(These may cause changes to the results of a simulation, or alter the form or content of the output.)

- The treatment of capillary pressure has been modified in runs employing the `PMISC` keyword. In the 93a release, the gas P_c was taken as the P_c value at the prevailing lean gas saturation, for all pressures. In the 94a release, the gas P_c is modified to reflect the `PMISC` miscibility function. In a miscible flood, the P_c is taken (as before) as the P_c at the prevailing lean gas saturation. However, in the fully immiscible case, the P_c is taken as the P_c at the total gas saturation (solvent + lean gas).

New keywords

`MSFN`	Miscible relative permeability tables.

Altered keywords

`TLMIXPAR`	New item 2 to prevent the mixing calculation being applied to the density.

New developments for 93A release

This section summarizes the changes to ECLIPSE Solvent Model between the 92A release and the 93A release.

New facilities

- The ECLIPSE 200 Local Grid Refinement option can now be used in conjunction with the Solvent Model.

The Surfactant Model development history

New developments for 93A release

This section summarizes the changes to the Surfactant Model between the 92A release and the 93A release.

New facilities

- It is now possible to use the Surfactant Model in conjunction with the ECLIPSE 200 Polymer Flood Model. The two models are largely independent, however both components are assumed to affect the water viscosity. The water viscosity is modified by two independent multipliers, and hence the polymer and surfactant are assumed not to interact.

The Wellbore Friction Option development history

New developments for 97A release

This section summarizes the changes to the Wellbore Friction Option between the 96A release and the 97A release.

Changes to the behavior of existing facilities

(These may cause changes to the results of a simulation, or alter the form or content of the output.)

- When a workover with an action '+CON' is performed on a friction well, i.e. 'close worst-offending connection and all below it' (see Item 7 of the WECON keyword, for example), this is now interpreted as meaning 'close the connection and all **upstream** of it'. This will affect workovers in horizontal and multilateral wells, where previously the connections to close were selected according to their depth, while now they are selected according to their length down the tubing and multilateral well topology is also taken into account.
- If the WFRICTN or WFRICTNL keyword is repeated for a particular well (e.g. to add a new branch or new connections), ECLIPSE first re-initializes the well structure. Thus the WFRICTN data for the **entire** well must be supplied each time the keyword is used, not just the additional connections. This change prevents problems from occurring in restart runs if the original WFRICTN keywords are still present in the SCHEDULE Section and the SKIPREST keyword is not used.

New developments for 96A release

This section summarizes the changes to the Wellbore Friction Option between the 95A release and the 96A release.

New facilities

- Friction effects can now be calculated in multi-lateral wells.
- The tubing diameter may vary along the length of the well bore.
- The production flow rates from all connections upstream to a given point in the well bore can be output to the Summary file. These can be useful, for example, to show the flow rates from individual branches in multi-lateral wells.
- Cases where the well solution has difficulty in converging may be helped by using OPTIONS keyword switch 55.

New keywords

SUMMARY section

(L)C(O,W,G)FRU

Outputs the oil, water and gas production flow rates from all connections upstream to (and including) the specified connection. The 'L' prefix is required for local grids.

Altered keywords

WFRICTN, WFRICTNL

For record 2 and subsequent records Item 8 (Item 9 for the WFRICTNL keyword) can be used to define a diameter for the tubing at the connection, or range of connections, that is different to the default value defined in record 1.

The order in which the connections are input is now significant. See the entries for the WFRICTN or WFRICTNL keywords for precise details.

RUNSPEC section

FRICTION New item 2 to specify the maximum number of branches joining together at any one point, for multi-lateral wells.

New developments for 94A release

This section summarizes the changes to the Wellbore Friction Option between the 93A release and the 94A release.

New facilities

- Defaults for the connection lengths in the WFRICTN keyword:

  Distances from the bottom hole reference point to the start and end of each connection's perforations can now be defaulted, if the well fully penetrates perpendicularly through the centre of each grid block. If the well penetrates a contiguous row or column of grid blocks, these may be specified as a "range" in a single data record. ECLIPSE will calculate the tubing lengths to each connection by summing the grid block thicknesses in the direction of penetration.

Altered keywords

WFRICTN Defaults are available for Items 4 and 5 of the 2nd and subsequent records.

New items 6 and 7 in the 2nd and subsequent records, to specify penetration direction and a range of grid blocks.

ECLIPSE 300: developments in 99A

New facilities ECLIPSE 300 99A

Grid data

- The FAULTS keyword can now be used to define a set of faults, which can subsequently have their transmissibilities (and diffusivities) modified with the MULTFLT keyword.
- It is now possible to import a grid file from GRID. The IMPORT keyword specifies the name of the binary file to be imported from the GRID program at the current position in the data file.
- It is now possible to override the order of the axes for the nested factorization solver by using the SOLVDIRS keyword from ECLIPSE 100.
- New keywords NINEXZ and NINEYZ will invoke a vertical 9-point scheme, either in the XZ or the YZ plane. This feature is designed for studies of horizontal wells.

Properties and Solution data

- Several Rock Compaction options have been added for compatibility with ECLIPSE 100. In particular, irreversible rock compaction can be used with the existing ROCKTAB keyword by specifying the 1st argument of ROCKCOMP (in RUNSPEC) as IRREVERS. Hysteretic rock compaction can be invoked via the new keyword ROCKTABH which must be requested by the setting the 1st argument of ROCKCOMP as HYSTER. The OVERBURD keyword can be used to index the ROCKTAB or ROCKTABH keywords against the difference between the cell and overburden pressures. If the overburden pressure is greater than the cell pressure, rather than indexing against a negative effective pressure, by setting the 1st argument of the ROCKOPTS keyword to STRESS the effective pressure is redefined as the overburden pressure (entered using keyword OVERBURD) minus the cell pressure.
- It is now possible to allow the capillary pressure modification with surface tension to be active above the reference surface tension value. The user can request to be warned when the surface tension exceeds a predefined value. See the MISCSTR keyword.
- The three-point end-point scaling option has been made compatible with the treatment in ECLIPSE 100. Normalized relative permeability curves can now also be treated consistently with ECLIPSE 100. See the SCALECRS keyword. For further information, refer to the “Saturation Functions” chapter of the “ECLIPSE 300 Technical Description”.
- The ability to specify directional end-point scaling and/or directional relative permeability tables has been added. These are triggered by the modified ENDSCALE keyword and the SATOPTS keyword, both specified in the RUNSPEC section.
- It is now possible to obtain debug information about the scaled relative permeability curves for selected grid cells. See the EPSDEBUG keyword.
- The WSF and GSF keywords have been available as aliases to the SOF2 and SGFN keywords for use in GASWAT runs. The keywords are now documented and can be output via the RPTPROPS keyword.

- An alternative gravity drainage model is now available for dual porosity systems. The model is based on a formulation proposed by P. Quandalle and J.C. Sabathier in "Typical Features of a Multipurpose Reservoir Simulator" SPE Reservoir Engineering, November 1989. The model can improve the match with finely discretized models in mixed wettability cases (where the *Pc* curve is both positive and negative depending on the saturation). See the `GRAVDRM` keyword in the `RUNSPEC` section and the "Dual Porosity" chapter of the "ECLIPSE 300 Technical Description"
- The aquifer keywords `AQUCON` and `AQUCT` may now be entered in the `SOLUTION` section, for compatibility with ECLIPSE 100.

Well model

- Keyword `WPIMULT` has been extended to allow connection factors of individual connections to be multiplied by user-defined factors.
- A well may now be moved between groups by re-entering the `WELLSPEC` (or `WELSPECL`) keyword to specify the name of its new group.
- Cut-back limits may be set on the gas and water fractions of a production well and on the pressures of connected grid blocks for a production or injection well. If any of these limits are violated, the well will be set to operate at a reduced target rate. Cut-back reversal limits may also be set, which if violated, will cause the well to operate at an increased target rate. See keyword `WCUTBACK`.
- Extensions to the keyword `WPAVE` now allow the user to avoid two potential discontinuities in the average grid block pressures as reported to the summary file under the control of the `WBP` series of mnemonics. The first discontinuity occurs when a well is shut and the depth correction to the grid block pressures (which uses an average wellbore density) is set to zero. This problem is avoided by using a reservoir density for the depth correction, or switching the correction off altogether. The second discontinuity in the reported pressures occurs when previously closed connections are opened and thus included in the default averaging over all **open** connections. This can be avoided by electing to perform the pressure averaging over all currently **defined** connections, whether open or closed.

Output

- The AIM status of each cell (IMPES or implicit) can now be output to the print or restart files by using the mnemonic `AIM` in `RPTSOL`/`RPTSCHED`/`OUTSOL`.

Performance

- IMPES cases now run faster by reducing some unnecessary re-calculation associated with the AIM option.
- The performance of the linear solver has been improved for cases involving pinchouts and other instances of in-line non-neighbor connections.
- The CPR (Constrained Pressure Residual) acceleration technique is now available in the linear solver. This will typically improve the performance of fully implicit runs, usually arising from black oil or thermal reservoir models. This method is activated by using the `CPR` keyword in the `SCHEDULE` section.

- An improved form of the Nested Factorization preconditioner to solve the linear equations is now used, which will typically improve solver performance in all cases. The preconditioner now uses a ROWSUM constraint rather than a COLSUM constraint. Refer to the "Solutions of Linear Equations" chapter of the "ECLIPSE 300 Technical Description" for more information. Pre-99A functionality can be restored by setting the 20th switch in the `OPTIONS3` keyword > 0.
- It is now possible to modify the convergence tolerances for the flash calculations. This is **not** recommended for general use. See the `FCCRIT` keyword.

Special extensions

Local grid refinement

- By default, all radial refinements have their circles completed automatically. The keyword `NOCIRCLE` can be used to prevent this.
- An option is now available to coarsen the global grid in regions where accuracy is not so important. The keyword `COARSEN` specifies a box of cells in which to apply grid coarsening.
- A local grid version of the `WPIMULT` keyword has been introduced, so that PI multipliers can be set for individual well connections within an LGR.
- The performance of the linear solver on nested LGR cases has been improved.
- Restart files now have enhanced compatibility for nested LGRs where inactive cells are present in the parent grid.

Flux boundaries

- It is now possible to use the `EQUALS, ADD,` and `MULTIPLY` keywords to set the `FLUXNUM` array.
- The `FLUXNUM` array can now be output to the print file and is also output to the `INIT` file.

Multi-segment wells

Multi-segment wells are now available in black oil or compositional mode, as a new extension in the 99A release. This facility is designed for horizontal and multi-lateral wells, although it can be used to provide a more detailed analysis of fluid flow in standard or deviated wells. A wellbore and any lateral branches may be divided into a number of 1-dimensional segments. The multi-phase flow in a segment is solved fully implicitly with the reservoir grid equations. Pressure drop can be calculated from a choice of either a homogeneous flow model, where all phases flow with the same velocity, or from pre-calculated VFP tables. This second option allows the ability to model flow control devices such as chokes and valves. Pressure losses include frictional, accelerational and hydrostatic components. Crossflow is modelled more realistically, as the fluid mixture can vary throughout the well. Also wellbore storage effects are modelled more accurately. Multi-segment wells are defined by first introducing the well specifications in the usual way with keyword `WELLSPEC`, `WELSPECS` or `WELSPECL`, then defining the completions by means of keyword `COMPDAT` or `COMPDATL`. The segment structure is then defined with the keyword `WELSEGS` and finally the completions are located within the segments with keyword `COMPSEGS` (or `COMPSEGL` if the multi-segment well is defined within an LGR). VFP tables to be used for segment pressure drop look-up are specified

with keyword WSEGTABL. The user may modify the properties of individual well segments during the simulation by using the keyword WSEGPROP. Other options include the ability to apply scaling factors to multiply the frictional pressure drops calculated for each segment (keyword WSEGMULT). This option is described in the "Multi-segment Wells" chapter of the "ECLIPSE 300 Technical Description".

Parallel Options

A number of facilities available with serial runs have been extended and/or enhanced for parallel executions; these are outlined below.

- General nested LGRs are now available with the PARALLEL option. The partition of the LGRs is "inherited" from the global grid (i.e. that given via PDIVX, PDIVY, NPROCX, NPROCY). LGRs which cross partition boundaries are split, and will cause an additional communication overhead, as well as some degradation of algorithmic linear solver performance. The LGR keywords which are now fully available are CARFIN, ENDFIN, REFINE, NXFIN, NYFIN, NZFIN, HXFIN, HYFIN, HZFIN.
- The TRACER keyword (as well as TRACK) is now available with the PARALLEL option.
- Numerical aquifers which cross domain boundaries (i.e. the AQUNUM and AQUCON keywords) are now available with the PARALLEL option. If the numerical aquifer is modelled with multiple cells, these cells may span several partitions, and the connections between the aquifer cells and the reservoir may span several partitions. Any aquifer connections which cross partition boundaries will result in extra communication, and thus reduce the parallel performance.

  Carter Tracy aquifers (AQUCT keyword) have a much smaller communication overhead, and thus less degradation of parallel performance will result.
- Dual porosity and dual permeability options (DUALPORO and DUALPERM keywords) are now available with the PARALLEL option.
- The treatment of wells with completions that span multiple partitions has been enhanced, to reduce communication costs. However, the communication overhead of having completions which cross partitions will degrade parallel performance.
- The NNC keyword, which enables any two cells within the reservoir to be connected, is now available with the PARALLEL option. If these connect across partitions, performance degradation will result.
- The GETSOL keyword is now available with PARALLEL.
- The THERMAL keyword is now available with PARALLEL.

Behavioral changes ECLIPSE 300 99A

These may cause changes to the results of a simulation, or alter the form or content of the output.

Properties

- The FACTLI keyword will now be honoured when the value 1.0 is input. In previous versions of ECLIPSE 300 the value 1.0 was taken as a default and hence a FACTLI value was calculated from the gas-oil contact composition in the case of equilibration option 1 (11th data item in EQUIL, single phase systems).

- The PVCO keyword now requires 6 columns instead of 5, for compatibility with the ECLIPSE 100 keyword. The requirement can be over-ruled by setting the 24th switch of the OPTIONS3 keyword.
- If an Li factor of less than 0.9 and greater than 1.1 is calculated to match the GOC when using compositional equilibration option 1, an error message will be issued. This error message can be downgraded to warning by using the 33rd switch in the OPTIONS3 keyword.
- The stability test algorithm used in the flash calculation to decide whether a cell is single or two phase has been modified. Single phase cells that convert to a two phase state now utilize a vectorized flash algorithm, backed up by a scalar method if the calculation fails to converge. Prior to 99A the scalar algorithm was used for all cells changing to a two phase state. This change may lead to small differences in the solution (within convergence tolerance). The pre-99A algorithm can be requested by setting the 29th switch in the OPTIONS3 keyword.

Well model

- For injection wells with THP limits imposed, the potential is now correctly calculated as the maximum flow rate subject to both THP and BHP constraints. Previously the THP constraint was not consistently honoured in well potential calculations. If such injection wells are under group control, their behavior may change as well potential rates are used to set default guide rates and thus determine how a group injection target is apportioned between the wells. Pre-99A functionality can be restored, if required, by setting the 32nd switch of the OPTIONS3 keyword.
- The calculation of the reported group and field liquid composition has changed. Previously this was calculated by summing the well component volume rates, while now the summation is performed over well component molar rates. Pre-99A functionality can be restored by setting the 32nd switch of the OPTIONS3 keyword.
- The 2nd argument of keyword SEPCOND, which was previously ignored, now sets a default separator for wells subordinate to the named group.

Output

- The SATNUM array is now output to the INIT file when the SATNUM keyword has not been entered (SATNUM = 1 everywhere). This is useful when plotting scaled relative permeability curves in GRAF.
- The summary quantities BDENO, BDENG and BDENW are now output as densities rather than pressure/height.
- The Gas Accounting tables in the print file have been modified. The ‘Reinj. Volume’ now refers to re-cycled produced gas that is re-injected. Similarly, the Injection Fractions table now shows the fractions of advanced, re-cycled and make-up gas used to form the re-injected gas; these fractions sum to unity. The Gas Accounting tables are explained in the “Wells & Groups” chapter of the “ECLIPSE 300 Technical Description”.
- By default well and group names are now internally converted to, and reported as, upper case names. The case of letters must not be used alone in a name to distinguish between wells or between groups. This internal case conversion may be switched off using 31st switch in the OPTIONS3 keyword.

- A message is now written if a production well VFP table contains at least one point where different THP values result in the same BHP value for a given flow rate, water fraction and gas fraction. This may cause problems for the well model if a well flows in this region of the VFP table while not under THP control, because a unique THP value cannot be found for a given BHP value. Also, the warning message written if a production well VFP table contains BHP values that decease with increasing THP values is now only written once per table.

Miscellaneous

- The `RUNSPEC` section keyword `OPTIONS` has been renamed `OPTIONS3` to distinguish it from the ECLIPSE 100 keyword, whose arguments are totally different. It can now be entered in any section. Entry of an `OPTIONS` keyword will now result in an error message.
- The keyword `DEBUG` has been renamed `DEBUG3` to distinguish it from the ECLIPSE 100 keyword, whose arguments are totally different. Entry of a `DEBUG` keyword will now result in an error message.
- The `REGIONS` section keywords (`FIPNUM`, `SATNUM` etc.) must contain enough data to fill the current input box; missing data will result in an error. In previous versions of ECLIPSE 300 any missing data for these keywords was defaulted.

Special extensions

Local grid refinement

- In previous versions of ECLIPSE 300, the circle was not completed for radial refinements. Now, by default, all radial refinements specified will have their circle completed, unless this option is switched off. To do this, the NOCIRCLE keyword should be specified within the RADFIN/ENDFIN bracket of the radial refinement(s) that you do not wish to be completed.

Parallel Options

- The CPU time, elapsed time and memory usage data that are written to the Print file at every report step represent the maximum usage across all processes, not that of the first process as in 98a.

New keywords ECLIPSE 300 99A

New RUNSPEC section keywords

E100NOSI	Switch to NOSIM mode if any ECLIPSE 100 keywords are encountered that are not recognized or are substantially different from their ECLIPSE 300 counterparts.
GRAVDRM	Invokes the alternative gravity drainage model for dual porosity runs.
NINEXZ	Select NINEPOINT option only in the XZ-plane.
NINEYZ	Select NINEPOINT option only in the YZ-plane.
ROCKCOMP	Indicates that Rock Compaction is to be used.
SATOPTS	Specifies options for directional and hysteretic relative permeabilities.
WSEGDIMS	Indicates that multi-segment wells are to be used and provides dimensioning data.

New GRID section keywords

COARSEN	Specifies a box of cells for grid coarsening.
FAULTS	Defines a set of faults, which can have their transmissibilities (and diffusivities) modified using the MULTFLT keyword.
IMPORT	Imports grid file data from the GRID program.
MULTFLT	Modifies the transmissibilities (and diffusivities) across a fault, previously defined using the FAULTS keyword.
NOCIRCLE	All radial refinements have their circles completed automatically. To switch this option off for any refinement, use the NOCIRCLE keyword within the RADFIN/ENDFIN bracket of that refinement.
SOLVDIRS	Overrides the principal directions for the linear solver.

New PROPS section keywords

EPSDEBUG	Allows the user to output the scaled relative permeability and capillary pressure data to the debug file, either as a table or as a GRAF user file.
OVERBURD	Specifies that Rock Compaction table are indexed against an effective pressure equal to the cell pressure minus the overburden pressure.
ROCKOPTS	Sets options associated with Rock Compaction.
ROCKTABH	Data tables for the Hysteretic Rock Compaction option.
SCALECRS	Allows the three-point end-point scaling method and normalized relative permeability curves to be treated consistently with ECLIPSE 100.
SOLVDIRS	Overrides the principal directions for the linear solver.

New REGIONS section keywords

KRNUM	Specifies table numbers for directional relative permeabilities.

New SOLUTION section keywords

FCCRIT	Modifies flash convergence criteria.

New SUMMARY section keywords

AIMFRAC	Outputs the fraction of implicit cells in an AIM run.
ELAPSED	Outputs the elapsed time in seconds.
MLINEARS	Outputs the number of linear iterations per timestep.
MSUMLINS	Outputs the total number of linear iterations since the start of the run.
MSUMNEWT	Outputs the total number of Newton iterations since the start of the run.
NEWTON	Outputs the number of Newton iterations used for each timestep.
NLINEARS	Outputs the average number of linear iterations per Newton iteration for each timestep.
PERFORMA	Outputs performance data for the run; this includes all of the mnemonics TCPU, ELAPSED, NEWTON, NLINEARS, MLINEARS, MSUMLINS and MSUMNEWT.
SOFR, SWFR, SGFR	Multi-segment well segment oil, water and gas flow rate.
SOFV, SWFV, SGFV	Multi-segment well segment oil, water and gas flow velocity.
SOHF, SWHF, SGHF	Multi-segment well segment oil, water and gas holdup fraction.
SPR	Multi-segment well segment pressure.
SPRD	Multi-segment well segment pressure drop.
SPRDF, SPRDH, SPRDA	Multi-segment well segment friction, hydrostatic and acceleration pressure drop components.
SWCT, SGOR, SOGR, SWGR	Multi-segment well segment water cut, gas-oil ratio, oil-gas ratio and water-gas ratio.
TCPU	Outputs the current CPU usage in seconds.
TIMESTEP	Outputs the timestep length.

New SCHEDULE section keywords

COMPSEGL	Defines the location of multi-segment well completions in a local grid.
COMPSEGS	Defines the location of completions in a multi-segment well.
CPR	Invokes the Constrained Pressure Residual (CPR) linear solver.
FCCRIT	Modifies the flash convergence criteria.
NEXTSTEP	Is an alias for NEXT, setting the maximum size of the next timestep.
WCUTBACK	Sets well cut-back limits to constrain water and gas coning in a production well by reducing the flow rate.
WELSEGS	Designates a well to be a multi-segment well, and defines its segment structure.

`WPIMULTL` Local grid version of `WPIMULT`, to apply PI multipliers to individual well connections.

`WSEGMULT` Multiplies segment frictional pressure drops in a multi-segment well.

`WSEGPROP` Modifies segment properties in a multi-segment well.

`WSEGTABL` Calculates multi-segment well segment pressure drops from VFP tables.

New global keywords

OPTIONS3 The keyword replaces the OPTIONS keyword, which previously only belonged to the RUNSPEC section.

Altered keywords ECLIPSE 300 99A

Altered RUNSPEC section keywords

ENDSCALE The keyword can now take up to 4 arguments for compatibility with the ECLIPSE 100 keyword.

EQLDIMS A 5th argument has been added, setting dimensions for tracer initialization, for compatibility with ECLIPSE 100.

HYKR The keyword is now documented.

HYPC The keyword is now documented.

MISCIBLE The keyword can now take up to 3 arguments for compatibility with the ECLIPSE 100 keyword.

NOHYKR The keyword is now documented.

OPTIONS The keyword has been renamed OPTIONS3 to distinguish it from the ECLIPSE 100 keyword.

It can now be entered in any section, and is no longer specifically a RUNSPEC section keyword.

New switches 16 to 35 have been added (see the OPTIONS3 keyword documentation).

WELLDIMS Item 10 is now documented.

Altered GRID section keywords

AQUCON, AQUCT

Are now also valid in the SOLUTION section, for compatibility with ECLIPSE 100.

EQUALS, ADD, MULTIPLY

Can now be used with the FLUXNUM mnemonic.

INRAD As well as defining the inner radius for radial geometry, the keyword can also be used to define the inner radius of a radial local grid refinement by placing it within the appropriate RADFIN/ENDFIN bracket.

RADFIN The keyword now has an additional argument in position 9 to allow compatibility with the ECLIPSE 100 keyword. The default value (i.e. 1*) should be used in this version of ECLIPSE 300.

RPTGRID The FLUXNUM array can now be output using the FLUXNUM mnemonic.

Fault data can now be output using the FAULTS mnemonic.

The NNC mnemonic is now able to output both non-neighbor connections and in-line neighbor connections (which have been removed), if appropriate.

Altered PROPS section keywords

FACTLI	Now honours a value of 1.0 in all cases, instead of interpreting this value as a default.
GSF	The keyword is now documented.
MISCSTR	New data items 2 and 3 to input the maximum expected surface tension and the maximum surface tension used to scale the capillary pressures.
PVCO	The keyword now requires 6 columns instead of 5, for compatibility with the ECLIPSE 100 keyword. The requirement can be over-ruled by setting the 24th switch of the OPTIONS3 keyword.
RPTPROPS	New mnemonics have been added to output end point scaling arrays.
WSF	The keyword is now documented.

Altered REGIONS section keywords

RPTREGS	New mnemonics to output directional relative permeability table numbers.

Altered SOLUTION section keywords

RPTSOL	New arrays are now available and aquifer grid block connections can now be output using the AQUCON mnemonic.

Altered SUMMARY section keywords

BDENO, BDENW, BDENG

These quantities are now output as densities (mass per unit volume, e.g. lb/ft3) rather than pressure/height.

Altered SCHEDULE section keywords

CVCRIT	New items 16 to 19 control convergence criteria for the CPR linear solver.
GRUPINJE	The MIX option is now documented in item 2.
NSTACK	New item 2 to control stack size for the CPR linear solver.
OUTSOL	New arrays are available for output to the RESTART file.
RPTSCHED	New arrays are available for output.
SEPCOND	The second argument may be used to define a group for which this separator acts as a default. All wells that can trace their parentage directly to this group will be assigned this default separator unless a separator has been explicitly defined using the WSEPCOND or WELLSPEC keyword. Previously this argument was ignored.
WPAVE	New items 3 and 4 allow control over how reported grid block pressures are depth corrected and whether the pressures are averaged over all connections or only those currently open.
WPIMULT	New items 3 to 5 allow PI multipliers to be defined for individual connections.

Altered global keywords

`DEBUG` The keyword has been renamed `DEBUG3` to distinguish it from the ECLIPSE 100 keyword.

ECLIPSE 300: developments in 98A

New developments

Grid data

- It is now possible to use the Leverett J-function. See the JFUNC keyword and the chapter on end point scaling in the technical description.
- The keyword MAPUNITS can now be specified to define the units used for the MAPAXES data, which defines the simulation grid position with respect to maps.
- The REFINE keyword can now be used in the SOLUTION section.

Properties & equilibration

- An option to make gas/oil relative permeabilities velocity dependent has been added. See the VELDEP, VDKRO and VDKRG keywords and the chapter on Non-Darcy flow in the technical description.

Well model

- Well Inflow can be modelled by a modified form of the Fevang and Whitson Pseudo-Pressure technique. See the PSEUPRES and PICOND keywords and the chapter on Pseudo Pressure in the technical description.
- Well revivals are now possible when the well has been shut due to either a WCT, GOR or WGR economic reason. Specifying WREV in the seventh argument of WECON accomplishes this.
- The GPMAINT keyword can be used to maintain the average pressure in user defined pressure maintenance regions using previously defined groups. These regions can be defined with the PMANUM keyword with FIELD as default. An elementary integral control theory algorithm is used to control either production or injection.

Output

- It is now possible to output index files, RSSPEC and INSPEC files, describing the arrays written to the INIT and RESTART files. See the RUNSPEC keywords INSPEC and RSSPEC and the chapter on file formats in the technical description.

Performance

- It is now possible to suppress the potential reversal chops. See the OPTIONS keyword.

Behavioral changes

These may cause changes to the results of a simulation, or alter the form or content of the output.

Properties

- The definition of the reference surface tension, if defaulted, is now taken to be the field average initial surface tension rather than the cell-by-cell initial value. See the `MISCSTR` keyword and the 15th data item in the `OPTIONS` keyword.
- The `DPCDT` option in `AIM` cases has been modified. In implicit cells rather than limit the capillary pressure change, which can cause convergence problems, the rate of change of capillary pressure with respect to saturation is limited. See the `DPCDT` keyword in `RUNSPEC`.

Equilibration

- The equilibration has been modified in cases where water-oil and gas-oil transition zones closely overlap. If in these cases the calculated oil saturations are negative then the water and gas saturations are re-calculated from the gas-water capillary pressure, which is taken as the sum of the water-oil and gas-oil capillary pressures. The old method is available for back compatibility by setting the 12th `OPTIONS` switch to 1.

Output

- The memory usage is now reported along with the CPU and elapsed time during the run.
- The dataset echo now reports the `INCLUDE` file depth rather than the line numbers.
- If capillary pressures are output to the restart file for graphics the mnemonics have been changed from `PCW,PCG,PCO` to `PCWD,PCGD,PCOD` to prevent a conflict with the end-point scaling keyword, `PCW,PCG` on the `INIT` file.
- When any end-point versus depth keywords are encountered in the `SOLUTION` section, then the processed endpoints are written to the `INIT` file. When any endpoints are explicitly entered, the reported values in the `INIT` file reflect values copied to refined cells.

New keywords

New RUNSPEC section keywords

PHOST	Allows the user to specify which domain will be executed on which processor, within the parallel computing options.
VELDEP	Specifies which model for gas and oil capillary number, gas and oil Forchheimer model are to be used.

New GRID section keywords

JFUNC	Allows the user to activate the Leverett J-function option, which scales the water-oil and/or gas-oil capillary pressure functions according to the grid porosity and permeability.
MAPUNITS	Defines the units of measurement used for the MAPAXES data.

New PROPS section keywords

VDKRG	Specifies the control parameters for the *gas* capillary number and Forchheimer models. These models are an extension of the existing relative permeability model and therefore data must be provided for each saturation region.
VDKRO	Specifies the control parameters for the *oil* capillary number and Forchheimer models. This keyword mirrors the corresponding keyword for gas, VDKRO.

New SCHEDULE section keywords

PICOND	Changes default settings for the Pseudo-Pressure option, namely the spacing of table points below and above the saturation pressure as well as the explicit damping factor.
PSEUPRES	Activates the Pseudo-Pressure option for Well Inflow.
GPMAINT	Data for pressure maintenance groups.

Altered keywords

CVCRIT	New 10th data item to set minimum linear convergence criterion.
	New 11th data item to set maximum acceptable non-linear criterion.
	New 12th data item to set maximum acceptable sum of non-linear residuals.
	New 13th data item to set maximum molar density change in a non-linear iteration.
	New 15th data item to set maximum pressure change in a non-linear iteration.

`OPTIONS` New 11th data item to suppress potential reversal chop.

New 12th data item to request the pre-98a equilibration method.

New 13th data item to request the pre-98a sum of specific volume derivatives.

New 14th data item to request the pre-98a use of average specific volume normalization

New 15th data item to request the pre-98a default surface tension definition.

`DPCDT` In the case of implicit AIM cells the dP_c/dT limit has been replaced by a limit on the $dP_c/dSaturation$ which can be controlled be the second data item.

Removed GRID section keywords

`HTFIN` The option which dictates the size ratios of each cell in a local grid refinement in the azimuthal direction is no longer available.

`NRFIN` If a radial local grid refinement covers more than one global cell in the radial direction, `NRFIN` was used to dictate how many local cells each of the global cells was divided into. This keyword is no longer available. Refer to argument 6 in the `RADFIN` keyword.

`NTFIN` The `NTFIN` keyword was used to dictate how many local cells each of the global cells was divided into, given that the radial local grid refinement covers more than one global cell in the azimuthal direction. This keyword can no longer be used. Refer to argument 7 in the `RADFIN` keyword.

ECLIPSE 300: developments in 97A

New facilities

Parallel processing

The 97A version of ECLIPSE 300 may be run in parallel on suitable computers. These must support a message passing system. Examples are the IBM SP2 & Silicon Graphics machines.

Keywords involved are PARALLEL, NPROCX, NPROCY, PDIVX and PDIVY.

CO_2 solution in the aqueous phase

CO_2 may be allowed to dissolve in the aqueous phase. Solubility data may be entered using the SOLUBILITY keyword or a default set generated internally.

Keywords involved are CO2SOL, RSWVD and SOLUBILITY

MEMORY keyword

Allows the user to set the memory requirement dynamically. The code must be installed with this option set up (the DYNAMIC select code in the main routine). A single load module may then be used, and the MEMORY keyword entered to request the number of Mbyte of main storage required and the amount of character storage required. (Character storage is in 1000 character*8 units). The default is 100 MByte and 10,000 character*8 words.

New SUMMARY section keywords

Well, group and field reservoir volume flow and totals at reference conditions:

Production rates & totals

WOVPR, WGVPR, WWVPR, WOVPT, WGVPT, WWVPT GOVPR, GGVPR, GWVPR, GOVPT, GGVPT, GWVPT FOVPR, FGVPR, FWVPR, FOVPT, FGVPT, FWVPT

Injection rates and totals

WOVIR, WOVIT, WWVIR, WWVIT, WGVIR, WGVIT GOVIR, GOVIT, GWVIR, GWVIT, GGVIR, GGVIT FOVIR, FOVIT, FWVIR, FWVIT, FGVIR, FGVIT

Total reservoir hydrocarbon molar density

BMLST will output the block total hydrocarbon molar density. This is the sum over components of the BMLSC values.

Stage gas compositions for wells. WYMF1, WYMF2 and WYMF3 allow the stage gas compositions to be obtained.

Wet gas with well history matching option

An extra, 11th, argument has been added to the WCONHIST keyword to enter wet gas production rates. The WHISTCTL keyword has the option of requesting that the well follows the wet gas production.

Changes to hysteresis keywords

A new set of keywords have been introduced to control hysteresis options. The HYST keyword is required to enable the hysteresis option, then HYKR or NOHYKR to control relative permeability hysteresis and HYPC or NOHYPC to control capillary pressure hysteresis.

AIMFRAC keyword

The treatment of implicit cells in the AIM option has been changed. The program will now set the criterion for making a cell implicit so that a target fraction of the cells are selected. The default value of this target fraction is 1. This faction may be modified using the AIMFRAC keyword, although 1 is usually a reasonable value. The percentage of cells which are implicit is reported in the log and print file output at the end of the one line step summary.

ACTION & ACTDIMS keywords

The ACTION keyword allows a set of well or group schedule section keywords to be stored with a condition. If this condition is satisfied the keywords will be processed. At present the conditions may only be applied to field oil, water and gas production rates, field WCT and GOR and average field pressure.

The number of ACTION clauses allowed and the number of lines in each may be set using the ACTDIMS keyword in the SCHEDULE section.

TRACK keyword

This keyword enables the production and fluid in place from above and below either the original gas-oil contact or a specified depth to be tracked. This is done internally by setting up a set of tracers to follow the components. These can be attached to all the components, for the highest accuracy, or to a subset, in which case the missing component tracer concentrations are obtained by interpolation.

Production from above and below the GOC or specified depth is accounted separately in the fluid-in-place reports, the field production report, the group reports and the well reports. A range of new summary mnemonics are available to follow this production split:

FOPRA, FOPRB, FGPRA, FGPRB	Field oil and gas rates (above and below)
FOPTA, FOPTB, FGPTA, FGPTB	Field oil and gas totals (above and below)
GOPRA, GOPRB, GGPRA, GGPRB	Group oil and gas rates (above and below)
GOPTA, GOPTB, GGPTA, GGPTB	Group oil and gas totals (above and below)
WOPRA, WOPRB, WGPRA, WGPRB	Well oil and gas rates (above and below)
WOPTA, WOPTB, WGPTA, WGPTB	Well oil and gas totals (above and below)

PSSTA keyword

This keyword uses the saturation pressure calculation to assign single phase hydrocarbon states as oil or gas, rather than the Li correlation. It gives a more precise state identification although this involves the extra cost of the P_{sat} evaluation.

GDFILE

This keyword allows the definition of a grid to be read from an ECLIPSE grid file. This may be formatted or unformatted, and provides all the geometry information for the simulation. This also provides a method by which third party gridding packages can input geometry information.

Saturation clipping

If `OPTIONS` argument 3 is set to 1 any one phase cells which are flashed after equilibration to two phases are clipped back to one phase. This is to treat a problem which may occur above a gas-oil contact in compositional modelling, in which the fluid is saturated at the contact, so that lower pressures at higher level cause over-saturation and small oil saturations to appear. The true solution is that the gas is saturated at the reservoir pressure, with no oil saturation. The saturation clipping option will set up this state.

Ninepoint option

The `NINEPOINT` keyword will add diagonal non-neighbor connections to reduce grid orientation in cases of adverse mobility injection.

Separate MISCKR & MISCPC keywords

These allow the `MISCIBLE` option to be applied separately to relative permeability and capillary pressure calculations. If `MISCIBLE` is used alone, this is the same as `MISCKR` and `MISCPC`.

RESVNUM keyword

When entering `COORD` data for cases with multiple reservoirs, it is sometimes more convenient to enter data for each reservoir separately. This may be done by entering `RESVNUM` to define a reservoir index and then entering data for that reservoir only.

DGRDT keyword

This keyword may be used to limit the rate at which defaulted guide rates may vary for wells under group control.

WELDEBUG keyword

This keyword enables debug output to be requested for a specific well.

Changes to existing facilities

EQUALS etc.

EQUALS, MULTIPY, ADD and COPY may be used with SIGMA, SIGMAGD and DZMATRIX in dual porosity runs.

Output to INIT file

Dual porosity arrays such as SIGMA, SIGMAGD and DZMATRIX are output to the INIT file. If end point scaling is used endpoint data is also output.

End-point scaling

Vertical end-point scaling has been made compatible with ECLIPSE 100. The old ECLIPSE 300 version can be invoked by setting the 6th parameter of the OPTIONS keyword to 1.

RPTPRINT changes

If RPTPRINT argument 3 is set greater than 1 then reservoir oil and reservoir gas will be flashed separately to surface conditions by region.

If RPTPRINT argument 14 is set greater than 0 then VFP table are output to the print file in neat format.

Alternative GRID section keyword names

Keywords such as PERMI are now accepted as alternatives to PERMX or PERMR. This is to enable Fill output files to be read.

Extra information in restart files

When a RESTART record is written so that a flexible restart can be performed (using the RESTART keyword in the SOLUTION section), this file now contains extra information such as the initial surface tensions or the maximum saturations, as required by options such as MISCIBLE. This allows the restart run to more accurately reproduce the continued base run. This change is back-compatible - older restart files which do not contain these items may still be used with 97a.

Output of flows to GRAF

OUTSOL arguments now exist which enable flows to be written to the restart files. These may be used to produce arrow diagrams showing fluid movement. The mnemonics are FLOC1 for component 1, FLOC2 for component 2 etc.

On most UNIX workstations the buffers for the save and restart files are flushed after each write operation. This avoids problems with incomplete files, (particularly in the unified case), which have previously occurred when a run ended early.

COORDSYS keyword

A new argument allows transmissibilities to be generated between cells with different reservoir numbers.

Output of NNC data to grid file

Surface tensions for cells containing one hydrocarbon phase

The reference oil-gas surface tension value is assigned to single phase hydrocarbon cells, rather than a zero value.

AQUDIMS keyword

The dimensions input with this keyword are now treated in the same way for numerical and analytical aquifers. The second dimension is the number of lines of `AQUCON` connection data in all cases.

ECLIPSE 300: developments in 96A

New facilities

LBCCOEF

This keyword allows non-default coefficients to be entered for the Lorentz-Bray-Clark correlation.

IMPSAT mode

This keyword selects the IMPSAT solution option. This treats pressure and phase saturations implicitly, but phase compositions explicitly. This method is ideal for runs involving large capillary pressures, which would cause instability in IMPES or AIM methods.

GASWAT

The GASWAT option provides a method of modelling gas phase/aqueous phase equilibria using an equation of state. The Peng Robinson equation of state is modified following the suggestions of Soreide and Whitson to obtain accurate gas solubilities in the aqueous phase. This currently only supports two phase gas-water systems: this will be extended to a full three phase equation of state model in 97a.

- Associated keywords:

`AQCOEF`	A-coefficient terms for water phase
`BICAQ1`	Non-default aqueous phase binary coefficients: constant term
`BICAQ2`	Non-default aqueous phase binary coefficients: T term
`BICAQ3`	Non-default aqueous phase binary coefficients: T^2 term
`CALVAL`	Component calorific values
`GQUALITY`	Set group gas quality target

- Associated summary keywords:

`WEPR`	Well energy production rate
`WEPT`	Well energy production total
`WEIR`	Well energy injection rate
`WEIT`	Well energy injection total
`WGQ`	Well gas quality
`GEPR`	Group energy production rate
`GEPT`	Group energy production total
`GEIR`	Group energy injection rate
`GEIT`	Group energy injection total
`GGQ`	Group gas quality
`FEPR`	Field energy production rate

FEPT	Field energy production total
FEIR	Field energy injection rate
FEIT	Field energy injection total
FGQ	Field gas quality

New Hysteresis options

Capillary pressure hysteresis has been added. If only relative permeability hysteresis is required the NHOYPC keyword can be used.

A new method of treating hysteresis has been added. This uses a weighting fraction between the drainage and imbibition curves. This is due to J. Jargon and is selected using the HYSTJ keyword. HYSTK becomes an alias for the HYST keyword to select the alternative Killough method.

Local Grid Refinement options

Local grids may now adjoin or be nested. The AMALGAM keyword to amalgamate LGRs is accepted. The CARFIN and RADFIN keywords which define a local grid both have an optional 12th argument to select the parent grid: this is FIELD by default for high level grids.

Summary file keywords for LGR completion data (LOCPR etc.) and LGR cell data (LBSGAS etc.) have been added.

ROCKNUM

This is like PVTNUM, but allows rock function table numbers to be defined independently of pressure table numbers. If ROCKNUM is not set, it defaults to the PVTNUM values.

Production system options

The handling of gas in the production system has been developed. The main changes are:

- Fuel, sales and re-injection ordering

  The order in which fuel, sales gas and re-injection are taken from the produced gas is variable.

  The new keyword is WTAKEGAS - possible arguments are FRS, FSR, RFS, RSF, SFR, SRF. The default is FSR (fuel, then sales, the remainder being available for re-injection).

- Excess gas accounting

  Gas not used for fuel, re-injection or specified sales will be separately accounted as excess gas. Export gas is then sales gas plus excess gas. Export gas rates and totals are reported in the print file and as new summary file items.

- Satellite groups

  A gas source or sink may be introduced into the production system by adding a satellite group. The satellite groups act like fixed rate gas wells.

  The new keywords are GSATPROD, GSATINJE and GSATCOMP.

- Injection gas mixtures

  Injection gas specification. A new type of injection fluid has been added: this is a mixture. A mixture may be defined by using the keywords WINJMIX or WINJORD.

- Summary file keywords for gas handling:

GSGR	Group sales gas rates
GSGT	Group sales gas totals
GFGR	Group fuel gas rates
GFGT	Group fuel gas totals
GRGR	Group reinjection gas rates
GRGT	Group reinjection gas totals
GEGR	Group export gas rates
GEGT	Group export gas totals
GEXGR	Group excess gas rates
GEXGT	Group excess gas totals
FSGR	Field sales gas rates
FSGT	Field sales gas totals
FFGR	Field fuel gas rates
FFGT	Field fuel gas totals
FRGR	Field reinjection gas rates
FRGT	Field reinjection gas totals
FEGR	Field export gas rates
FEGT	Field export gas total
FEXGR	Field excess gas rates
FEXGT	Field excess gas total

- Availability control of wells and groups

  If a well or group is re-injecting the gas production of a well or group we have previously only considered the composition being injected. The amount of gas available has affected this only if make-up gas has been specified.

  If the new WAVAILIM switch is used, the well or group will switch to availability control if limited by the produced gas rates. This is rather like re-injection control. It will switch back to the next most restrictive control if this ceases to be a constraint.

GETGLOBAL

Keyword to extract global solution only when reading solution data.

RSM

Turn on output to RSM file during run.

RPPO, RPPG and RPPW summary items

Region pressures corrected to datum. Note oil and gas are hydrocarbon weighted, water is pore volume weighted.

New SUMMARY keywords

FOIPR, FGIPR, FWIPR for Field Oil, Gas and Water In Place at Reservoir conditions.

E100 compatible well options

WELDRAW to control well drawdown

WPIMULT to modify well PI values.

Changes to existing facilities

COMPFL

The separate flashing through the separator system of the flow from each of the completions of a well, previously done only when the COMPFL keyword was specified, is now carried out for all runs. The COMPFL keyword will still be read, but has no effect.

PINCH

The PINCH keyword has been modified to include further E100 compatible options.

For back compatibility, the 95a pinchout treatment can still be obtained using the fourth argument of the OPTIONS keyword.

Capillary pressure treatment

In IMPES and AIM modes capillary pressure can cause instability. Rather than freezing the capillary pressure we now limit the rate at which it can change from step to step. This stabilizes the run without seriously affecting the modelling of imbibition driven recovery. By default IMPES and AIM modes have a maximum rate of change of capillary pressure $(dP_c)/dT = 0.01$ Psi/Day. This limit can be reset using the DPCDT keyword. The option of freezing the capillary pressures is retained through the FREEZEPC keyword - this will have the same effect as setting DPCDT to 0.

The FULLIMP and IMPSAT modes have no restriction on the changes in capillary pressure allowed, although DPCDT and FREEZEPC may be used if required.

Treatment of hydrostatic head

The hydrostatic head terms in the well model, which correct the pressure from the bhp reference depth to the wellbore pressure in a given completion are now treated implicitly. This improves the stability of the well model in many cases.

Suppressing pressure setting in gas-oil contact equilibrium

If there is a fluid contact (oil-gas in most cases, or water-gas in GASWAT) an extra 11th argument on the EQUIL keyword may be used to suppress the setting of the pressure as the contact to the fluid saturation pressure. This will result in a non-equilibrium initial state, but may be better than a large pressure shift when the compositions are poorly known.

Treatment of sales gases

Sales gas rates should be specified using the WELLSALE and GRUPSALE keywords. A warning will be issued if WELLINJE or GRUPINJE are used.

WELLDIMS and CHANDIMS

The previously unused 7th argument is now used for the maximum number of mixtures. A new, 9th, argument has been added for the maximum number of mixture items.

ECLIPSE 500: developments in 99A

New facilities ECLIPSE 500 99A

General

- ECLIPSE 500 is Year 2000 compliant.

RUNSPEC

- New keywords `NINEXZ` and `NINEYZ` will invoke a vertical 9-point scheme, either in the XZ or the YZ plane. This feature is designed for studies of horizontal wells and is also available in ECLIPSE 300.

GRID & Edit

- The `COARSEN` keyword can now be used to specify a box of cells for grid cell coarsening.

Properties & equilibration

- Water densities are now calculated using analytic expressions by default. Previously, these expressions were obtained by specifying `THANALB`. A new keyword `THTABB` will cause the densities to be calculated from tables, as before.
- If `THTABB` is specified, then `OPTIONS3`(22) will cause the program to use an alternative set of tables from Haywood instead of the tables from Perry and Green. Warnings are now given if any of the tables are extrapolated.
- Temperature-dependent K-value tables can be input. Use the keyword `KVTEMP` to specify up to 9 temperatures at which K-value tables are provided, then use keywords `KVTABT1`, `KVTABT2`,... to set the K-value tables at these temperatures.
- ECLIPSE 500 will warn if calculated gas densities > oil densities > water densities. This can happen if, for instance, the specified Z-factors are inconsistent or if ZMFVD has heavy but volatile components below the oil-water contact.
- Several Rock Compaction options have been added for compatibility with ECLIPSE 100. For more details see the ECLIPSE 300 manual.

Well model

- OPTIONS3(28) sets the gravity density in the well-bore to the initial gravity density in the reservoir. This reduces non-linearities, speeds-up the run, and may be a better model.

Behavioral changes ECLIPSE 500 99A

These may cause changes to the results of a simulation, or alter the form or content of the output.

GRID

- The check for a rock-filled state has changed to “pore volume < MINPV” instead of “pore volume < 10**(-20)”, where MINPV is the value set in the MINPV keyword
- Pore volume multipliers are calculated using rock region numbers if they have been set using the ROCKNUM keyword, instead of always using PVT region numbers (from PVTNUM).

Solution

- A bug was found and fixed that caused an erroneous gas phase pressure to be calculated and used during initialization with the EQUIL keyword. This may result in a different initialization with this release of ECLIPSE 500. An options switch has been put in to maintain back-compatibility with earlier versions. If significant differences in initialization occur, set OPTIONS3(35)>0.

Output

- As in ECLIPSE 300, the summary quantities BDENO,BDENG and BDENW are now output as densities rather than pressure/height.

New keywords ECLIPSE 500 99A

New RUNSPEC section keyword

ROCKCOMPSpecifies Rock Compaction is to be used.

New GRID section keywords

COARSEN Specifies a box of cells for grid cell coarsening.

New PROPS section keywords

The following keywords are described in more detail in the ECLIPSE 300 manual.

OVERBURDAllows Rock Compaction table to be indexed against effective pressure = cell pressure - overburden pressure.

ROCKOPTSSets options associated with Rock Compaction.

SCALECRSAllows the three-point end-point scaling method and normalized relative permeability curves to be treated consistently with ECLIPSE 100.

THTABBUse tabular lookup of water densities instead of the default analytic function.

New global keywords

OPTIONS3The keyword replaces the OPTIONS keyword, which previously only belonged to the RUNSPEC section.

Altered keywords ECLIPSE 500 99A

Altered RUNSPEC section keywords

OPTIONSThe keyword has been renamed OPTIONS3 to distinguish it from the ECLIPSE 100 keyword.

It can now be entered in any section, and is no longer specifically a RUNSPEC section keyword.

New switches 16 to 34 have been added (see the OPTIONS3 keyword documentation).

Altered PROPS section keywords

BDENO, BDENG, BDENW

Now output as densities (mass per unit volume, e.g. lb/ft3) rather than pressure/height.

Changes to existing facilities ECLIPSE 500 99A

RUNSPEC

- The default option is now LIVE instead of DEAD.

Properties

- The 4th and 5th parameters of KVCR which calculates K-values using Crookston's formula can now be negative. A warning is printed instead of an error message.

Bibliography

Appendix H

Fetkovich, M. J. *A Simplified Approach to Water Influx Calculations - Finite Aquifer Systems* [Ref. 1]
J. Pet. Tech, *Page 814-828,* July 1971

Carter, R. D. and Tracy, G. W. *An Improved Method for Calculating Water Influx* [Ref. 2]
Trans. AIME 219; J. Pet. Tech, *Page 58-60,* Dec. 1960

Van Everdingen A.F and Hurst W. *The Application of the Laplace Transformation to Flow Problems in Reservoirs* [Ref. 3]
Petroleum Transactions, AIME, volume 186, *Page 305-324,* 1949

Chang, Y-B, Coats B. K., and Nolen, J. S. *A Compositional Model for CO2 Floods Including CO2 Solubility in Water.* [Ref. 4]
SPE 35164, Proc. Permian Basin Oil and Gas Recovery Conference, Midland, Texas, 1996

Kazemi, H. *Numerical Simulation of Water-Oil Flow in Naturally Fractured Reservoirs* [Ref. 5]
SPE 5719, SPEJ, *Page 317-326,* Dec 76

Dake, L.P. *Fundamentals of Reservoir Engineering* [Ref. 6]
Elsevier, *Page 256,* 1978

Reid, R. C., Prausnitz, J. M. and Sherwood, T. K. *The properties of Liquids and Gases* [Ref. 7]
McGraw-Hill, *Page 546,* 1977

Quandalle, P. and Sabathier, J. C. *Typical Features of a Multipurpose Reservoir Simulator* [Ref. 8]
SPE Reservoir Engineering, November 1989

Gilman, J. R. and Kazemi, H. *Improved Calculation for Viscous and Gravity Displacement in Matrix Blocks in Dual-Porosity Simulators* [Ref. 9]
SPE 16010, San Antonio, 1987

Martin, J. J. *Cubic Equations of State-Which?* [Ref. 10]
I and EC Fundamentals, Vol. 18, *Page 81,* May 1973

Coats, K. H. *Simulation of Gas Condensate Reservoir Performance* [Ref. 11]
SPE paper no. 10512, presented at the Sixth SPE Symposium on Reservoir Simulation, New Orleans, 1982

Lorentz, J., Bray, B. G. and Clark, C. R. J. *Calculating Viscosity of Reservoir Fluids from their Composition* [Ref. 12]
J.Pet.Tech.1171, *Page 231,* 1964

Michelsen, M. L. *The isothermal flash problem. Part I. Stability* [Ref. 13]
Fluid Phase Equilibria Vol 9, *Page 1-19,* 1982

Dennis, J. E. and Schnabel, R. B. *Numerical Methods for Unconstrained Optimisation and Nonlinear Equations* [Ref. 14]
Prentice-Hall, 1983

Crowe, C. M. and Nishio, M. *Convergence Promotion in the Simulation of Chemical Processes - The General Dominant Eigenvalue Method* [Ref. 15]
AIChEJ, Vol. 23, No. 3, *Page 528-529,* May 1975

Soriede, I., and Whitson, C.H. *Peng Robinson predictions for hydrocarbons, CO2, N2 and H2S with pure water* [Ref. 16]
Fluid Phase Equilibria 77, *Page 217-240,* 1992

Cook, R. E., Jacoby, R. H. and Ramesh, A. B. *A Beta-type Reservoir Simulator for Approximating Compositional Effects During Gas Injection* [Ref. 17]
SPE 4274, 1974

Aziz, K. and Settari, A. *Petroleum Reservoir Simulation* [Ref. 18]
Applied Science Publishers, London, *Page 398,* 1979

Killough, J. E. *Reservoir Simulation with History-dependent Saturation Functions* [Ref. 19]
Trans. AIME 261, *Page 37-48,* 1976

Carlson, F. M. *Simulation of Relative Permeability Hysteresis to the Non-Wetting Phase* [Ref. 20]
SPE 10157, San Antonio, 1981

Land, C. E. *Calculation of Imbibition Relative Permeability for Two- and Three-Phase Flow from Rock Properties* [Ref. 21]
Soc. Pet. Eng. J. *Page 149-156,* June 1968

Pedrosa, O. A., and Aziz, K. *Use of Hybrid Grid in Reservoir Simulation* [Ref. 22]
SPE 13507, 1985

Ewing, R. E. et al. *Efficient Use of Locally Refined Grids for Multiphase Reservoir Simulation* [Ref. 23]
SPE 18413, 1989

Todd, M. and Longstaff, W. *The Development, Testing and Application of a Numerical Simulator for Predicting Miscible Flood Performance* [Ref. 24]
SPE 3484, 1972

Zuber, N. and Findlay, J. A. *Average Volumetric Concentration in Two-Phase Flow Systems* [Ref. 25]
J. Heat Transfer, Trans. ASME, Ser. C, vol 87, *Page 453-468,* 1965

Aziz, K., Govier, G. W. and Fogarasi, M. *Pressure Drop in Wells Producing Oil and Gas* [Ref. 26]
J. Cdn. Pet. Tech. *Page 38-48,* Jul.-Sep. 1972

Ansari, A. M. et al. *A Comprehensive Mechanistic Model for Upward Two-Phase Flow in Wellbores* [Ref. 27]
SPE Production & Facilities, *Page 143-151,* May 1994

Hasan, A. R. and Kabir, C. S. *A Study of Multiphase Flow Behavior in Vertical Wells* [Ref. 28]
SPE Production Engineering, *Page 263-272,* May 1988

Shi, H., Holmes, J.A. et al. *Drift-Flux Modeling of Multiphase Flow in Wellbores* [Ref. 29]
SPE 84228, presented at the SPE Annual Technical Conference and Exhibition, Denver, Oct. 2003

Shi, H., Holmes, J.A. et al. *Drift-Flux Parameters for Three-Phase Steady-State Flow in Wellbores* [Ref. 30]
SPE 89836, presented at the SPE Annual Technical Conference and Exhibition, Houston, Sept. 2004

Harmathy, T. Z. *Velocity of Large Drops and Bubbles in Media of Infinite and Restricted Extent* [Ref. 31]
AIChEJ, Vol. 6, *Page 281,* 1960

Wallis, G. B. and Makkenchery, S. *The Hanging Film Phenomenon in Vertical Annular Two-phase Flow* [Ref. 32]
Trans. ASME, Ser. I, Vol. 96, *Page 297,* 1974

Wallis, G. B. *One-dimensional Two-phase Flow* [Ref. 33]
McGraw-Hill, 1969

Hasan, A. R. and Kabir, C. S. *A Simplified Model for Oil/Water Flow in Vertical and Deviated Wellbores* [Ref. 34]
SPE Production & Facilities, *Page 56-62,* Feb. 1999

Beggs, H. D. *Production Optimization Using Nodal Analysis* [Ref. 35]
OGCI Publications, Oil & Gas Consultants Inc., Tulsa, *Page 81,* 1991

Hagedorn, A. R. and Brown, K. E. *Experimental Study of Pressure Gradients Occurring During Continuous Two-Phase Flow in Small-Diameter Vertical Conduits* [Ref. 36]
J. Pet. Tech. *Page 475-484,* April 1965

Rozon, B. J. *A Generalized Finite Volume Discretization Method for Reservoir Simulation* [Ref. 37]
SPE 18414, Proceedings of the Reservoir Simulation Symposium, Houston, Texas, Feb. 1989

Ponting, D. K. *Corner Point Geometry in Reservoir Simulation* [Ref. 38]
Proceedings of the Joint IMA/SPE European Conference on the Mathematics of Oil Recovery, Cambridge, July 1989

Yanosik, J. L. and McCracken, T. A. *A Nine-Point, Finite-Difference Reservoir Simulator for Realistic Prediction of Adverse Mobility Ratio Displacements* [Ref. 39]
Trans. AIME, *Page 573-592,* Aug. 1979

Hegre, T. M., Dalen, V. and Henriquez, A. *Generalised Transmissibilities for Distorted Grids in Reservoir Simulation* [Ref. 40]
SPE 15622, Oct. 1986

Shiralkar, G. A., and Stephenson, R. E. *A General Formalism for Simulating Physical Dispersion and a New Nine-Point Scheme.* [Ref. 41]
SPE Reservoir Engineering, Feb. 1991

Stone, H. L. *Probability Model for Estimating Three-Phase Relative Permeability* [Ref. 42]
Trans AIME (JPT), 249, *Page 214-218,* 1970

Stone, H. L. *Estimation of Three-Phase Relative Permeability and Residual Oil Data,* [Ref. 43]
Can.Pet.Tech., Vol 12, *Page 53-61,* 1973

Vinsome, P. K. W. *Orthomin, an Iterative Method for Solving Sparse Banded Sets of Simultaneous Linear Equations* [Ref. 44]
SPE 5729, 4th SPE Symposium on Reservoir Simulation, Los Angeles, 1976

Appleyard, J. and Cheshire, I. M. *Nested Factorization* [Ref. 45]
SPE 12264, 7th SPE Symposium on Reservoir Simulation, San Francisco, 1983

Wallis, J. R. *Incomplete Gaussian Elimination as a Preconditioning for Generalised Conjugate Gradient Acceleration* [Ref. 46]
SPE 12265, 7th SPE Symposium on Reservoir Simulation, San Francisco, 1983

Holmes, J. A. *Enhancements to the Strongly Coupled, Fully Implicit Well Model: Wellbore Crossflow Modeling and Collective Well Control* [Ref. 47]
SPE 12259, 7th SPE Symposium on Reservoir Simulation, San Francisco, 1983

Ponting, D. K., et al. *An Efficient Fully Implicit Simulator* [Ref. 48]
Soc. Pet. Eng. J., 23, *Page 544,* 1983

Cheshire, I. M. and Pollard, R. *Recent Advances in Nested Factorization* [Ref. 49]
Proceedings of the Cambridge Conference on Mathematical Techniques for the Oil Industry, July 1987

Wallis, J. R., Kendall, R. P. and Little, T. E. *Constrained Residual Acceleration of Conjugate Residual Methods* [Ref. 50]
SPE 13536, 1985

Saad, Y. *Iterative Methods for Sparse Linear Systems* [Ref. 51]
PWS Publishing Company, Boston, 1996

Coats, K. H. *An Equation of State Compositional Model* [Ref. 52]
SPE 8284, SPEJ, *Page 363,* Oct. 1980

Bardon, C. and Longeron, D. G. *Influence of Very Low Interfacial Tensions on Relative Permeabilities* [Ref. 53]
SPE 7609, SPEJ, Oct. 1980

Coats, K. H. *In-Situ Combustion Model* [Ref. 54]
SPE Journal, *Page 533-554,* December 1980

Perry, R. H. and Green, D.W. *Perry's Chemical Engineers' Handbook* [Ref. 55]
McGraw Hill, Sixth Edition, 1984

Reid, R.C., Prausnitz, J. M. and Polling, B. E. *The Properties of Gases and Liquids* [Ref. 56]
McGraw-Hill, 1987

Prats, M. *Thermal Recovery* [Ref. 57]
SPE Monograph 7, *1*986

Crookston, H. B., Culham, W. E. and Chen, W. H. *Numerical Simulation Model for Thermal Recovery Processes* [Ref. 58]
Soc. Pet. Eng. J., Trans. AIME, 267, *Page 37-58,* Feb. 1979

Edmister, W. C. and Lee, B. I. *Applied Hydrocarbon Thermodynamics, Volume 1* [Ref. 59]
Gulf Publishing Company, Second Edition, 1984

Wingard, J. S. *Multicomponent Multiphase Flow in Porous Media with Temperature Variation* [Ref. 60]
PhD thesis, Department of Petroleum Engineering, Stanford University, 1988

Trangenstein, J. A. *Analysis of a Model and Sequential Numerical Method for Thermal Reservoir Simulation* [Ref. 61]
The Mathematics of Oil Recovery, Ed King, P.R., Clarendon Press, Oxford, 1992

Tortike, W. S. and Farouq Ali, S. M. *Saturated Steam Property Functional Correlations for Fully Implicit Thermal Reservoir Simulation* [Ref. 62]
SPEJ, 11, *Page 471-474,* 1989

Grabowski, J. W. and Rubin, B. *A Preliminary Numerical Simulation Study of In-situ Combustion in a Cold Lake Oil Sands Reservoir* [Ref. 63]
The Journal of Canadian Petroleum Technology (Montreal), *Page 79-89,* April-June, 1981

Ejiogu, G. C. and Fiori, M. *High-Pressure Saturated-Steam Correlations* [Ref. 64]
J. Pet. Tech. 39, 12, *Page 1585-1590,* 1987

Haywood, R. W. *Thermodynamic Tables in SI (Metric) Units* [Ref. 65]
Cambridge University Press, Third Edition, 1990

Farouq Ali, S. M. *Oil Recovery by Steam Injection* [Ref. 66]
Producers Publishing Co., Bradford, PA, 1970

Aziz, K., Ramesh, A. B. and Woo, P. T. *Fourth SPE Comparative Solution Project: Comparison of Steam Injection Simulators* [Ref. 67]
Journal of Petroleum Technology, *Page 1576-1584,* December 1987

Wilson, G. *A Modified Redlich-Kwong EOS, Application to General Physical Data Calculations* [Ref. 68]
Paper 15C, presented at the Annual AIChE National Meeting, (Cleveland, Ohio), 1968

Rubin, B. and Blunt, M. J. *Higher-Order Implicit Flux Limiting Schemes for Black Oil Simulation* [Ref. 69]
SPE 21222, 11th SPE Symposium on Reservoir Simulation, Anaheim, 1991

Kenyon, D. E. and Behie, A. *ThirdSPEComparativeSolutionProject:GasCyclingofRetrogradeCondensateReservoirs* [Ref. 70]
SPE 12278, 7th SPE Symposium on Reservoir Simulation, San Francisco, 1983

Haaland, S.E. *Journal of Fluid Engineering, volume 105* [Ref. 71]
Trans. ASME, 89, 1983

Welty, J.R., Wicks, C.E., and Wilson, R.E. *Fundamentals of Momentum Heat and Mass Transfer* [Ref. 72]
John Wiley and Sons, 1984

Nghiem, L. X., Collins D. A. and Sharma, R. *Seventh SPE Comparative Solution Project: Modeling of Horizontal Wells in Reservoir Simulation* [Ref. 73]
SPE 21221, 11th SPE Symposium on Reservoir Simulation, Anaheim, 1991

Whitson, C. H. and Fevang, Ø. *Generalised Pseudopressure Well Treatment in Reservoir Simulation* [Ref. 74]
IBC Conference on Optimisation of Gas Condensate Fields (Aberdeen), June 1997

Gunasekera, D., Childs, P., Cox, J. and Herring, J. *A Multi-Point Flux Discretization Scheme for General Polyhedral Grids* [Ref. 75]
SPE 48855, International Oil & Gas Conference and Exhibition, Beijing, China, 1998

Aavatsmark, I., Barkve, T. and Mannseth, T. *Control-Volume Discretization Methods for 3D Quadrilateral Grids in Inhomogeneous, Anisotropic Reservoirs* [Ref. 76]
SPE 38000, SPE Reservoir Simulation Symposium, Dallas, Texas, June 1997

Beattie, C.I., Boberg, T.C. and McNab, G.S. *Reservoir Simulation of Cyclic Steam Stimulation in the Cold Lake Oil Sands* [Ref. 77]
SPE 18752, SPE Reservoir Engineering, May 1991

Vinsome, P.K.W. and Westerveld, J. *A simple method for predicting cap and base rock heat losses in thermal reservoir simulators* [Ref. 78]
The Journal of Canadian Petroleum Technology, (Montreal), *Page 87-90,* July-September 1980

Mott, R. *Calculating Well Deliverability in Gas Condensate Reservoirs* [Ref. 79]
The Tenth European Symposium on Improved Oil Recovery, Brighton, UK, Aug. 1999

Leibovici, C.F., Barker, J.W. and Waché, D. *A Method for Delumping the Results of a Compositional Reservoir Simulation* [Ref. 80]
SPE 49068, SPE Annual Technical Conference & Exhibition, New Orleans, 1998

Pedersen, K. S., Fredenslund, Aa. and Thomassen, P. *Properties of Oils and Natural Gases* [Ref. 81]
Contributions in Petroleuam Geology and Engineering, Vol. 5, Gulf, Houston, 1989

Holmes, J.A., Barkve, T. and Lund, Ø. *Application of a Multisegment Well Model to Simulate Flow in Advanced Wells* [Ref. 82]
SPE 50646, SPE European Petroleum Conference, The Hague, October 1998

Bowers, B.E., Brownlee, R.F. and Schrenkel, P.J. *Development of a Downhole Oil/Water Separation and Reinjection System for Offshore Application* [Ref. 83]
SPE Prod. and Facilities, 15, *Page 115-122,* May 2000

Stone, T.W., Bennett, J., Law, D.H.-S. and Holmes, J.A. *Thermal Simulation with Multisegment Wells* [Ref. 84]
SPE 78131, SPE Reservoir Evaluation & Engineering, *Page 206-218,* June 2002

Hibbitt, Karlsson & Sorensen, Inc. *ABAQUS Theory Manual* [Ref. 85]
Hibbitt, Karlsson & Sorensen, Inc. 1998

Patillo, P.D., Kristiansen, T.G., Sund, G.V., and Kjelstadli, R.M. *Reservoir Compaction and Seafloor Subsidence at Valhall* [Ref. 86]
SPE/ISRM Eurock '98, Trondheim, Norway, 1998

Stone, T., Bowen, G., Papanastasiou, P., Fuller, J. *Fully Coupled Geomechanics in a Commercial Reservoir Simulator* [Ref. 87]
SPE European Petroleum Conference, Paris, France, 23-24 October, 2000

Coats, K.H. *IMPES Stability: The Stable Step* [Ref. 88]
SPE 69225, SPE Reservoir Simulation Symposium, Houston, Texas, 11-14 February 2001

Landau, L.D. and Lifshit, E. M. *Theory of Elasticity 3rd Edition* [Ref. 89]
Butterworth-Heinemann, Oxford, 1988

Mavko, G., Mukerji T. and Dvorkin J. *The Rock Physics Handbook: Tools for Seismic Analysis in Porous Media* [Ref. 90]
Cambridge University Press, 1998

Brie A., Pampuri F.,Marsala A.F. and Meazza O. *Shear Sonic Interpretation in Gas-Bearing Sands* [Ref. 91]
SPE 30595, SPE Annual Technical Conference & Exhibition, Dallas, October 1995

Han G., Dusseault, M. *Description of fluid flow around a wellbore with stress-dependent porosity and perm* [Ref. 92]
Journal of Petroleum Science and Engineering 1064, *Page 1-16,* 2003

Zimmerman, R.W. *Compressibility of Sandstones* [Ref. 93]
ELSEVIER, New York, 1991

Zakirov, I.S., Aanonsen, S.I., Zakirov, E.S. and Palatnik, B.M. *Optimizing Reservoir Performance by Automatic Allocation of Well Rates* [Ref. 94]
6th European Conference on the Mathematics of Oil Recovery, Leoben, September 1996

Gill,P.E., Murray, W. and Wright, M.H. *Practical Optimization* [Ref. 95]
Academic Press, 1986

Sokolnikoff, I.S. and Redheffer, R.M. *Mathematics of Physics and Modern Engineering* [Ref. 96]
McGraw-Hill, 1967

Stone, T.W., Xian, C., Fang, Z., Manalac, E., Marsden, R., Fuller, J., *Coupled Geomechanical Simulation of Stress Dependent Reservoirs,* [Ref. 97]
SPE 79697, SPE Reservoir Simulation Symposium, Houston, Texas, USA, 3-5 February, 2003

Stüben, K. *SAMG USER's Manual Release 21b1* [Ref. 98]
Fraunhofer Institute SCAI, Schloss Birlinghoven, D-53754 St. Augustin, Germany, July 2002

Whitson, C.H., Fevang, O. and Saevareid, A. *Gas Condensate Relative Permeability for Well Calculations* [Ref. 99]
SPE 56476, SPE Annual Technical Conference, Houston, Texas, USA, October 1999

van Lingen, P., Sengul, M., Daniel, J. and Cosentino, L. *Single Medium Simulation of Reservoirs with Conductive Faults and Fractures* [Ref. 100]
SPE 68165, SPE Middle East Oil Show, Bahrain, March 2001

Abbott, M.M. and van Ness, H.G. *Thermodynamics with chemical applications* [Ref. 101]
Schaum's Outline Series, McGraw-Hill, 1989

Barker, J.W. and Fayers F.J. *Transport Coefficients for Compositional Simulation With Coarse Grids in Heterogeneous Media* [Ref. 102]
SPE 22591, October 1991

Arri, L.E., Yee, D., Morgan, W.D. and Jeansonne, N.W *Modeling Coalbed Methane Production with Binary Gas Sorption* [Ref. 103]
SPE 24363, May 1992

Pal, R. and Rhodes, E. *Viscosity/Concentration Relationships for Emulsions* [Ref. 104]
Journal of Rheology, 33, (7), *Page 1021-1045,* 1989

Valle, A. *Three Phase Gas-Oil-Water Pipe Flow* [Ref. 105]
Ph.D. Thesis, Imperial College, London, 2000

Liu, Q., Stone, T., Han, G., Marsden, R., Shaw, G. *Coupled Stress and Fluid Flow Using a Finite Element Method in a Commercial Reservoir Simulator* [Ref. 106]
SPE 88616 SPE Asia Pacific Oil and Gas Conference, October 2004

Shaw, G., Stone, T. *FiniteVolumeMethodsforCoupledStress/FluidFlowinaCommercialReservoirSimulator* [Ref. 107]
SPE 93430 SPE Reservoir Simulation Symposium, January/February 2005

Charbeneau, Randall J. *Groundwater Hydraulics and Pollutant Transport* [Ref. 108]
Prentice Hall, 2000

Palmer, I., Mansoori, J. *How Permeability Depends on Stress and Pore Pressure in Coalbeds: A New Model* [Ref. 109]
SPE 52607 SPE Reservoir Evaluation & Engineering, December 1998

Spycher, N. and Pruess, K. *CO2-H2O mixtures in the geological sequestration of CO2.II. Partitioning in chloride brines at 12-100 C and up to 600 bar* [Ref. 110]
Geochimica et Cosmochimica Acta, Vol. 69, No. 13, *Page 3309-3320,* 22005

Zaytsev, I.D., and Aseyev, G.G. *Properties of Aqueous Solutions of Electrolytes,* [Ref. 111]
CRC Press, 1993

Kell, G.S and Whalley, E. *Reanalysis of the density of liquid water in the range 0-150 C and 0-1 kbar* [Ref. 112]
J. Chem. Phys., Vol 62, No. 9, May 1975

Vesovic, V. Wakeham, W. A., Olchowy, G.A., Sengers, J.V., Watson, J.T.R., Millat, J.
The Transport Properties of Carbon Dioxide [Ref. 113]
J. Phys. Chem. Ref. Data, Vol. 19, No.3, 1990

Fenghour, A., Wakeham, W.A., Vesovic, V.
The Viscosity of Carbon Dioxide [Ref. 114]
J. Phys. Chem. Ref. Data, Vol. 27, No.1, 1999

Weinstein, H.G., Chappelear, J.E., Nolen, J.S.
Second Comparative Solution Project: A Three-Phase Coning Study [Ref. 115]
SPE 10489 and Journal of Petroleum Technology, *Page 345-353,* March 1986

Potter, Babcock and Brown
A new method for determining the solubility of salt in aqueous solutions at elevated temperatures [Ref. 116]
Research U.S. Geol. Surv. 5, no. 3, *Page 389-395,* 1977

de Montleau, P., Cominelli, A., Neylon, K., Rowan, D., Pallister, I., Tesaker, O. Nygard I.
Production Optimization under Constraints Using Adjoint Gradients [Ref. 117]
10th European Conference on the Mathematics of Oil Recovery , 4-7 September 2006

Henderson, G.D., Danesh, A., Tehrani, D.H., Al-Kharusi, B.
The Relative Significance of Positive Coupling and Inertial Effects on Gas Condensate Relative Permeabilities at High Velocity [Ref. 118]
SPE 62933, October 2000

Mathew J. Mavor, William D. Gunter
Secondary Porosity and Permeability of Coal vs. Gas Composition and Pressure [Ref. 119]
SPE 90255, January 2006

Hustad, O.S., Hansen, A.G.
A Consistent Formulation for Three-Phase Relative Permeabilities and Phase Pressures based on Three Sets of Two-Phase Data [Ref. 120]
RUTH: A Norwegian Research Program on Improved Oil Recovery, Program Summary, Norwegian Petroleum Directorate, *Page 183-194,* Stavanger 1996
(see also **Proceedings of the Eighth European Symposium on Improved Oil Recovery**, Vienna, Austria, May 1995)

Hustad, O.S., Kløv, T., Lerdhahl, T. R., Berge, L.,I., Stensen, J., Å., Øren, P-E
Gas Segregation During WAG Injection and the Importance of Parameter Scaling in Three-Phase Models [Ref. 121]
SPE /DOE Improved Oil Recovery Symposium, Tulsa, Oklahoma, April 2002, SPE 75138

Hustad, O.S.
A Coupled Model for Three-Phase Capillary Pressure and Relative Permeability [Ref. 122]
SPE Journal, *Page 59-69 ,* March 2002, SPE 74705.

Baker, L. E.
Three-Phase Relative Permeability Corrections [Ref. 123]
SPE/DOE 17369, April 1988

Hustad, O. S. and Holt, T.
Gravity Stable Displacement of Oil by Hydrocarbon Gas After Waterflooding [Ref. 124]
SPE/DOE Eighth Symposium on Enhanced Oil Recovery, Tulsa, Oklahoma, April 1992, SPE 24116.

Kovscek, A.R. *Reservoir Simulation of Foam Displacement Processes* [Ref. 125]
Seventh UNITAR International Conference on Heavy Crudes and Tar Sands, Beijing, China, October 1998.

Cheng, L., Reme, A.B., Shan. D., Coombe, D.A and Rossen, W.R. *Simulating Foam Processes and High and Low Foam Qualities* [Ref. 126]
SPE/DOE Symposium on Improved Oil Recovery, Tulsa, Oklahoma, April 2000, SPE 59287.

Whitson, C. H. and Fevang, Ø. *Modeling Gas Condensate Well Deliverability* [Ref. 127]
SPE Annual Technical Conference and Exhibition, Dallas, Texas, October 1995, SPE 30714.

Schlumberger *VISAGE Manual* [Ref. 128]
Schlumberger, 2008

Singh, K. and Whitson, C.H. *Gas Condensate Pseudopressure in Layered Reservoirs* [Ref. 129]
SPE Abu Dhabi International Petroleum Exhibition and Conference, Abu Dhabi, UAE, November 2008, SPE 117930

StatoilHydro ASA *MethodandApparatusforDeterminingCapillaryPressuresinaThreePhaseFluidReservoir* [Ref. 130]
United States Patent Application No. 12/266307, in the name of StatoilHydro ASA

Hustad, O. S. and Browning, D. J. *A Fully Coupled Three-Phase Model for Capillary Pressure and Relative Permeability for Implicit Compositional Reservoir Simulation* [Ref. 131]
SPE/EAGE Reservoir Characterization and Simulation Conference, Abu Dhabi, UAE, October 2009, SPE 125429

Spycher, N. and Pruess, K. *A Phase-Partitioning Model for CO2-Brine Mixtures at Elevated Temperatures and Pressures: Application to CO2-Enhanced Geothermal Systems* [Ref. 132]
Transp. Porous Med. Springer, 2July 2009

Index

Appendix I

A

Adaptive Implicit Method . 26

Adjoint Gradient Screening Option49

Adsorption
- and COALPP. 168
- and environmental tracer274 to 275
- Competitive, CO2 into coal167
- Foam 321
- Gas re-adsorption into coal163
- on coal surface. 162
- Polymer 781
- Surfactant 993

AIM Method. 347

Alkaline flooding 57

Alkaline Model. 57

ALQ
- in VFP table 687

ALQ Values
- Changing 511

Amalgamation
- Cartesian LGRs 567, 571
- Cell 563

API Tracking 30, 69

Aqueous Phase 143, 401

Aquifer 35
- Flux Boundary Conditions313

Aquifer Modeling 73
- Carter-Tracy 76
- Constant Flux 81
- Fetkovich. 79
- Numerical 74

Artificial Lift. 511, 1230

Artificial Lift Quantity
- and gas injection. . . . 386
- in network branch. . . 394
- in network flow lines. 684
- in outlet branch704 to 705
- in VFP table. 687
- in VFP tables 392
- Multi-level Compression695
- Optimization 392

Automatic
- Chokes 695, 701
- Compressors367, 369, 372, 694, 700
- Refinement 594
  - Restrictions. 596
- Workover 581, 667, 1171

Automatic Opening 507, 1214

B

Balancing, Network 687

Block Centre Geometry . . . 25

Brine 39

Brine Tracking. . 30, 103, 126

C

Calorific Value
- Control . 418 to 419, 426
- Gas 417, 425

Calorific Values 92, 411

Capillary
- Pressure 992

Carbon Dioxide Solution143, 401, 621

Cell Amalgamation. 563

Chemical Reactions
- Examples 157
- Formulation 150
- Keywords 148
- Output 156

Chokes
- Automatic. 695, 701

Closing Wells. 514, 1211

Coal Bed Methane . . .42, 161
- Compositional Effects170
- Dual Porosity 162
- Example
  - Basic Test 176
  - CO_2 Injection. 180
- Examples 176
- Keywords 173, 188
- Using ECLIPSE 100 169
- Using ECLIPSE 300 191

COALADS 167

Combustion 1068

Co-mingling. 34

Compatibility 195
Compositional Effects
Coalbed 167
Compressed Vertical Equilibrium1151
Compressors 40
Activating 513
Automatic367, 369, 372, 694, 700
Network 684, 687
Connection
Factor 1189
Non-Neighbor. 27
Opening 507, 1214
Well . . 1187, 1207, 1212
Constant Flux Aquifers 81
Construction, Network . . .686
Control
Injection. 33
Output1247, 1251, 1257, 1263, 1275
Production 32
Sales Gas 33
Well. 31
Convergence Criteria 350
Convergence Reports 201
Corner Point Geometry 25
Crossflow . . . 34, 1208, 1212

D

Data Requirements 544
DCQ 40
Calculating 365
Decay . . . . . 274 to 275
Delivery Capacity
Estimation 366
Delumping. 1129
Density
3-component. 965
4-component. 964
D-Factor 1192
Diagonal Transmissibilities707
Diffusion 211
Coefficient 162
Coefficients for ECLIPSE 100215
Coefficients for ECLIPSE 300220
Methane from Coal162, 167
Miscible Flow. 970
with Environmental Tracer option 274 to 275
Diffusivity 162
Diffusivity Calculations . . 227
Directional Relative Permeability29
Downtime 1214
Drainage Region 313
Drilling Queue506, 514, 1217
Drilling Wells506, 1214, 1217, 1228
Dual Porosity . . 28, 233, 574
Coal Bed Methane . . 162
Discretized matrix model263, 266
Drainage process Sigma factor258
Gravity drainage238 to 239, 241, 256
Integrated capillary pressure260
Keyword summary. . 249
Linear equation solution254
Modifying recovery versus time252
Recovery Mechanisms238
Relative permeability scaling253
Restrictions 248
Sigma factor for drainage258
Specific Facilities. . . 246
Transfer functions . . 251
Transmissibility Calculations234, 236

E

ECLIPSE Features 25
Economic Limit . . . 427, 581
Efficiency Factor. 1214
End Point Scaling
LGRs 575
Energy Control 418, 423, 427
Environmental Tracers 41, 269
Example 280
Keywords 276
Partitioned Tracers . . 271
Restrictions . . . 275, 327
Using 274
Equations of State 287
Hydrocarbon Mixtures294
Three Parameter. . . . 293
Two parameter. 287
Equilibration170 to 171, 192, 543
Algorithm 547
Initial conditions . . . 548
Extended Network 693

F

Fetkovich Aquifers. 79
Figure 950
File Handling. 303
File internal format . . 305
Introduction 303
Fine Grid Equilibration. . . . 35
Flash Calculations 290
FLEXlm licenses 827
Flowlines 815
Fluid In Place
Accurate Calculation. 549
Reports 1124
Flux Boundary Conditions39, 309
Keywords 317
LGRs. 315
Limitations. 316
No-flow Boundary Conditions315
Pressure Boundary Conditions314
Using. 310
Wells 313
Flux Region
Multiple. 312
Single 310
Foam 319
Adsorption 321
Conservation Equation320
Decay 321
Example 332
Keywords 328
Simulation 320
Using. 326
Foam Model 42
Foamy oil 1068
Formulation of Equations . 341
Convergence Criteria 350
Flows 352
IMPES and AIM Methods347
Newton Iteration. . . . 357
Reservoir Densities. . 356
Variable Sets 343
Free Format Input 25
Frictional Pressure Loss . 1163
Fully Implicit Technology. . 26

G

Gas
- Calorific Value. 417, 425
- Calorific Value Control 42
- Energy Control418, 423, 427
- Field Operations 40
- Mobility Reduction . 321
- Rate
  - Control 365
  - Output 365
  - Sales 365
- Supply Contracts . . . 370
- Well 34

Gas Accounting Tables . 1223

Gas Calorific Value Control417
- Adjusting Guide Rates419
- Controlling Energy Production Rate 423
- Restrictions 429
- Summary of Keywords430
- Using the Facility. . . 425

Gas Field Operations 359
- Calorific Value Control427
- Description of Facilities361
- Energy Control 427, 430
- Introduction. 359
- Keywords 380
- Multiple Contract Groups370
- Network 374
- Reservoir Coupling . 825
- Restrictions 379
- Using 373

Gas Lift
- Network 687

Gas Lift Optimization 41, 385
- Group 388
- Individual Well 386
- Keywords 396
- Network 389, 394
- Restrictions 395
- Using 392
- VFP Tables 392

Gas Mode
- Single 169
- Two 170

Gas Plants 397

GASWAT 405

Geomechanics. 433

Geometry
- Block Centre 25
- Corner Point. 25
- Options 25

GI Pseudo-compositional . 449
- Example 456
- Keywords 454
- SUMMARY. 455
- Using. 453

GI Pseudo-Compositional Model41

GLIFTOPT 393

Gradient Option. . . . 463, 727
- Data File Overview. . 468
- Example . 494, 739, 747
- Introduction . . . 463, 727
- Keywords 487, 745
- Output 473
- Technical Description 482

Graphics 1259

Grid
- Reports 573

Grid Orientation Effect. . . 707

Group
- Control 1216, 1221
- Economic Limit 427
- Economic Limits . . 1221
- Efficiency factor. . . 1214
- Hierarchy. 33, 1216
- Injection Control. 33
- Production Control . . . 32
- Production Rules . . . 393

Group Production Rules
- Activating compressors513
- Changing VFP table . 510
- Changing well ALQ . 511
- Closing wells during operations514
- Drilling new wells . . 506
- Opening new connections507
- Reducing THP limit . 509
- Use with Network Option515

Groups
- Controls. . 375, 581, 815
  - Calorific Value . . 418 to 419, 426
  - Energy 418, 423, 427
  - Network 683
- Multiple Contract . . . 372
  - Defining 370
  - Simulating 370
- Production Rate Limits393
- Sub-sea Completion Manifolds683, 689

Guide Rates
- Adjusting 419
- Calculating 426

H

Hierarchy. 33

Highly Deviated Well 34

Horizontal Well34, 40, 568, 635, 1163, 1172, 1189, 1204

Hydrocarbon Mixtures
- Equations Of State. . 294

Hydrostatic Equilibration
- Basic 547

Hysteresis 29, 517
- Capillary pressure . . 526
- Modeling oil wet systems525
- Non-wetting phase. . 519
- Using the option . . . 518
- Wetting phase 523

Hysteresis in WAG Floods 529
- Non-wetting phase. . 531
- Residual oil modification534
- Using the option . . . 530
- Wetting phase model 535

I

IMPES Method. .26, 341, 347

Independent Reservoir Regions537
- Examples 540
- Merging multiple corner point grids 539

Initial Conditions 543

Injection
- Control. 33
- Network 684
- Solvent. 172, 969

In-place LGRs 580

J

Jacobian matrix . . . . 26 to 27

K

Keywords
Network 704

L

Langmuir Isotherm . . 162, 167
Lift
Applying 511
LIFTOPT. 392
Local Grid Coarsening563, 590
Implementation. 590
Keywords. 597
Upscaling 591
Local Grid Refinement39, 563
Automatic 594
Block Sizes. 568
Cartesian 567
Editing 573
End Point Scaling . . .575
Example. 601
Flux Boundary Conditions315
Grid Data 584
Keywords. 597
Pore Volume 584
Radial 564, 585
Restrictions . . .580, 582
Rock Properties. 570
Switching Off 576
Switching On 576
Transmissibility 584
Unstructured. 575
Well 566, 581
Lumping 1129

M

Manifold Groups
Network 683, 689
Manifold Pressure 39
Master Reservoir . . .812, 819
Maximum Decrease Factors209
Minimum Step Codes 209
Miscible 31
Components 957
Flow
Numerical Diffusion 970
Miscible Flood Modeling. 609
Component density . 614
Control with pressure 615
Effect of water saturation611
Numerical diffusion . 617
Relative permeability 612
Todd-Longstaff Mixing Parameter 610
Using 619
Viscosity 613
Mixing Parameter . . 957, 965
Multi Porosity. 627
Multi-lateral Well635, 637, 1163, 1166, 1175
Multi-segment Wells43, 45, 635
Converting Friction Wells672
Down-hole separators 657
Drift Flux Slip Model 648
Introduction. 635
Keywords 436, 444, 674
Restrictions . . . 443, 670
The Multi-segment Well Model637
Thermal 662
Using 442, 665
VFP Tables 645

N

Nested Factorization 26
Network 681
Balancing 687
Chokes 695, 701
Compressors684, 687, 694, 700
Construction 686
ECLIPSE 300 702
Efficiency Factors . . 698
Extended. 693
Gas Field Operations 374
Gas Lift. 687
Gas Lift Optimization 394
Group Production Control683
Injection 684
Keywords 704
Pumps. 684, 687
Reservoir Coupling . 815
Standard 686
Structure 693
Using 686, 698
NGLs. 397
Nine-point Schemes 707
NNC. 27
Non-Darcy Flow 711
Non-Neighbor Connection . 27
Numbering Convention . . 586
Numerical Aquifers 74

O

Oil Saturation Properties. . 879
Opening New Connections 507
Output Control1247, 1251, 1257, 1263, 1275

P

Parallel Option. 749
Activating 750
Efficiency 752
Keywords 760
LGRs. 756
Memory Scaling 759
Partitioning 751
Performance. . . 752, 757
Pedersen Method, use of. . 291
Permeability
Reductions 781
Petro-elastic Model 763
Phase Options 25
Phase States. 296
Pipeline. 39
Polymer Adsorption 781
Polymer Flood. 777
Example
Problem 1 789
Problem 2 793
Fluid Viscosity 780
Keywords 785
Polymer Adsorption . 781
Processes 777
Simulation 778
Using. 783
Polymer Flood Model. 38
Pore Volume
Dead 781
LGRs. 584

Potential Calculations . . . 799
Potentials
Initial Contact Corrected800
Well506, 802, 1217 to 1218
Pressure
Boundary Conditions 314
Capillary 992
Dependence. 966
Frictional Loss. . . . 1163
Pressure Maintenance . . 1227
Production
Control . . . . . 32 to 33
Production Optimization728, 731
Productivity Index 1204, 1209
Pump Performance 39
Pumps
Network 684, 687
PVT
Tables 28

R

Recovery Mechanisms238, 805
Details of calculation 810
Introduction. 805
Solution gas drive . . 807
Summary output . . . 809
Water drive 808
Recovery Plant 399
Reducing The THP Limit. 509
Relative Permeability . . . 959
Directional 29
Near Critical Gas . . . 901
Near Critical Oil . . . 901
Surfactant 991
Velocity Dependent . 716
Relative permeability
Miscible flood modeling612
Re-perforating. 507
Reports
Grid 573
Timing 574, 752
Reservoir Coupling . . 42, 811
Files 816, 828
Gas Field Operations 825
Keywords 833
Licenses 827
Restrictions 818
Surface Networks . . . 815
Synchronization 813
Using. 819
Reservoir Optimization. . . 736
Restart 1269, 1279
Restart Runs 314
Restarts. 835
ECLIPSE 100. 836
ECLIPSE 300. 841
Restrictions
Automatic Refinement596
Environmental Tracers275, 327
Flux Boundary Conditions316
Gas Calorific Value Control429
Gas Field Operations. 379
Gas Lift Optimization 395
Local Grid Refinement580, 582
Multi-segment Wells443, 670
Reservoir Coupling . . 818
Retubing 510
Rivers 845
Introduction 845
Rock
Compaction 29
Data 28
Rock Compaction 859
Tabulated compaction curves860
Water induced compaction864
RPTGRID 1251
RPTPROPS 1247
RPTRST 1257
RPTSCHED 1263
RPTSOL 1275
Run-time dimensioning . . . 27

S

Sales Gas 1226
Sales Gas Control 33
Sales Gas Rate
Control 365
Output 365
Salinity 94, 412
Saturation
Table Scaling 29
Saturation Functions. 869
Consistency requirements899
Gas saturation properties875
Near Critical Relative Permeabilities 901
Oil saturation properties877
Table end points . . . 898
Three phase oil relative permeability models . . . 879
Water saturation properties873
Saturation Table Scaling . 905
Consistency requirements918
Example 919
Miscellaneous points .911
Scaling of capillary pressure functions. 907
Scaling of relative permeability functions . 908
Special applications . 913
Scale Deposition. 1209
Seasonality Profile Generation361
Separators
Down-hole 657
Shale gas 921
Shared Memory Parallel . . 41
Shear
Thinning. 781
Single Gas Mode 169
Skin Factor. 1192
Slave Reservoir 812, 821
Solid phase. 1042
Introduction 925
Solution Change Target . . 208
Solution of the Linear Equations937
Convergence Criteria 944
GMRES Method . . . 953
Initialization 941
Material balance . . . 938
Nested Factorization 939
Orthomin 943
Solution procedure. . 942
Solution Procedure for LGRs952
Treatment of wells . . 946
Solvent 955
Applications 955
Example 975
Initiating. 967
Injection 172, 969
Keywords 972
Numerical Diffusion 970
Pressure Dependence 966
PVT 963, 967
Relative Permeability959, 968

Todd and Longstaff . .957
Using 967
Solvent Model 40
Standard Network. 686
Surface Networks 39
Surface Tensions
Effects 983
Evaluation 983
Use 984
Use of MISCNUM . .987
Surfactant 989
Adsorption 993
Capillary Pressure . . .992
Conservation Equation990
Example. 1000
Keywords. 61, 997
Relative Permeability 991
Simulation 990
Using 995
Surfactant Model 41
Swing Factor 362

T

Temperature. 30
Template Matching. 1232
Tensor Permeability 1015
Discretization 1017
Introduction 1015
Thermal Option 43, 1023
Converting Data . . .1083
Features 1072
Formulation 1037
Keywords. 1025
Properties 1045
Selected bibliography1090
THP Limit
Reducing 509
Throughput Target 208
Time Truncation Error . . .208
Timesteps 579
Timing Reports574, 752, 1093
Todd and Longstaff. . .40, 957
Todd-Longstaff Mixing Parameter610
Total Compressibility Checks1099
Tracer
Tracking. 30
Tracer Tracking. 1117
Control of numerical diffusion1119
Delumping 1129
Introduction. 1117
Tracking original fluids1123
Using the facility . . 1121
Tracers
Environmental 269
Gas Calorific Value . 425
Tracking
API. 30, 69
Brine 30, 103, 126
Tracer 30, 1117
Tracer Concentration 425
Tracer Name 425
Transmissibility
Calculations 577
LGRs 584
Transmissibility Calculations1103
Block Center 1105
Cartesian. 1105
Corner Point 1108
Dual Porosity. . 234, 236
HALFTRAN 1109
OLDTRANR. 1106
Radial 1111
Variations 1114
Transport coefficients354 to 355
Tuning 573
Two Gas Mode 170

U

Units 411, 1133
Conventions 1133
Conversion factors . 1136
Unstructured LGRs 575

V

Velocity Dependent Relative Permeabilities. . 716
Vertical Equilibrium28, 574, 1139
Cell saturation distribution1142
Compressed. 1151
Miscellaneous points1149
Relative permeabilities and capillary pressures. 1146
Residual flow model 1147
Using the VE option 1141
Vertical Flow Performance1230, 1232
VFP Table Numbers
Changing 510
VFP Tables . . 71, 1230, 1232
Viscosity
3-phase 964
4-phase 963
Fluid 780
Viscosity Evaluation. 291

W

Water Induced Compaction864
Weighted Incremental Gradient 387 to 388
Well
Closing 514, 1211
Co-mingling. 34
Completion . . 507, 1212
Coning. 1213
Connection507, 1207, 1212, 1214
Control 31, 1210
Crossflow34, 1208, 1212
Drilling506, 514, 1214, 1217, 1228
Economic Limit . . . 1211
Efficiency factor. . . 1214
Flux Boundary Conditions313
Gas 34
Highly Deviated 34
Horizontal34, 40, 568, 635, 1163, 1172, 1189, 1204
Limit 1210
Multi-lateral635, 637, 1163, 1166, 1175
Multi-segment . 635, 637
Opening. 1214
Potential506, 802, 1217 to 1218
Productivity Index . 1204
Treatment in Local Grids581
Workover507, 514, 1211 to 1212, 1218, 1221, 1228
Well Inflow Performance
Connection transmissibility factor 1189
Flow-dependent skin factor1192
Injecting connections1201
Phase mobilities . . . 1194
Productivity index. . 1204

Special inflow equations 1195
Well bore head term 1203
Well Modeling Facilities 1207
Group and Field Controls 1216
Vertical Flow Performance 1230, 1232
Well Completions . 1207
Well Controls and Limits 1210
Wellbore
Friction 40
Wellbore Friction . . 672, 1163
Example 1179
Graphical Presentation 1172
Keywords 1178
Multi-lateral Wells . 1166
Using. 1166
Workovers 1171
Wells
Multi-segment. . . .43, 45
Potentials 802
Wildcard Characters . . . 1232
WLIFTOPT 392
Workover 507, 514, 1211 to 1212, 1218, 1221, 1228
Automatic 581, 667, 1171