Schlumberger

ECLIPSE

Technical Description

2004A

SCHLUMBERGER INFORMATION SOLUTIONS

Proprietary notice

Copyright © 1982 - 2004 Schlumberger. All rights reserved.

No part of the ECLIPSE Technical Description may be reproduced, stored in an information retrieval system, or translated or retransmitted in any form or by any means, electronic or mechanical, including photocopying and recording, without the prior written permission of the copyright owner.

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 limitation any warranties of merchantability or fitness for a particular purpose.

Patent information

Schlumberger ECLIPSE reservoir simulation software is protected by US Patents 6,018,497, 6,078,869 and 6,106,561, and UK Patents GB 2,326,747 B and GB 2,336,008 B. Patents pending.

Service mark information

The following are all service marks of Schlumberger:

The Calculator, Charisma, ConPac, ECLIPSE 100, ECLIPSE 200, ECLIPSE 300, ECLIPSE 500, ECLIPSE Office, EDIT, Extract, Fill, Finder, FloGeo, FloGrid, FloViz, FrontSim, GeoFrame, GRAF, GRID, GridSim, NWM, Open-ECLIPSE, PetraGrid, PlanOpt, Pseudo, PVT*i*, RTView, SCAL, Schedule, SimOpt, VFP*i*, Weltest 200.

Trademark information

Silicon Graphics and IRIX are registered trademarks of Silicon Graphics, Inc. OpenGL® and the oval logo are trademarks or registered trademarks of Silicon Graphics, Inc. in the United States and/or other countries worldwide. OpenInventor and WebSpace are trademarks of Silicon Graphics, Inc. IBM, AIX and LoadLeveler are registered trademarks of International Business Machines Corporation. Sun, SPARC, Solaris, Ultra and UltraSPARC are trademarks or registered trademarks of Sun Microsystems, Inc. Macintosh is a registered trademark of Apple Computer, Inc. UNIX is a registered trademark of UNIX System Laboratories. Motif is a registered trademark of the Open Software Foundation, Inc. The X Window System and X11 are registered trademarks of the Massachusetts Institute of Technology. PostScript and Encapsulated PostScript are registered trademarks of Adobe Systems, Inc. OpenWorks and VIP are registered trademarks of Landmark Graphics Corporation. Lotus, 1-2-3 and Symphony are registered trademarks of Lotus Development Corporation. Microsoft, Windows, Windows NT, Windows 95, Windows 98, Windows 2000, Windows XP, Internet Explorer, Intellimouse, Excel, Word and PowerPoint are either registered trademarks or trademarks of Microsoft Corporation in the United States and/or other countries. Netscape is a registered trademark of Netscape Communications Corporation. AVS is a registered trademark of AVS Inc. ZEH is a registered trademark of ZEH Graphics Systems. Ghostscript and GSview are Copyright of Aladdin Enterprises, CA. GNU Ghostscript is Copyright of the Free Software Foundation, Inc. Linux is Copyright of the Free Software Foundation, Inc. IRAP is Copyright of Roxar Technologies. LSF is a registered trademark of Platform Computing Corporation, Canada. VISAGE is a registered trademark of VIPS Ltd. Cosmo is a trademark and PLATINUM technology is a registered trademark of PLATINUM technology, inc. PEBI is a trademark of Veritas DGC Inc./HOT Engineering GmbH. Stratamodel is a trademark of Landmark Graphics Corporation. GLOBE*trotter*, FLEX*lm* and SAM*report* are registered trademarks of GLOBEtrotter Software, Inc. CrystalEyes is a trademark of StereoGraphics Corporation. Tektronix is a registered trade mark of Tektronix, Inc. GOCAD and JACTA are trademarks of T-Surf. Myrinet is a trade name of Myricom, Inc. This product may include software developed by the Apache Software Foundation (http://www.apache.org). Copyright (c) 1999-2001 The Apache Software Foundation. All rights reserved. MPI/Pro is a registered trademark of MPI Software Technology, Inc. The TGS logo is a trademark of TGS, Inc.

Table of Contents, ECLIPSE Technical Description

Table of Contents, ECLIPSE Technical Description 3
List of Figures 12
List of Tables 17

Chapter 1 - Introduction 23
Overview 23
ECLIPSE features 25
Ancillary programs 37
Additional programs 39
Licenses 50

Chapter 2 - API Tracking 51

Chapter 3 - Aquifer Modeling Facilities 55
Introduction 55
Numerical aquifer 56
Carter-Tracy aquifer 58
Fetkovich aquifer (ECLIPSE 100) 61
Constant flux aquifer (ECLIPSE 100) 64

Chapter 4 - Brine Tracking 65

Chapter 5 - Carbon Dioxide Solution in the Aqueous Phase 67

Chapter 6 - Chemical Reactions 69
Introduction 69
Chemical reaction keywords 70
Formulation 72
Output 79
Examples 80

Chapter 7 - The Coal Bed Methane Model 83
Introduction 83
Modeling compositional effects 86
Using the coal bed methane model 88
Summary of keywords 92
Examples 94

Chapter 8 - Compatibility Between ECLIPSE 100 Special Options 107

Chapter 9 - Convergence Reports 113

Chapter 10 - Diffusion 123
Introduction 123
Keywords 124
Diffusion coefficients: ECLIPSE 100 125
Diffusion coefficients: ECLIPSE 300 128
The cross phase diffusion option (ECLIPSE 100) 132
Diffusivity calculations 133

Using the molecular diffusion option 138

Chapter 11 - Dual Porosity 139

Introduction 139
Transmissibility calculations 140
Recovery mechanisms 142
Facilities specific to dual porosity runs 150
Restrictions on dual porosity runs 152
Keyword summary 153
Transfer functions 155
Modifying the recovery versus time response (ECLIPSE 100) 156
Scaling the fracture relative permeability (ECLIPSE 100) 157
Solution of the linear equations 158
Calculation of fractional volumes for gravity drainage 160
Variation of the sigma factor with drainage process 162
Integrated capillary pressure option 163
Discretized matrix model (ECLIPSE 100) 166

Chapter 12 - Environmental Tracers 169

Introduction 169
Using the environmental tracer option 171
Partitioned tracers 173
Summary of keywords 174
Example problem 176

Chapter 13 - Equations of State 185

Two-parameter equations of state 185
Flash calculations 189
Viscosity evaluation 190
Three parameter equations of state 192
Use of the equation of state for hydrocarbon mixtures 193
Calculation of phase states 195

Chapter 14 - Equations of State Regions 197

Introduction 197
Reservoir equation of state 198
Surface equation of state 200

Chapter 15 - File Handling in ECLIPSE 201

Introduction 201
File internal format 203
GRAF files 205

Chapter 16 - Flux Boundary Conditions 207

Introduction 207
Using flux boundary conditions 208
Summary of keywords 215

Chapter 17 - The Foam Model 216

Introduction 216
The simulation model 217
Using the foam model 220
Summary of keywords 221
Example data set 223

Chapter 18 - Formulation of the Equations 232
Introduction 232
Description of equations 233
Variable sets 234
The IMPES and AIM methods 238
Instability in IMPES 240
Convergence criteria for residual equations 241
Flows 244
Reservoir densities 248
Newton iteration of the non-linear residual 249

Chapter 19 - Gas Field Operations Model 251
Introduction 251
Description of the facilities 253
Applying separate gas supply contracts to individual groups 263
Using the gas field operations model 266
Summary of keywords 273

Chapter 20 - Gas Lift Optimization 276
Introduction 276
Using the gas lift optimization facility 282
Summary of keywords 286

Chapter 21 - Gas Plants and NGLs 287
Gas plant tables 287
The recovery plant option 289

Chapter 22 - The GASWAT Option 291
Introduction 291
Equation of state modifications 292
Calorific values 296
Salinity 297
Relative permeability tables 298
Initialization 299
Summary items 300

Chapter 23 - Gas Calorific Value Control 301
Introduction 301
Adjusting guide rates to control mean calorific value 303
Controlling the energy production rate 307
Using the facility 309
Summary of keywords 314

Chapter 24 - Geomechanics 315
Introduction 315
Implementation of geomechanics in ECLIPSE 300 317
Summary of keywords 323
Using the coupled geomechanics model 325

Chapter 25 - Gi Pseudo-compositional Model 331
Introduction 331
Using the Gi pseudo-compositional model 335
Summary of keywords 336
Example problem 337

Chapter 26 - The Gradient Option 344
Introduction 344
Data file overview 349
Output from the Gradient option 355
Gradient option technical description 364
Summary of keywords 368
Example problem (ECLIPSE 100 only) 373

Chapter 27 - Group Production Rules 383
Introduction 383
Drilling new wells 384
Opening new connections (re-perforating) 385
Reducing the THP limit 387
Changing VFP table numbers (retubing) 388
Changing well ALQ values (applying lift) 389
Activating compressors 391
The effect of closing wells during drilling and workovers 392
Using production rules with the network option 393

Chapter 28 - Hysteresis 394
Introduction 394
Using the hysteresis option 395
Relative permeability hysteresis in the non-wetting phase 396
Relative permeability hysteresis in the wetting phase 401
Modeling oil wet systems 403
Capillary pressure hysteresis 404

Chapter 29 - Hysteresis in WAG Floods 408
Introduction 408
Using the WAG hysteresis option 409
Non-wetting phase model (Gas) 410
Modification to the residual oil 413
Wetting phase model (Water) 414

Chapter 30 - Independent Reservoir Regions 416
Introduction 416
Using the option 417
Merging two or more corner point grids 418
Examples 419

Chapter 31 - Initializing the Study 421
Introduction 421
Data requirements 422
The equilibration algorithm 425
Calculating the initial conditions 426
Non-equilibrium initialization (ECLIPSE 300) 431

Chapter 32 - Local Grid Refinement and Coarsening 434
Introduction 434
Local grid refinement 435
Geometry and grid data in LGRs 456
Local grid coarsening 462
Automatic refinement (ECLIPSE 100) 466
Summary of keywords 469
Example LGR problem 472

Chapter 33 - Miscible Flood Modeling 479
Introduction 479
The Todd-Longstaff mixing parameter model 480
Effect of water saturation 481
Relative permeability model 482
Viscosity model 483
Component density model 484
The control of miscibility with pressure 486
Control of numerical diffusion in miscible flow simulation 488
Using the ECLIPSE miscible flood option 490

Chapter 34 - Multi-phase flash 492
Introduction 492
Multi-phase keywords 493
Using multi-phase 494

Chapter 35 - Multi-segment Wells 495
Introduction 495
The multi-segment well model 497
Pressure drops from VFP tables 503
The drift flux slip model 506
Down-hole separators (ECLIPSE 100) 515
Thermal multi-segment well model 520
Using the multi-segment well model 524
Converting friction wells to multi-segment wells 531
Summary of keywords 533

Chapter 36 - The Network Option 538
Introduction 538
Using the standard network option 544
The extended network model 552
The network model in ECLIPSE 300 562
Summary of keywords 564

Chapter 36 - Nine-point Schemes 567

Chapter 37 - Non-Darcy Flow 571
Introduction 571
Construction of flows 575
Velocity-dependent relative permeabilities 576

Chapter 38 - Parallel option 581
Introduction 581
Activating a parallel run 582
Partitioning 583
Running LGRs in parallel (ECLIPSE 100) 588
Performance bottlenecks 589
Memory scaling 591
Summary of keywords 592

Chapter 39 - Petro-elastic Model 593
Introduction 593
Assumptions 594
The Petro-elastic Model 595
Implementation of the Petro-elastic model 604

Chapter 40 - The Polymer Flood Model 607
Introduction 607
Using the polymer flood model 614
Summary of keywords 615
Example problems 618

Chapter 41 - Potential Calculations 628
Phase potentials 628
Well potentials 631

Chapter 42 - PVM 633
PVM introduction 633
Installing PVM 634
Starting PVM 636
Adding remote hosts to PVM 639
Troubleshooting 641
PVM versions used by GeoQuest releases 643
Limitations of PVM 644
..... 645

Chapter 43 - Recovery Mechanisms 646
Introduction 646
Solution gas drive 648
Water drive 649
Summary output 650
Details of the recovery calculation 651

Chapter 44 - Reservoir Coupling 653
Introduction 653
Using the reservoir coupling facility 662
Sharing licenses with the slaves 670
Reservoir coupling file structure 671
Summary of keywords 676

Chapter 45 - Restarts 678
Introduction 678
Restarts in ECLIPSE 100 679
Restarts in ECLIPSE 300 685

Chapter 46 - Rock Compaction 687
Introduction 687
Tabulated compaction curves 688
Water induced compaction (ECLIPSE 100) 692

Chapter 47 - Saturation Functions 693
Introduction 693
Water saturation properties 695
Gas saturation properties 697
Oil saturation properties 699
Three phase oil relative permeability models 702
Table end points 710
Consistency requirements 711
Near critical oil and gas relative permeabilities 713

Chapter 48 - Saturation Table Scaling 716
Introduction 716
Scaling of capillary pressure functions 718
Scaling of relative permeability functions 719
Miscellaneous points 723
Special applications 725
Consistency requirements 730
Example of end point scaling 731

Chapter 49 - Solid phase 733
Introduction 733
Solid keywords 734
Solid properties 736

Chapter 50 - Solution of the Linear Equations 744
Introduction 744
Material balance 745
Nested factorization preconditioning 746
Initialization for nested factorization preconditioner 748
Solution procedure for nested factorization 749
Orthomin 751
Convergence criteria 752
Treatment of wells 754
Nested factorization for LGRs 755
The ECLIPSE 300 WARP linear solver 757

Chapter 51 - The Solvent Model 763
Introduction 763
The Todd and Longstaff mixing parameter model 765
The relative permeability model 767
The PVT model 771
The pressure dependence of miscibility 775
Control of numerical diffusion in miscible flow simulation 776
Using the solvent model 778
Summary of keywords 781
Example problem 783

Chapter 52 - Surface Tension Effects 790
Evaluation of surface tensions 790
Use of surface tensions 791
Use of the MISCNUM option 795

Chapter 53 - The Surfactant Model 796
Introduction 796
The simulation model 797
Using the surfactant model 802
Summary of keywords 804
Example problem 806

Chapter 54 - Temperature option 814
Introduction 814

Chapter 55 - Tensor Permeability 819
Introduction 819

Discretization 821
Orthogonality error 823
Restrictions 825

Chapter 56 - Thermal Option 826

Introduction 826
Thermal keywords 828
Formulation 836
Thermal properties 845
Thermal features. 867
Converting a blackoil data set to thermal 877
Examples 882
Selected bibliography 884

Chapter 57 - Timing Reports 886

Chapter 58 - Total Compressibility Checks 889

Chapter 59 - Transmissibility Calculations 893

Introduction 893
Cartesian transmissibility calculations 895
Radial transmissibility calculations 901
Variations on the transmissibility calculation 904

Chapter 60 - Tracer Tracking 907

Introduction 907
Control of numerical diffusion 909
Using the ECLIPSE tracer tracking facility 910
Tracking original fluids 912
Delumping and lumping 918
Diffusion and dispersion 919
Treating passive tracers as non-linear 920

Chapter 61 - Units 921

The unit conventions 921
Conversion factors 923

Chapter 62 - Vertical Equilibrium 925

Introduction 925
Using the VE option 927
The cell saturation distribution 928
VE relative permeabilities and capillary pressures 933
Residual flow model 934
Miscellaneous points 936
Compressed vertical equilibrium option 938

Chapter 63 - The Wellbore Friction Option 941

Introduction 941
Using the wellbore friction option 944
Graphical presentation of friction effects in horizontal wells 950
Summary of keywords 956
Example problem 957

Chapter 64 - Well Inflow Performance 964
Introduction 964
The connection transmissibility factor 966
The flow-dependent skin factor 969
The phase mobilities 972
Special inflow equations 973
Injecting connections 980
The well bore head term 982
The productivity index 983

Chapter 65 - Well Modeling Facilities 985
Well completions 985
Well controls and limits 988
Group and field control facilities 994
Vertical flow performance 1009

AppendixA - RUNSPEC section: pre-95A fixed record structure 1011

AppendixB - RPTPROPS: pre-2002A integer controls 1022
`RPTPROPS` ..... Controls on output from PROPS section 1022

AppendixC - RPTGRID - definition with integer controls 1027
`RPTGRID` ..... Controls on output from GRID section 1027

AppendixD - RPTRST keyword: pre-2002A integer controls 1032
`RPTRST` 1032

AppendixE - RPTSCHED: pre-2002A integer controls 1037
`RPTSCHED` ..... Controls on output from SCHEDULE section 1037

AppendixF - RTPSOL: pre-2002A integer controls 1048
`RPTSOL` ..... Controls on output from SOLUTION section 1048

List of Figures

Chapter 1 - Introduction **23**

Chapter 2 - API Tracking **51**

Chapter 3 - Aquifer Modeling Facilities **55**

Chapter 4 - Brine Tracking **65**

Chapter 5 - Carbon Dioxide Solution in the Aqueous Phase **67**

Chapter 6 - Chemical Reactions **69**

Chapter 7 - The Coal Bed Methane Model **83**

Chapter 8 - Compatibility Between ECLIPSE 100 Special Options **107**

Chapter 9 - Convergence Reports **113**

Chapter 10 - Diffusion **123**

Chapter 11 - Dual Porosity **139**

Figure 11.1 A simple, dual porosity, dual permeability system. 141
Figure 11.2 A typical block of matrix material containing oil and water 143
Figure 11.3 A representative matrix block in a fracture potential gradient 147
Figure 11.4 Estimating the potential differences to the surrounding cells 148
Figure 11.12 A simple reservoir grid used to simulate a gas/oil transfer function 155
Figure 11.13 Initial fractional volume of water below the water below the water contact 160
Figure 11.14 Initial fractional volume of gas above the gas contact 161
Figure 11.15 Integrated capillary pressure 163
Figure 11.16 Nested matrix sub-cells in linear geometry 166
Figure 11.17 Nested matrix sub-cells in cylindrical and spherical geometry 167

Chapter 12 - Environmental Tracers **169**

Chapter 13 - Equations of State **185**

Chapter 14 - Equations of State Regions **197**

Chapter 15 - File Handling in ECLIPSE **201**

Chapter 16 - Flux Boundary Conditions **207**

Figure 16.1 A sequence of restarts that together make a full field run 212

Chapter 17 - The Foam Model **216**

Chapter 18 - Formulation of the Equations **232**
Figure 18.1 The most common instability using IMPES 240

Chapter 19 - Gas Field Operations Model **251**
Figure 19.1 Sales Gas Production Rate and Delivery Capacity 272

Chapter 20 - Gas Lift Optimization **276**
Figure 20.1 Optimum Lift Gas Injection Rate 277
Figure 20.2 Incremental gradient calculation 280

Chapter 21 - Gas Plants and NGLs **287**

Chapter 22 - The GASWAT Option **291**

Chapter 23 - Gas Calorific Value Control **301**

Chapter 24 - Geomechanics **315**
Figure 24.1 Control volumes (light grey grid) for rock force balance 319

Chapter 25 - Gi Pseudo-compositional Model **331**

Chapter 26 - The Gradient Option **344**
Figure 26.1 Order for writing records 356

Chapter 27 - Group Production Rules **383**

Chapter 28 - Hysteresis **394**
Figure 28.1 A typical pair of relative permeability curves for a non-wetting phase 396
Figure 28.2 A typical pair of relative permeability curves for a wetting phase 401
Figure 28.3 The water capillary pressure case 404
Figure 28.4 The gas capillary pressure case 406

Chapter 29 - Hysteresis in WAG Floods **408**
Figure 29.1 A typical drainage process followed by an imbibition process 410
Figure 29.2 Schematic diagram showing a typical secondary drainage curve 411
Figure 29.3 Two-phase and three-phase relative permeability curves 414
Figure 29.4 Subsequent drainage relative permeability 415

Chapter 30 - Independent Reservoir Regions **416**

Chapter 31 - Initializing the Study **421**

Chapter 32 - Local Grid Refinement and Coarsening **434**
Figure 32.1 Single column radial grid refinement 435
Figure 32.2 Single column radial grid refinement with NTHETA=1 436
Figure 32.3 Radial refinement in a four column box 437
Figure 32.4 Radial refinement in a four cell box with NTHETA=4 437
Figure 32.5 Cartesian grid refinement 439
Figure 32.6 A composite refinement 444
Figure 32.7 The numbering convention used by RADFIN for connecting radial LGRs to host cells 458

Figure 32.8 The numbering conventions used by RADFIN4 for connecting radial LGRs to host cells 458
Figure 32.9 Well positioning in automatically refined grids 468

Chapter 33 - Miscible Flood Modeling 479
Figure 33.1 The two-point upstream projection technique 488

Chapter 34 - Multi-phase flash 492

Chapter 35 - Multi-segment Wells 495
Figure 35.1 A multi-lateral, multi-segment well 497
Figure 35.2 Well segments 498
Figure 35.3 Allocating connection flows to segments 498
Figure 35.4 Hydrostatic head components 500
Figure 35.5 Segment diagram for a well with a down-hole water separator 516
Figure 35.6 Heat conduction between a well an the formation 521
Figure 35.7 Heat conduction between the tubing and the casing 523
Figure 35.8 Segment node positions with one segment per grid block connection 525
Figure 35.9 Thermal contact lengths.....527

Chapter 36 - The Network Option 538
Figure 36.1 common wellhead manifold and pipeline 539
Figure 36.2 Well THP versus Group Flow Rate for Wellstream and Pipeline 539
Figure 36.3 An example network 540
Figure 36.4 The network structure 552
Figure 36.5 The group control hierarchy 553

Chapter 36 - Nine-point Schemes 567
Figure 37.1 Splitting each five-point transmissibility into two half-transmissibilities.....568
Figure 37.2 Deriving the nine-point transmissibilities using diagonal coupling between cells.....569

Chapter 37 - Non-Darcy Flow 571

Chapter 38 - Parallel option 581

Chapter 39 - Petro-elastic Model 593

Chapter 40 - The Polymer Flood Model 607

Chapter 41 - Potential Calculations 628
Figure 42.1 Calculating the initial contact corrected potential 629

Chapter 42 - PVM 633

Chapter 43 - Recovery Mechanisms 646

Chapter 44 - Reservoir Coupling 653
Figure 44.1 Three coupled reservoirs; one is the master.....655
Figure 44.2 Three reservoirs coupled by a “dummy reservoir” master 655

Chapter 45 - Restarts **678**

Chapter 46 - Rock Compaction **687**
Figure 46.1 Rock Compaction: Hysteresis option 688
Figure 46.2 Rock Compaction: Boberg Hysteresis option 689
Figure 46.3 Initial multipliers 689

Chapter 47 - Saturation Functions **693**
Figure 47.1 The default 3-phase oil relative permeability model assumed by ECLIPSE 703
Figure 47.2 Calculating the minimum residual oil saturation from the water saturation 704
Figure 47.3 Calculating the minimum residual oil saturation from the gas saturation 705
Figure 47.4 Calculating the minimum residual oil saturation for drainage and imbibition with hysteresis 705
Figure 47.5 Ternary diagram showing the linear interpolation used to calculate S_{min} and S_{max} 708
Figure 47.6 Initial saturations for each zone 710
Figure 47.7 Ternary diagram showing mobile fluid end-points 712
Figure 47.8 The effect of gas injection 713

Chapter 48 - Saturation Table Scaling **716**
Figure 48.1 Two-point scaling 722
Figure 48.2 Three-point scaling 722
Figure 48.3 Water saturation versus depth for a typical grid cell that intersects the oil-water transition zone 725
Figure 48.4 Modeling a depth variation of critical saturation 727

Chapter 49 - Solid phase **733**

Chapter 50 - Solution of the Linear Equations **744**
Figure 50.1 The structure of the Jacobian matrix for a simple 3*2*3 problem. 746
Figure 50.2 Two color (red-blue) split suitable for a 4-processors parallel run 759
Figure 50.3 The sub-grids from with amalgamation of LGRs and base grid 759
Figure 50.4 The sub-grids from with no amalgamation of LGRs and base grid. 760

Chapter 51 - The Solvent Model **763**
Figure 51.1 The two point upstream projection technique 776

Chapter 52 - Surface Tension Effects **790**
Figure 52.1 Two graphs to illustrate the scaling of a rock curve 792
Figure 52.2 Two graphs to illustrate the scaling of a straight line 792

Chapter 53 - The Surfactant Model **796**
Figure 53.1 Calculation of the relative permeability 799

Chapter 54 - Temperature option **814**

Chapter 55 - Tensor Permeability **819**
Figure 55.1 Stencil for MPFA flow in 2D between cells (i, j) and (i + 1, j) 821
Figure 55.2 K-orthogonality error indicator 823

Chapter 56 - Thermal Option 826

Chapter 57 - Timing Reports 886

Chapter 58 - Total Compressibility Checks 889

Chapter 59 - Transmissibility Calculations 893
Figure 59.1 Y- and Z- transmissibility expressions 899

Chapter 60 - Tracer Tracking 907
Figure 60.1 Total oil production rate and contributions from above and below GOC 917

Chapter 61 - Units 921

Chapter 62 - Vertical Equilibrium 925
Figure 62.1 A cell with approximate hydrostatic equilibrium 925
Figure 62.2 Oil-water vertical equilibrium 928
Figure 62.3 Oil-gas vertical equilibrium 929
Figure 62.4 Gas-water vertical equilibrium 930
Figure 62.5 Three-phase vertical equilibrium 930
Figure 62.6 Three phase equilibrium with immobilized oil 931
Figure 62.7 Relative permeability behavior for a residual flow model 934
Figure 62.8 V.E Curves (SATNUM) 935
Figure 62.9 Residual flow (RESIDNUM) 935

Chapter 63 - The Wellbore Friction Option 941
Figure 63.1 A trunk and branch multi-lateral well 945
Figure 63.2 A bird's foot multi-lateral well 945
Figure 63.3 Defining multi-lateral well geometry 946
Figure 63.4 A multi-lateral well with sub-branches 947
Figure 63.5 Friction effects in a horizontal well 951

Chapter 64 - Well Inflow Performance 964

Chapter 65 - Well Modeling Facilities 985
Figure 65.1 A five-level hierarchy 994

AppendixA - RUNSPEC section: pre-95A fixed record structure 1011

AppendixB - RPTPROPS: pre-2002A integer controls 1022

AppendixC - RPTGRID - definition with integer controls 1027

AppendixD - RPTRST keyword: pre-2002A integer controls 1032

AppendixE - RPTSCHED: pre-2002A integer controls 1037

AppendixF - RTPSOL: pre-2002A integer controls 1048

List of Tables

Chapter 1 - Introduction **23**

Chapter 2 - API Tracking **51**

Chapter 3 - Aquifer Modeling Facilities **55**

Chapter 4 - Brine Tracking **65**

Chapter 5 - Carbon Dioxide Solution in the Aqueous Phase **67**

Chapter 6 - Chemical Reactions **69**

Table 6.1 Chemical reaction summary keywords 70
Table 6.2 RPTRST, and RPTSCHED output controls 70
Table 6.3 Example names for components 80
Table 6.4 Example stoichiometry 80
Table 6.5 Example reaction rate constants, activation energy and reaction enthalpy values 80

Chapter 7 - The Coal Bed Methane Model **83**

Table 7.1 Coal bed methane SUMMARY output controls 93

Chapter 8 - Compatibility Between ECLIPSE 100 Special Options **107**

Chapter 9 - Convergence Reports **113**

Chapter 10 - Diffusion **123**

Chapter 11 - Dual Porosity **139**

Table 11.1 Dual porosity keywords 153

Chapter 12 - Environmental Tracers **169**

Table 12.1 Keywords used to specify environmental tracer data 171
Table 12.2 Environmental tracer SUMMARY output controls 175

Chapter 13 - Equations of State **185**

Table 13.1 The coefficients, m1 and m2, variation with the equation of state 186
Table 13.2 The dependence of and on the equation of state 188

Chapter 14 - Equations of State Regions **197**

Chapter 15 - File Handling in ECLIPSE **201**

Chapter 16 - Flux Boundary Conditions **207**

Chapter 17 - The Foam Model **216**

Table 17.1 Foam SUMMARY output keywords 221

Chapter 18 - Formulation of the Equations 232

Chapter 19 - Gas Field Operations Model 251
Table 19.1 Useful gas field operations SUMMARY output controls 266
Table 19.2 Gas field operations SUMMARY output controls 273

Chapter 20 - Gas Lift Optimization 276
Table 20.1 Gas lift optimization SUMMARY output controls 286

Chapter 21 - Gas Plants and NGLs 287
Table 21.1 Keywords to obtain reports of recovered NGLs and gas 289

Chapter 22 - The GASWAT Option 291

Chapter 23 - Gas Calorific Value Control 301
Table 23.1 Gas calorific value control SUMMARY output controls 314

Chapter 24 - Geomechanics 315

Chapter 25 - Gi Pseudo-compositional Model 331
Table 25.1 Gi pseudo-compositional SUMMARY output controls 336

Chapter 26 - The Gradient Option 344

Chapter 27 - Group Production Rules 383

Chapter 28 - Hysteresis 394

Chapter 29 - Hysteresis in WAG Floods 408

Chapter 30 - Independent Reservoir Regions 416

Chapter 31 - Initializing the Study 421

Chapter 32 - Local Grid Refinement and Coarsening 434

Chapter 33 - Miscible Flood Modeling 479

Chapter 34 - Multi-phase flash 492
Table 34.1 Solid properties that can be output to the summary file 493
Table 34.2 Solid properties that can be output with RPTRST, RPTSCHED and RPTSOL 493

Chapter 35 - Multi-segment Wells 495
Table 35.1 The relationship between the dimensionless pipe diameter and the critical Kutateladze number 510
Table 35.2 Multi-segment wells SUMMARY output controls 533

Chapter 36 - The Network Option 538
Table 36.1 Network SUMMARY output controls (ECLIPSE 100) 564
Table 36.2 Network SUMMARY output controls (ECLIPSE 300) 564

Chapter 36 - Nine-point Schemes **567**

Chapter 37 - Non-Darcy Flow **571**

Chapter 38 - Parallel option **581**
Table 39.1 The division of cells between processors using the `PSPLITX` keyword. 587

Chapter 39 - Petro-elastic Model **593**

Chapter 40 - The Polymer Flood Model **607**
Table 41.1 Polymer flood SUMMARY output controls 616

Chapter 41 - Potential Calculations **628**

Chapter 42 - PVM **633**
Table 42.1 PVM versions used by GeoQuest releases 643

Chapter 43 - Recovery Mechanisms **646**
Table 43.1 Mnemonics for reporting recovery mechanism SUMMARY data 650

Chapter 44 - Reservoir Coupling **653**
Table 44.1 Reservoir coupling SUMMARY output controls 676

Chapter 45 - Restarts **678**

Chapter 46 - Rock Compaction **687**

Chapter 47 - Saturation Functions **693**

Chapter 48 - Saturation Table Scaling **716**

Chapter 49 - Solid phase **733**
Table 49.1 Solid properties that can be output to the summary file 735
Table 49.2 Solid production data 735
Table 49.3 Solid properties that can be output with RPTRST, RPTSCHED and RPTSOL 735

Chapter 50 - Solution of the Linear Equations **744**

Chapter 51 - The Solvent Model **763**
Table 51.1 Solvent SUMMARY output controls 782

Chapter 52 - Surface Tension Effects **790**

Chapter 53 - The Surfactant Model **796**
Table 53.1 Surfactant SUMMARY output controls 805

Chapter 54 - Temperature option **814**
Table 54.1 Default values used in calculating K-values 817
Table 54.2 Controlling summary output for temperature option 818
Table 54.3 Controlling summary output for temperature option 818

Chapter 55 - Tensor Permeability 819

Chapter 56 - Thermal Option 826
Table 56.1 Rock-fluid properties 829
Table 56.2 Hydrocarbon properties 829
Table 56.3 Water properties 831
Table 56.4 Field, Group and Well energy rates 831
Table 56.5 Miscellaneous quantities 832
Table 56.6 Multi-segment wells 832
Table 56.7 Inter-region flow quantities 833
Table 56.8 Block quantities 833
Table 56.9 RPTRST, RPTSOL and RPTSCHED output controls 834
Table 56.10 K-value coefficients (Field units) 847
Table 56.11 K-value coefficients (Metric units) 848
Table 56.12 Summary of convergence criteria and time stepping criteria 876
Table 56.13 Thermal example files 882

Chapter 57 - Timing Reports 886

Chapter 58 - Total Compressibility Checks 889

Chapter 59 - Transmissibility Calculations 893

Chapter 60 - Tracer Tracking 907

Chapter 61 - Units 921
Table 61.1 A table of the units used for four conventions 921
Table 61.2 Constants used in the four unit conventions 922
Table 61.3 Some useful conversion factors 923

Chapter 62 - Vertical Equilibrium 925

Chapter 63 - The Wellbore Friction Option 941
Table 63.1 Wellbore friction SUMMARY output controls 956

Chapter 64 - Well Inflow Performance 964

Chapter 65 - Well Modeling Facilities 985

AppendixA - RUNSPEC section: pre-95A fixed record structure 1011

AppendixB - RPTPROPS: pre-2002A integer controls 1022
Table B.1 `RPTPROPS` output controls 1023

AppendixC - RPTGRID - definition with integer controls 1027
Table C.1 `RPTGRID` output controls 1028

AppendixD - RPTRST keyword: pre-2002A integer controls 1032
Table D.1 `RPTRST` output controls 1033

AppendixE - RPTSCHED: pre-2002A integer controls 1037
Table E.1 `RPTSCHED` output controls 1038

AppendixF - RTPSOL: pre-2002A integer controls 1048
Table F.1 `RPTSOL` output controls 1049

-

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 FORTRAN77 and operate on any computer with an ANSI-standard FORTRAN77 compiler and with sufficient memory. Suitable computers include SUN SPARCstation, IBM RS/6000, Silicon Graphics, and Pentium (or better) PCs with more than 24 Mb of RAM.

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). The Parallel option is available on PC, IBM/SP, SGI Origin and SUN Enterprise computers.

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 39).

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 434). 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 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. 9]. 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 185 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 291 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. The programs FILL, GRID or FloGrid can be used to prepare corner-point data for ECLIPSE. GRAF, GRID or ECLIPSE Office may 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 efficiency by the ECLIPSE solution procedures.

See "Transmissibility Calculations" on page 893 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 timestep. IMPES and semi-implicit techniques cannot be used on such problems unless the timesteps 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 232 for further information.

ECLIPSE 300

Adaptive Implicit and IMPES (Compositional)

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 232 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 744.

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 40), 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.

ECLIPSE 100

Vertical equilibrium

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 925 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 139 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 716 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 408).

See "Hysteresis" on page 394 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 687 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 907 for further information.

ECLIPSE 100

API tracking

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 timestep to update APIs.

See "API Tracking" on page 51 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 65 for further information.

ECLIPSE 100

Temperature model

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 814 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 479 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 790 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, 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. In ECLIPSE 100 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 988 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 994 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 1000 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 1004 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 964 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 44) or the ECLIPSE 100 Wellbore Friction option (see "Wellbore friction" on page 41) 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 973 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 421 for further information.

Aquifers

Aquifers may be represented by a choice of two analytic models, a constant flux model (ECLIPSE 100 only) 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 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 55 for further information.

Ancillary programs

Ancillary programs available with ECLIPSE are described below.

FILL

FILL generates unconventional grid systems based on corner point geometry. Grids may be distorted to fit along fault lines. Sloping displacement faults, complex scissor faults, distorted wedge shaped grids etc. may be generated easily using FILL.

As its name implies, FILL is specially designed to interpolate sparse data which may only be known at a few points on the grid. Missing values are 'filled' in to generate complete grid data files for ECLIPSE. For example, rock properties such as porosity need only be specified at a few points in each grid layer and FILL provides the rest.

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 timesteps. 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.

PSEUDO

PSEUDO is the ECLIPSE Pseudo Package for generating coarse grid pseudo functions for low-cost sensitivity studies. PSEUDO uses ECLIPSE `RESTART` files, generated during a fine grid simulation, to generate pseudos for a coarse grid model. The fine grid model may be a cross-section with many layers or a large 3D model. The coarse grid model may be a 1D, 2D or 3D model and is formed by amalgamating neighboring cells of the fine grid model into large coarse grid cells.

Because PSEUDO runs separately from ECLIPSE, it is possible to redefine the coarse grid and generate the new pseudos inexpensively. Directional pseudo relative permeabilities, generated by PSEUDO and appropriate to each face of the coarse grid, may be input to the ECLIPSE coarse grid model to reproduce the results of the fine grid model or to perform low-cost sensitivity studies.

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.

ECLIPSE 100

Polymer flood model

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 timesteps.

See "The Polymer Flood Model" on page 607 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 538 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 207 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 timesteps 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 434 for further information.

ECLIPSE 100

Wellbore friction

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 941 for further information.

ECLIPSE 100

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,
- Automatic operation of compressors in the network, whenever they are needed to achieve the production target.

See "Gas Field Operations Model" on page 251 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 763 for further information.

ECLIPSE 100

GI pseudo-compositional model

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 hydro-carbons. 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 331 for further information.

ECLIPSE 100

Surfactant model

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 796 for further information.

ECLIPSE 100

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 276 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 581 for further information.

ECLIPSE 100

Environmental tracer

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 169 for further information.

ECLIPSE 100

Coal bed methane

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 83 for further information.

ECLIPSE 100

Foam model

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 216 for further information.

ECLIPSE 100

Reservoir coupling

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 653 for further information.

ECLIPSE 100

Gas calorific value control

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 301 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 495 for further information.

ECLIPSE 300

Thermal option

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 826 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.

GRID

GRID is an interactive program used to build reservoir models, design simulation grids and produce input data for ECLIPSE. The basic geological model consisting of contours, faults, map features and well locations may be digitized or input directly from industry standard map files. The geological model is used as a background display to aid the construction of the simulation grid.

Features include color graphics, corner-point or conventional geometry, sloping or vertical faults, areal, cross sectional or three dimensional displays, contouring back of grid properties to compare with original input data, flexible easy to use editing and comprehensive help facilities. Locally refined and coarsened grids can be prepared and displayed, for use with the Local Grid Refinement option.

The ECLMAP options in GRID provide facilities for handling seismic shot line data, volumetric calculations and depth conversion of seismic horizons or map grids.

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. Timesteps 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 44). 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 predictions 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.

In addition, GridSim is available as a grid and simulation data processor for FrontSim. It can be used to edit and visualize 3D solid graphics, and to visualize streamlines and line graphics. GridSim can also be used as a pre/post processor for ECLIPSE.

Licenses

To run ECLIPSE, and also to run any of the additional ECLIPSE Options, you need to obtain an appropriate FLEX license.

In addition, 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. A DATACHECK license will also allow a cross-section simulation case (either NX=1 or NY=1) to run.

The FLEX license checking procedure checks for the availability of an appropriate license for an ECLIPSE Options when a keyword activating the feature is encountered. In most cases the activating keyword is in the RUNSPEC section and is thus encountered at the beginning of the run. Activating keywords for some features, however, may not be encountered until the SCHEDULE section, perhaps 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.

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.

API Tracking

Chapter 2

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL

Introduction

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 923. 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.

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 3

Introduction

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

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

Two types of aquifer model are common to both simulators. These are:

- Numerical aquifers
- Carter-Tracy aquifers.

ECLIPSE 100 also has two additional models:

- Fetkovich aquifers
- Constant flux aquifers.

Numerical aquifers are represented by a one-dimensional row of cells within the simulation grid. The other three 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.

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

ECLIPSE 100

Transmissibility values are calculated using the OLDTRAN block centered method (see "Transmissibility Calculations" on page 893).

The transmissibility between two aquifer cells is thus given by:

$$\mathrm{TR} = \frac{\mathrm{CDARCY}}{1/T_i + 1/T_j} \qquad \text{[EQ 3.1]}$$

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
LENGTH_i	its length.

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 etc.) 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):

$$T_c = \frac{\mu_w \phi C_t r_o^2}{k_a c_1} \qquad \text{[EQ 3.2]}$$

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)

$$\beta = c_2 h \theta \phi C_t r_o^2 \qquad \text{[EQ 3.3]}$$

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

$$t_D = \frac{t}{T_c} \qquad \text{[EQ 3.4]}$$

The Carter-Tracy model uses tables of dimensionless time versus a dimensionless pressure influence function. The default table is for an infinite reservoir 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

$$p_{a0} - \bar{p} = \frac{Q_a}{\beta}\mathrm{PI}_D(t_D) \qquad \text{[EQ 3.5]}$$

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

$$\overline{Q_{ai}} = \alpha_i\{a - b[p_i(t + \Delta t) - p_i(t)]\} \qquad \text{[EQ 3.6]}$$

where

$$a = \frac{1}{T_c}\left\{\frac{\beta\Delta p_{ai} - W_a(t)\mathrm{PI}_D{}'(t + \Delta t)_D}{\mathrm{PI}_D(t + \Delta t)_D - t_D\mathrm{PI}_D{}'(t + \Delta t)_D}\right\} \qquad \text{[EQ 3.7]}$$

and

$$b = \frac{\beta}{T_c[\mathrm{PI}_D(t + \Delta t)_D - t_D\mathrm{PI}_D{}'(t + \Delta t)_D]} \qquad \text{[EQ 3.8]}$$

where

Δp_{ai}	is the pressure drop $p_{a0} + \rho g(d_i - d_a) - p_i(t + \Delta t)$
$\mathrm{PI}_D{}'$	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 3.6] contribute the residual for the implicit equations solved by ECLIPSE at time t. The cumulative aquifer influx $W_a(t)$ used in [EQ 3.7] is updated explicitly at the end of the timestep.

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 keyword AQUANCON.

ECLIPSE 100

If the initial aquifer pressure is defaulted, it is calculated from [EQ 3.11] 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.

ECLIPSE 100 only

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 (ECLIPSE 100)

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:

$$Q_{ai} = \frac{d}{dt}(W_{ai}) = J\alpha_i[p_a - p_i + \rho g(d_i - d_a)] \quad \text{[EQ 3.9]}$$

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.

If the ECLIPSE 100 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:

$$\alpha_i = \frac{m_i A_i}{\sum m_i A_i} \quad \text{[EQ 3.10]}$$

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

$$W_a = C_t V_{w0}(p_{a0} - p_a) \qquad \text{[EQ 3.11]}$$

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

$$T_c = \frac{C_t V_{w0}}{J} \qquad \text{[EQ 3.12]}$$

and has the dimension of time.

Under the assumption of uniform reservoir pressure in the connecting grid blocks, and by integrating [EQ 3.9] and [EQ 3.11], the average influx rate over the time interval Δt is expressed as

$$\overline{Q_{ai}} = \alpha_i J(p_a - p_i + \rho g(d_i - d_a))\left(\frac{1 - \exp(-\Delta t / T_c)}{\Delta t / T_c}\right) \qquad \text{[EQ 3.13]}$$

which is the form used by ECLIPSE 100 to compute the influx rates.

At the end of each timestep, the aquifer's cumulative total influx is incremented and its pressure is updated using [EQ 3.11].

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 keyword `AQUANCON`.

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 3.9] 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 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 (ECLIPSE 100)

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:

$$Q_{ai} = F_a \cdot A_i \cdot m_i \qquad \text{[EQ 3.14]}$$

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 keyword AQUANCON. 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.

Brine Tracking

Chapter 4

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

Introduction

The Brine Tracking facility enables ECLIPSE to model the mixing of waters with different salinities.

The concentration of salt within the water phase can modify both the water density and the viscosity. 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 initiated by the keyword BRINE in RUNSPEC.

The PVT properties

The PVT properties of water are specified using the PVTWSALT and BDENSITY keywords.

The PVTWSALT keyword replaces PVTW, and is used to specify the water properties at different values of the salt concentration. The first record of each table contains two items of data: the reference pressure and the reference salt concentration. The reference salt concentration defines the formation volume factor 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.

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. Keyword SALTVD supplies a table of salt concentrations versus depth for each equilibration region.

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

Wells and aquifers

The salinity of injected water can be specified independently for each water injector using the WSALT keyword. The default salt concentration is zero. Reports of group, well and connection flows can be obtained by setting the mnemonic WELLS in the RPTSCHED keyword.

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

The brine tracking calculation

The brine tracking calculation is performed after a converged solution for the timestep has been found, and the inter-block flows determined. A mass conservation equation is solved to update the salt concentrations. The new salt concentrations are then used to calculate the water properties for the subsequent timestep.

The distribution of the salt can be output using the mnemonic SALT in RPTSCHED, and the field, and for ECLIPSE 100, region fluid-in-place reports can be output by setting the mnemonic FIPSALT in RPTSCHED.

Carbon Dioxide Solution in the Aqueous Phase

Chapter 5

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

The CO_2 solution algorithm allows carbon dioxide to dissolve in the aqueous phase. This is switched on using the CO2SOL keyword in the RUNSPEC section. The basic model is a fugacity function for aqueous CO_2 which is constructed to match solubility data and which takes the form:

$$f^A_{co_2} = Pa_{CO_2}\phi(P) \qquad \text{[EQ 5.1]}$$

The phase equilibrium between the aqueous CO_2 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 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. To obtain details of the solubility data used in a simulation the SOLU mnemonic may be specified with the RPTPROPS keyword.

The function $\phi(P)$ is constructed by considering a pure CO_2/aqueous mixture - the gas phase fugacity is obtained using the equation of state.

The default aqueous phase properties follow the correlations given by Chang, Coats and Nolen [Ref. 3]. All or any of these properties may be overwritten; see the SOLUBILI keyword for details.

The initial dissolved CO_2 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. Note that the initial CO_2 concentration may not be in exact equilibrium with the hydrocarbon phases and initial flash modifies the input values slightly.

Chapter 6

Chemical Reactions

Introduction

x	ECLIPSE 100 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

REACTION	Number of reactions

PROPS section keywords

	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
	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

Table 6.1 Chemical reaction summary keywords

Field	Block	Information
FREAC	BREAC	Reaction rate
FREAT		Reaction total

SCHEDULE section keywords

Thermal option	REACCRIT	Reaction interpolation criteria

Table 6.2 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. 52].

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

$$\Sigma(S_{Rri} \cdot C_i) \rightarrow \Sigma(S_{Pri} \cdot C_i) \qquad \text{[EQ 6.1]}$$

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

$$R_r = V_b \cdot A_r \cdot \exp(-E_r/(RT)) \cdot \Pi c_{ri}^{\,n_{ri}} \qquad \text{[EQ 6.2]}$$

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,
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:

$$c_{ri} = \theta \cdot b_o \cdot S_o \cdot x_i \qquad \text{[EQ 6.3]}$$

If the reactant is in the gas phase:

$$c_{ri} = \theta \cdot b_g \cdot S_g \cdot y_i \qquad \text{[EQ 6.4]}$$

If the reactant is water in the water phase:

$$c_{ri} = \theta \cdot b_w \cdot S_w \qquad \text{[EQ 6.5]}$$

If the reactant is in all phases:

$$c_{ri} = \theta \cdot m_i \qquad \text{[EQ 6.6]}$$

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 6.4]:

$$c_{ri} = P \cdot y_i \qquad \text{[EQ 6.7]}$$

where P is the pressure.

If the reactant is in the solid phase [EQ 6.6] 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 6.2] 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 6.3] - [EQ 6.6], 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 6.2] can then be rewritten as

$$R_r = V_p \cdot \theta^{n_{rp}} \cdot A_r \cdot \exp(-E_r/(RT)) \cdot \Pi c'^{\,n_{ri}}_{ri} \qquad \text{[EQ 6.8]}$$

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 6.7]. 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 6.3] - [EQ 6.5] are modified, for example [EQ 6.3] 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 6.8] 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 6.7] or where the reactant is in all phases [EQ 6.6]. 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 6.2] is replaced with:

$$R_r = V_b \cdot A_r \cdot \exp(-E_r/(RT)) \cdot \Pi c_{ri}{}^{n_{ri}} \cdot \Pi d_{nijk} \qquad \text{[EQ 6.9]}$$

The equilibrium deviation terms d_{nijk} are given by one of two equations:

$$d_{1ijk} = \theta \cdot b_o \cdot S_o \cdot (x_i - y_j / K_k)$$ [EQ 6.10]

$$d_{3ijk} = \theta \cdot b_g \cdot S_g \cdot (K_k \cdot x_i - y_j)$$ [EQ 6.11]

The EQLDREAC keyword should be used to define the values of n, i, j and k, where

n	determines which of equations [EQ 6.10] - [EQ 6.11] 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 6.9] becomes

$$R_r = V_b \cdot A_r \cdot \exp(-E_r/(RT)) \cdot \Pi d_{nijk}$$ [EQ 6.12]

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 6.2] is replaced with:

$$R_r = V_b \cdot A_r \cdot \exp(-E_r/(RT)) \cdot \Pi c_{ri}^{\,n_{ri}} \cdot \Pi T_{rj}$$ [EQ 6.13]

The tracer multiplier terms T_{rj} are calculated explicitly (that is at the previous time step) as

$$T_{rj} = (\theta \cdot t_j \cdot c_c)^{n_{tj}}$$ [EQ 6.14]

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:

$$c_c = b_o \cdot S_o \cdot x_c \qquad \text{[EQ 6.15]}$$

If the tracer phase is gas:

$$c_c = b_g \cdot S_g \cdot y_c \qquad \text{[EQ 6.16]}$$

If the carrier is water and the tracer phase is water:

$$c_c = b_w \cdot S_w \qquad \text{[EQ 6.17]}$$

If the tracer is in all phases:

$$c_c = m_c \qquad \text{[EQ 6.18]}$$

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 timestep becomes:

$$R_{fl} = \frac{d}{dt}(V_p m_{fl}) - \sum_r [(S_{Pri} - S_{Rri}) \cdot R_r] = 0 \qquad \text{[EQ 6.19]}$$

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 timestep:

$$R_e = \frac{d}{dt}(V_b e) - \sum_r [H_r \cdot R_r] = 0 \qquad \text{[EQ 6.20]}$$

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 6.19] 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 6.20] 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 timestep and the previous one. Interpolation parameters can be specified using the `REACCRIT` keyword for the temperature used in the Arrhenius term in [EQ 6.2]:

$$T = \theta_T \cdot T_n + (1 - \theta_T) \cdot T_{n-1}, \quad 0 < \theta_T \leq 1$$

and the oil saturation used in [EQ 6.3]:

$$S_o = \theta_S \cdot S_{o,n} + (1 - \theta_S) \cdot S_{o,n-1}, \; 0 < \theta_S \leq 1$$

Output

The reaction rate [EQ 6.2] 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. 52] gives a thermal example with 5 components plus water:

Table 6.3 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 6.4 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 6.5 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 7

Introduction

x	ECLIPSE 100
	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.

The Coal Bed Methane model uses a modified Warren and Root dual porosity model to describe the physical processes involved in a typical coal bed methane project.

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.

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.

Adsorption/diffusion models

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:

$$F_g = \mathrm{DIFFMF} \cdot D_c \cdot (\mathrm{GC}_b - \mathrm{GC}_s) \qquad \text{[EQ 7.1]}$$

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:

$$\mathrm{DIFFMF} = \mathrm{DIFFMMF} \cdot \mathrm{VOL} \cdot \sigma \qquad \text{[EQ 7.2]}$$

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.

Kazemi [Ref. 4] has proposed the following form for σ:

$$\sigma = 4\left(\frac{1}{l_x^2} + \frac{1}{l_y^2} + \frac{1}{l_z^2}\right) \quad \text{[EQ 7.3]}$$

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. (Thus l_x, l_y and l_z are 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 either 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.

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.

$$F_g = \text{DIFFMF} \cdot D_c \cdot S_g \cdot \text{RF} \cdot (\text{GC}_b - \text{GC}_s) \quad \text{[EQ 7.4]}$$

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.

Modeling compositional effects

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 = $S_g/(S_g + S_{solv})$

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 = $S_g/(S_g + S_{solv})$

Using the 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:

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:

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:

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:

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:

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:

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:

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}.

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.

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.

Summary data

Three extra SUMMARY section keywords are available to output the coal solvent concentration on a field, region and block basis:

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:

`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

RUNSPEC section

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

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

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.
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

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:

Table 7.1 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
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

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

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

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
```

Compatibility Between ECLIPSE 100 Special Options

Chapter 8

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL

Introduction

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 Solvent 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
- Parallel options (in `USEFLUX` runs).

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.

Gradient option

The Gradient option is incompatible with:

- Parallel options
- 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 Brine Tracking and Temperature options 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
- The Gradient option
- `USEFLUX` runs (Flux Boundary Conditions).

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.

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:

- The Coal Bed Methane model
- Flux Boundary Conditions
- Multi-segment Wells
- The Wellbore Friction option.

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.

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 9

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 1 of 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 NTRAN=
  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 NTRAN=
  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 NTRAN=
  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 NTRAN=
  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 timestep 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 232 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:

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:

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).
DIFF	Following a CHOP the timestep 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.
DLYA	Time-step has been selected to enact a delayed action at the appropriate time.

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

$$\text{TIME TRNC ERR} = dS - \frac{dS_p}{dt_p} dt \qquad \text{[EQ 9.1]}$$

where

dS	is the saturation change during the current timestep, dt, and
dS_p	is the saturation change during the previous timestep, 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.

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:

```
Init : Initial step
Spec : Specified next step
SCT  : Solution change target
TPT  : Throughput target
PCT  : Pressure change target
MIF  : Max increase factor
TTE  : Time Trunction Error
TEC  : Temperature Change (thermal)
EnC  : Energy Change (thermal)
SAC  : Saturation Change (thermal)
CFL  : Stability limited
NIte : Number of Iterations
MDF  : Max decrease factor
Max  : Maximum step
Min  : Minimum step
MaxW : Maximum step after well modification
WAG  : Step to WAG change
Rep  : Step to report
HRep : Half step to report
Redu : Reduced step
```

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 693, so the use of an effective saturation is preferred. Consider the expression:

$$\Delta S' = \frac{dU^T}{dm_c} \cdot \Delta m_c \qquad \text{[EQ 9.2]}$$

where

$$u^T = \frac{m_t L}{b_o^m} + \frac{m_t V}{b_g^m} + \frac{m_w}{b_w^m} \qquad \text{[EQ 9.3]}$$

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 timestep 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 timestep 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 timestep, divided by the amount of that fluid (or component) in the grid block at the start of the timestep 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 timesteps 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 timestep 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 timestep is reduced.

Time truncation error (TTE)

A further control on the timestep is provided by the time truncation error (TTE) control. A comparison is made in the change in the solution over a step,

$$\Delta X_a = X^{T+\Delta T} - X^T \qquad \text{[EQ 9.4]}$$

with the change estimated from the rate of change at the previous step:

$$\Delta X_e = \frac{dX}{dT} \cdot \Delta T \qquad \text{[EQ 9.5]}$$

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

$$\left|\Delta X_a - \Delta X_e\right| \cdot \frac{dU^T}{dX} \qquad \text{[EQ 9.6]}$$

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, timestep 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 timestep, 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 10

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

	DIFFUSE	Enables molecular diffusion

GRID section

	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

	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

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 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

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.

Consider a gas component in the gas phase. The diffusion coefficient is given by the diffusive flow relationship (for more information see [Ref. 6]):

$$J_g = -c \cdot D_{gg} \cdot \frac{dx_g}{dl} \qquad \text{[EQ 10.1]}$$

The diffusive flow of gas at surface conditions is then given by:

$$F_{gg} = \frac{\mathrm{MW}_g \cdot J_g \cdot A \cdot \phi}{\rho_g} \qquad \text{[EQ 10.2]}$$

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:

$$M_{\text{gas}} = \frac{S_g \cdot \rho_g}{B_g \cdot \mathrm{MW}_g} \qquad M_{\text{oil}} = \frac{R_v \cdot S_g \cdot \rho_o}{B_g \cdot \mathrm{MW}_o} \qquad \text{[EQ 10.3]}$$

hence

$$c = \frac{S_g}{B_g}\left(\frac{\rho_g}{\mathrm{MW}_g} + \frac{R_v \cdot \rho_o}{\mathrm{MW}_o}\right) = \frac{S_g \cdot \rho_g}{B_g \cdot \mathrm{MW}_g}(1 + R_v F) \qquad \text{[EQ 10.4]}$$

$$x_g = \frac{\rho_g / \mathrm{MW}_g}{\rho_g / \mathrm{MW}_g + R_v \rho_o / \mathrm{MW}_o} = \frac{1}{1 + R_v F} \qquad \text{[EQ 10.5]}$$

with

$$F = \frac{\rho_o \cdot \mathrm{MW}_g}{\rho_g \cdot \mathrm{MW}_o} \qquad \text{[EQ 10.6]}$$

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 10.1], [EQ 10.5], [EQ 10.6] into equation [EQ 10.2] gives the diffusive flow of gas:

$$F_{gg} = -\frac{S_g}{B_g} \cdot \frac{F}{1 + R_v F} \cdot D_{gg} \cdot \frac{dR_v}{dl} \cdot A \cdot \phi \qquad \text{[EQ 10.7]}$$

In the simulator the diffusive flow of gas from cell i to j is:

$$F_{gg} = -\frac{S_g}{B_g} \cdot \frac{F}{1 + R_v F} \cdot D_{gg} \cdot (R_{vi} - R_{vj}) \cdot T_d \qquad \text{[EQ 10.8]}$$

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 $(R_{vi} + R_{vj})/2$
D_{gg}	is the gas-in-gas diffusion coefficient.

"Diffusivity calculations" on page 133 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:

$$F_{og} = \frac{S_g}{B_g} \cdot \frac{F}{1 + R_v F} \cdot D_{og} \cdot (R_{vi} - R_{vj}) \cdot T_d$$

$$F_{oo} = \frac{S_o}{B_o} \cdot \frac{F}{F + R_s} \cdot D_{oo} \cdot (R_{si} - R_{sj}) \cdot T_d$$

$$F_{go} = -\frac{S_o}{B_o} \cdot \frac{F}{F + R_s} \cdot D_{go} \cdot (R_{si} - R_{sj}) \cdot T_d$$

[EQ 10.9]

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. 6]:

$$J_i = -cD_i \frac{\partial x_i}{\partial d}. \qquad \text{[EQ 10.10]}$$

In the second model diffusion is driven by the gradient of the chemical potential:

$$J_i = -cD_i^a x_i \frac{1}{RT}\frac{\partial}{\partial d}[\mu_i - M_i G(h - h_o) + M_i D_i^T \ln(T)] \qquad \text{[EQ 10.11]}$$

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

$$\mu_i = \mu_{i0} + \mathrm{RT}\ln(f_i) \qquad \text{[EQ 10.12]}$$

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 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 10.10] 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 10.11]. 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 10.11] can be written as:

$$J_i = -cD_i^a x_i \frac{\partial}{\partial d}\ln(f_i). \qquad \text{[EQ 10.13]}$$

Where $\partial\ln(f_i)/\partial d$ can be written as:

$$\frac{\partial}{\partial d}\ln(f_i) = \frac{\partial}{\partial \ln(x_i)}\ln(f_i)\cdot\frac{\partial}{\partial x_i}\ln(x_i)\cdot\frac{\partial x_i}{\partial d} = \frac{\partial}{\partial \ln(x_i)}\ln(f_i)\cdot\frac{1}{x_i}\cdot\frac{\partial x_i}{\partial d} \qquad \text{[EQ 10.14]}$$

and we have ignored terms $\partial\ln(f_i)/\partial x_j$. [EQ 10.13] can then be written as:

$$J_i = -\left(c\cdot D_i^a\cdot\frac{\partial}{\partial \ln(x_i)}\ln(f_i)\cdot\frac{\partial x_i}{\partial d}\right). \qquad \text{[EQ 10.15]}$$

Comparing [EQ 10.10] and [EQ 10.15], D_i^a is given by:

$$D_i^a = \frac{D_i}{\partial\ln(f_i)/\partial\ln(x_i)}. \qquad \text{[EQ 10.16]}$$

D_i^a is therefore the activity-corrected diffusion coefficient of [Ref. 6].

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

$$J_i = J_{io} + J_{ig} \quad \text{[EQ 10.17]}$$

with J_{io}, the oil phase molar flux of component i given in terms of activity-corrected coefficients by:

$$J_{io} = -\left(\phi \cdot S_o \cdot b_o^m \cdot D_{io}^a \cdot x_i \cdot \frac{1}{RT} \cdot \frac{\partial}{\partial d}(RT\ln f_i - M_i Gh + M_i D_{io}^T \ln T)\right) \quad \text{[EQ 10.18]}$$

or in terms of normal diffusion coefficients:

$$J_{io} = -\left(\phi \cdot S_o \cdot b_o^m \cdot D_{io} \cdot \frac{\partial x_i}{\partial d}\right) \quad \text{[EQ 10.19]}$$

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:

$$F_i^{\text{diff}} = F_{io}^{\text{diff}} + F_{ig}^{\text{diff}} \quad \text{[EQ 10.20]}$$

where:

$$F_{io}^{\text{diff}} = T_D D_{io}^a \left(\frac{x_i S_o b_o^m}{RT}\right)_u \Delta(RT\ln f_i - M_i \mathrm{G}h + M_i D_{io}^T \ln T) \quad \text{[EQ 10.21]}$$

$$F_{ig}^{\text{diff}} = T_D D_{ig}^a \left(\frac{y_i S_g b_g^m}{RT}\right)_u \Delta(RT\ln f_i - M_i \mathrm{G}h + M_i D_{io}^T \ln T). \quad \text{[EQ 10.22]}$$

where (x_i, y_i) are the liquid and vapor mole fractions of component i, respectively.

In terms of normal diffusion coefficients:

$$F_{io}^{\text{diff}} = T_D D_{io}(S_o b_o^m)_u \Delta x_i \quad \text{[EQ 10.23]}$$

$$F_{ig}^{\text{diff}} = T_D D_{ig}(S_g b_g^m)_u \Delta y_i. \quad \text{[EQ 10.24]}$$

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:

$$T_D = \frac{\phi A}{d} \qquad \text{[EQ 10.25]}$$

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 133, 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 10.11], the diffusive flow will be zero when

$$\frac{\partial}{\partial d}[\mu_i - M_i g h + M_i D_i^T \ln(T)] = 0 \qquad \text{[EQ 10.26]}$$

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 10.26] 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 timesteps can be used if the simulation is run using the full implicit method `FULLIMP`.

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:

$$F_{gx} = -\frac{S_c}{B_o} \cdot \frac{F}{(F + R_{sj})} \cdot D_{gx} \cdot (R_{si}' - R_{sj}) \cdot T_d$$

$$F_{ox} = \frac{S_c}{B_o} \cdot \frac{F}{F + R_{sj}} \cdot D_{ox} \cdot (R_{si}' - R_{sj}) \cdot T_d$$

[EQ 10.27]

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

$$T_{D_{x_I}} = \frac{M_{D_{x_I}} A F_{\text{dip}}}{B} \qquad \text{[EQ 10.28]}$$

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:

$$A = \frac{(D_{x_J} D_{y_I} D_{z_I} R_{ntg_I} + D_{x_I} D_{y_J} D_{z_J} R_{ntg_J})}{(D_{x_I} + D_{x_J})} \qquad \text{[EQ 10.29]}$$

$$B = \frac{\left(\frac{D_{x_I}}{\phi_I} + \frac{D_{x_J}}{\phi_J}\right)}{2} \qquad \text{[EQ 10.30]}$$

$$F_{\text{dip}} = \frac{D_h^2}{D_h^2 + D_v^2} \qquad \text{[EQ 10.31]}$$

with:

$$D_h^2 = \left(\frac{D_{x_I} + D_{x_J}}{2}\right)^2 \qquad \text{[EQ 10.32]}$$

$$D_v^2 = (d_I - d_J)^2 \qquad \text{(where } d \text{ denotes depth)} \qquad \text{[EQ 10.33]}$$

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:

$$T_{D_{x_I}} = \frac{M_{D_{x_I}}}{\frac{1}{T_I} + \frac{1}{T_J}} \qquad \text{[EQ 10.34]}$$

with:

$$T_I = \frac{\phi_I R_{ntg_I} \boldsymbol{A} \cdot \boldsymbol{D_I}}{\boldsymbol{D_I} \cdot \boldsymbol{D_I}} \quad \text{[EQ 10.35]}$$

where

$$\boldsymbol{A} \cdot \boldsymbol{D_I} = A_x D_{x_I} + A_y D_{y_I} + A_z D_{z_I} \quad \text{[EQ 10.36]}$$

and

$$\boldsymbol{D_I} \cdot \boldsymbol{D_I} = D_{x_I}^2 + D_{y_I}^2 + D_{z_I}^2 . \quad \text{[EQ 10.37]}$$

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:

$$T_{D_{r_I}} = \frac{M_{D_{r_I}} F_{\text{dip}}}{\frac{1}{T_I} + \frac{1}{T_J}} \quad \text{[EQ 10.38]}$$

with:

$$T_I = \frac{\phi_I R_{ntg_I} D_{\theta_I} D_{z_I}}{D_{1p}} \quad \text{[EQ 10.39]}$$

$$D_{1p} = \frac{R_1^2}{(R_2^2 - R_1^2)} \ln(R_1 / R_2) + \frac{1}{2} \quad \text{[EQ 10.40]}$$

and

$$T_J = \frac{\phi_J R_{ntg_J} D_{\theta_J} D_{z_J}}{D_{2m}} \quad \text{[EQ 10.41]}$$

$$D_{2m} = \frac{R_3^2}{(R_3^2 - R_2^2)} \ln\left(\frac{R_3}{R_2}\right) - \frac{1}{2}, \quad \text{[EQ 10.42]}$$

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:

$$T_D = \text{DIFFMMF} \cdot \phi_m \cdot V \cdot \sigma \qquad \text{[EQ 10.43]}$$

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. 4] has proposed the following form for σ:

$$\sigma = 4\left(\frac{1}{l_x^2} + \frac{1}{l_y^2} + \frac{1}{l_z^2}\right) \qquad \text{[EQ 10.44]}$$

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 and DIFFCGAS 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 11

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:

$$\mathrm{TR} = \mathrm{CDARCY} \cdot K \cdot V \cdot \sigma \qquad \text{[EQ 11.1]}$$

where by default:

K	is taken as the X-direction permeability of the matrix blocks,
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.

Kazemi [Ref. 4] has proposed the following form for σ:

$$\sigma = 4\left(\frac{1}{l_x^2} + \frac{1}{l_y^2} + \frac{1}{l_z^2}\right) \qquad \text{[EQ 11.2]}$$

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:

$$\mathrm{TR} = \mathrm{CDARCY} \cdot V \cdot \left(\frac{f_x \cdot K_x}{l_x^2} + \frac{f_y \cdot K_y}{l_y^2} + \frac{f_z \cdot K_z}{l_z^2}\right) \qquad \text{[EQ 11.3]}$$

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 11.1 A simple, dual porosity, dual permeability system.

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 by including either keyword `GRAVDR` or `GRAVDRM` (described in the next section) in the `RUNSPEC` section and by giving a non-zero value to `DZMTRX` in the `GRID` section. 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 11.2 A typical block of matrix material containing oil and water

The pressure difference due to gravity alone is

$$\Delta P = DZ_{mat}(X_W - X_w)(\rho_w - \rho_o)g \qquad \text{[EQ 11.12]}$$

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

$$P_{Scow} = DZ_{mat} \cdot X_W(\rho_o - \rho_w)g \qquad \text{(fracture cells)}$$

$$P_{Scow} = DZ_{mat} \cdot X_w(\rho_o - \rho_w)g \qquad \text{(matrix cells)}$$

$$P_{Scog} = DZ_{mat} \cdot X_G(\rho_o - \rho_g)g \qquad \text{(fracture cells)}$$

$$P_{Scog} = DZ_{mat} \cdot X_g(\rho_o - \rho_g)g \qquad \text{(matrix cells)}$$

$$P_{Scwg} = DZ_{mat} \cdot X_G(\rho_w - \rho_g)g \qquad \text{(fracture cells)}$$

$$P_{Scwg} = DZ_{mat} \cdot X_g(\rho_w - \rho_g)g \qquad \text{(matrix cells)}$$

Thus, the flow of oil and gas from a fracture to a matrix cell in a gas oil system is computed as

$$F_g = \mathrm{TR} \cdot \mathrm{GMOB} \times \left(P_{of} - P_{om} + d_{fm}\rho_g g + P_{cogf} - P_{cogm} + \frac{\mathrm{DZ}_{mat}(X_G - X_g)(\rho_o - \rho_g)g}{2}\right)$$

[EQ 11.13]

$$F_o = \mathrm{TR} \cdot \mathrm{OMOB}\left(P_{of} - P_{om} + d_{fm}\rho_o g - \frac{\mathrm{DZ}_{mat}(X_G - X_g)(\rho_o - \rho_g)g}{2}\right)$$

[EQ 11.14]

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 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 421) 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:

$$\sigma_p^{\text{eff}} = f_p \cdot \sigma + (1 - f_p) \cdot \sigma_{gd}$$ [EQ 11.15]

where p =oil, gas, water, and

$$f_p = \frac{|\Delta P_p|}{|\Delta P_p| + |\Delta P_{pgd}|}$$ [EQ 11.16]

ΔP_{pij} is the usual potential difference term between two cells i and j:

$$\Delta P_{pij} = P_{pi} - P_{pj} + P_{pci} - P_{pcj} - G\bar{\rho}_p \Delta d$$ [EQ 11.17]

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:

$$\Delta P_o = -\frac{1}{2}(\Delta P_{gd}^{og} + \Delta P_{gd}^{ow})$$ [EQ 11.18]

$$\Delta P_g = \frac{1}{2}\Delta P_{gd}^{og}$$ [EQ 11.19]

$$\Delta P_w = \frac{1}{2}\Delta P_{gd}^{ow}$$ [EQ 11.20]

$$\Delta P_{gd}^{og} = G \times DZ_{\text{mat}}(\bar{\rho}_o - \bar{\rho}_g)(X_{ig} - X_{jg})$$ [EQ 11.21]

$$\Delta P_{gd}^{ow} = G \times DZ_{\text{mat}}(\bar{\rho}_w - \bar{\rho}_o)(X_{iw} - X_{jw})$$ [EQ 11.22]

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

An alternative gravity drainage model is available based on the formulation proposed by Quandalle and Sabathier in [Ref. 7].

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:

$$F = F_h + F_{up} + F_{down}$$ [EQ 11.23]

Where the horizontal flow is:

$$F_h = TR_{sigma} \cdot MOB_h \cdot (P_f + P_{cf} - P_m - P_{cm})$$ [EQ 11.24]

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:

$$F_v = TR_{sigmagd} \cdot MOB_v \cdot (P_f + P_{cf} - P_m - P_{cm} + (\rho_m - \rho_f^{star}) \cdot g \cdot (DZ_{\mathrm{mat}})/2)$$ [EQ 11.25]

$TR_{sigmagd}$ 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.

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 123.

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. 8]).

Figure 11.3 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,

$$F = \frac{T}{2} \cdot M_m(P_{m1} - P_1) + \frac{T}{2} \cdot M_m(P_{m1} - P_2) \qquad \text{[EQ 11.12]}$$

where:

T	is the matrix-fracture transmissibility
M_m	is the matrix mobility.

Hence

$$F = T \cdot M_m(P_{m1} - P_f) \qquad \text{[EQ 11.13]}$$

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

$$F = T \cdot M_f (P_f - P_{m3}) \qquad \text{[EQ 11.14]}$$

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

$$F = \frac{T}{2} \cdot M_f \cdot (P_1 - P_{m2}) - \frac{T}{2} \cdot M_m \cdot (P_{m2} - P_2) \qquad \text{[EQ 11.15]}$$

This can be written as

$$F = \frac{T}{2} \cdot M_f \cdot \left(P_f + \frac{(P_1 - P_2)}{2} - P_{m2}\right) - \frac{T}{2} \cdot M_m \cdot \left(P_{m2} - P_f + \frac{(P_1 - P_2)}{2}\right) \qquad \text{[EQ 11.16]}$$

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 11.16] can be written as a standard matrix-fracture flow with modified up-winding and an extra term associated with the viscous displacement

$$F = \frac{T}{2} \cdot M_f \cdot (P_f - P_{m2}) - \frac{T}{2} \cdot M_m \cdot (P_{m2} - P_f) + \frac{T}{4} \cdot G \cdot L \cdot (M_f - M_m) \qquad \text{[EQ 11.17]}$$

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 11.4 Estimating the potential differences to the surrounding cells

$$G = \frac{|P_l - P|}{2D_l} + \frac{|P - P_r|}{2D_r} \qquad \text{[EQ 11.12]}$$

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 timestep.

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.

Partially fractured reservoirs

ECLIPSE 100

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

ECLIPSE 100

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 section.

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 11.1 shows a summary of dual porosity keywords.

Table 11.1 Dual porosity keywords

Keyword	Status	Brief Description of Data	Availability
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
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
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

Table 11.1 Dual porosity keywords (Continued)

Keyword	Status	Brief Description of Data	Availability
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
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 11.12 A simple reservoir grid used to simulate a gas/oil transfer function

M1	F1
M2	F2

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 be 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:

$$K_r = \overline{K_r} + M(K_r(s) - \overline{K_r}) \qquad \text{[EQ 11.12]}$$

where

$\overline{K_r}$	is the relative permeability at $\bar{s} = (\text{SOWCR} + 1.0 - \text{SWCO})/2$
$K_r(s)$	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:

$$\bar{s} + M(s' - \bar{s}) = s \qquad \text{[EQ 11.13]}$$

where

$\bar{s}$	is (SOWCR + 1.0 – 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:

$$K_{rf}(S_w) = K_{rf}^{\text{tab}}(S_w) \cdot K_{rm\,\max} / K_{rf\max} \qquad \text{[EQ 11.14]}$$

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 $S_w = 1.0 - S_{ocrw}$
$K_{rf\max}$	is the maximum fracture relative permeability
S_{ocrw}	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 $S_g = 1.0 - S_{ocrg}$ 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

$$(B - L \cdot A^{-1} U)y = R_f - L \cdot A^{-1} R_m \qquad \text{[EQ 11.15]}$$

$$x = A^{-1}(R_m - Uy) \qquad \text{[EQ 11.16]}$$

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.

$$D = \begin{bmatrix} D_m & U \\ L & D_f \end{bmatrix} \qquad \text{[EQ 11.17]}$$

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 744). Using the notation of that chapter, an approximation, B, is constructed to the full Jacobian matrix

$$A = D + L1 + U1 + L2 + U2 + L3 + U3 \ , \qquad \text{[EQ 11.18]}$$

where the band terms $L1$ and $U1$ etc. represent flows between matrix or fracture cells. The following nested sequence of factorizations is used

$$\begin{aligned} B &= (P + L3)P^{-1}(P + U3) \\ P &= (T + L2)T^{-1}(T + U2) \\ T &= (G + L1)G^{-1}(G + U1) \\ G &= (\Gamma + L)\Gamma^{-1}(\Gamma + U) \end{aligned} \qquad \text{[EQ 11.19]}$$

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.

Calculation of fractional volumes for gravity drainage

The initial fractional volume of water below the water contact, X_{wi}, is

$$X_{wi} = \frac{S_{wi} - S_{wco}}{1 - S_{cohy} - S_{wco}} \qquad \text{[EQ 11.20]}$$

Figure 11.13 Initial fractional volume of water below the 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

$$X_w = 1 \text{ when}$$

$$S_w = S_{w\,\max} = X_{wi}(1 - S_{cohy}) + (1 - X_{wi})(1 - S_{crhy})$$

$$X_w = (S_w - X_{wi}(S_{crhy} - S_{cohy}) - S_{wco})/(1 - S_{crhy} - S_{wco}) \text{ when } S_w < S_{w\,\max}$$

If $S_w < S_{wi}$ then

$$X_w = 0 \text{ when } S_w = S_{w\,\min} = X_{wi}S_{wcr} + (1 - X_{wi})S_{wco}$$

$$X_w = (S_w - X_{wi}(S_{wcr} - S_{wco}) - S_{wco})/(1 - S_{cohy} - S_{wcr}) \text{ when } S_w > S_{w\,\min}$$

The initial fractional volume of gas above the gas contact, X_{gi}, is

$$X_{gi} = \frac{S_{gi} - S_{gco}}{1 - S_{lco} - S_{gco}} \qquad \text{[EQ 11.12]}$$

Figure 11.14 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.

Integrated capillary pressure option

When the 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

$$p_c = P_c(S_g) \text{ and } S_g = P_c^{-1}(p_c)$$

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 11.15 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

$$\overline{S_g} = \frac{1}{P} \cdot \int_0^P P_c^{-1}(p)\, dp \qquad \text{[EQ 11.12]}$$

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.) The gravity head P_{gdr} is proportional to the mobile oil saturation, assuming the fracture is filled with gas:

$$P_{gdr} = \frac{S_t - \overline{S_g}}{S_t} \cdot \Delta\rho \cdot g \cdot h \qquad \text{[EQ 11.13]}$$

Where

S_t	is 1 – SWCO – SOCRG
SWCO	is the connate water saturation
SOCRG	is the critical water saturation
$\Delta\rho$	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

$$P'_c(\overline{S_g}) = \frac{S_t - \overline{S_g}}{S_t} \cdot P \qquad \text{[EQ 11.14]}$$

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 11.12], which is stored in ECLIPSE as a table.

Notes

1 By construction, for any block height h the 'correct' final saturation is achieved by running the simulator to a stable state when:

$$P'_c(S) = \text{gravity drainage head,}$$

$$S = \bar{S} \text{ for that block height.}$$

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 11.16 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 11.17 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.

Solution method and efficiency

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 gravity drainage models (GRAVDR, GRAVDRM) or the Local Grid Refinement option.

Environmental Tracers

Chapter 12

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 gives 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.

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.

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:

$$F_t = \text{DIFF} \cdot D_c S_h (T_{ci} - T_{cj}) \qquad \text{[EQ 12.1]}$$

where

F_t	is the tracer flow
DIFF	is the diffusivity
D_c	is the diffusion coefficient
S_h	is the host phase saturation
T_{ci}	is the Tracer concentration in cell i
T_{cj}	is the Tracer concentration in cell j

The diffusivity (DIFF) is analogous to the transmissibility and has the form:

$$\text{DIFF} = \frac{A \cdot \phi}{d} \qquad \text{[EQ 12.2]}$$

where

A	is the interface area between the two cells
ϕ	is the porosity
d	is the distance between the cell centers.

The details of the diffusivity calculation can be found under "Diffusion" on page 123.

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.

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 12.1 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.
TBLK or TVDP / TNUM	SOLUTION	Specify the initial tracer concentration.
WTRACER	SCHEDULE	Inject tracer through wells.

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.

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.

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.

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 TRACER keyword. Note that the adsorption data (TRADS 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 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 new 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.

Restrictions

At present the adsorption, decay and diffusion options are only available for single phase tracers. Hence, 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.

Environmental tracers employing adsorption and decay can use either 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.

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.

Partitioned tracers

Partitioned tracers allows 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 the gas phase (say), ECLIPSE solves a conservation equation for the total amount of tracer, $c_g M_g$, in a grid block taking into account the inflow and outflow of gas. Actually ECLIPSE does not work with the mass of gas, but the surface volume; the surface density being divided out. This becomes important later. With a partitioned tracer you are allowed to nominate a second phase (in the TRACER keyword) into which the tracer may also dissolve. Note that for a partitioned surfactant it is necessary to do this for a tracer called 'SUR' which ECLIPSE recognizes as being the surfactant.

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.

Now, if we assume water was chosen as the 'solution' phase, the quantity conserved becomes $c_g V_g \rho_g + c_w V_w \rho_w$ where now both the inflow and outflow of gas and water have to be taken into account. However, the concentration of the tracer in the water (solution) phase is not independent of the concentration in the gas (free) phase; it is given by the relationship $\frac{c_w}{c_g} = K(P)$. The functions P versus K(P) are entered in tabular form with the TRACERKP keyword and are required for each grid block. The particular table to use is then specified by using the TRKPFxxx keyword in the REGIONS section. Again, if the partitioned tracer is a surfactant, the added tracer name should be 'SUR'. A table number to use within the wellbore is also required and is specified in item 5 of the TRACER keyword.

For an environmental tracer, adsorption, decay and diffusion are performed on the tracer in the 'solution' phase. For a surfactant, adsorption is applied to the water phase, that is the 'free' phase.

The partitioned tracer option must be enabled by entering the keyword PARTTRAC in the RUNSPEC section.

Summary of keywords

RUNSPEC section

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

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 functions K(P) for the partitioned tracers.
TRADS	Specify adsorption data.
TRDCY	Specify decay data.
TRDIF	Specify diffusion data.
TRROCK	Specify rock data for adsorption.

REGIONS section

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

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 12.2 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
FTIPT				RTIPT	BTIPT	Tracer In Solution
				RTFTS		Tracer Flow Total
FTADS				RTADS	BTADS	Tracer adsorption
FTDCY				RTDCY	BTDCY	Decayed tracer

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.

SCHEDULE section

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.

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    ================================
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
==============================================================
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    ============================================================
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
============================================================
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   ==========================================================
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
============================================================
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   ==========================================================

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 13

Two-parameter equations of state

x	ECLIPSE 100 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. 9] and [Ref. 10].

The generalized form of such an equation of state is:

$$Z^3 + E_2 Z^2 + E_1 Z + E_0 = 0 \qquad \text{[EQ 13.1]}$$

with:

$$E_2 = (m_1 + m_2 - 1)B - 1 \qquad \text{[EQ 13.2]}$$

$$E_1 = A - (m_1 + m_2 - m_1 m_2)B^2 - (m_1 + m_2)B$$

$$E_0 = -[AB + m_1 m_2 B^2 (B + 1)] \qquad \text{[EQ 13.3]}$$

The coefficients m_1 and m_2 depend upon the equation used. Refer to Table 13.1 for details.

Table 13.1 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:

$$\ln(f_i/(px_i)) = -\ln(Z-B) + \frac{A}{(m_1-m_2)B}\left(\frac{2\Sigma_i}{A} - \frac{B_i}{B}\right)\ln\left(\frac{Z+m_2B}{Z+m_1B}\right) + \frac{B_i}{B}(Z-1)$$

[EQ 13.4]

where:

$$\Sigma_i = \sum_j A_{ij}x_j$$

[EQ 13.5]

$$A = \sum_{j=1}^{n}\sum_{k=1}^{n} x_jx_kA_{jk}$$

[EQ 13.6]

$$B = \sum_{j=1}^{n} x_jB_j$$

[EQ 13.7]

$$A_{jk} = (1-\delta_{jk})(A_jA_k)^{1/2}$$

[EQ 13.8]

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:

$$A_j = \Omega_a(T,j)\frac{P_{rj}}{T_{rj}^2}$$

[EQ 13.9]

$$B_j = \Omega_b(T,j)\frac{P_{rj}}{T_{rj}} \qquad \text{[EQ 13.10]}$$

$\Omega_a(T,j)$ and $\Omega_b(T,j)$ 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,

$$\Omega_a(T,j) = A + B \cdot T^{1/2} + CT \qquad \text{[EQ 13.11]}$$

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 13.2.

Table 13.2 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:

$$f_{iL} = f_{iV} \quad \text{[EQ 13.12]}$$

As described above, the fugacities are functions of temperature, pressure and composition:

$$f_i = f_i(T, p, x_i) \quad \text{[EQ 13.13]}$$

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:

$$K_i = \frac{y_i}{x_i} \quad \text{[EQ 13.14]}$$

The mole fractions of each component in the liquid and vapor phases are given as:

$$x_i = \frac{z_i}{1 + (K_i - 1)V} \quad \text{[EQ 13.15]}$$

$$y_i = \frac{K_i z_i}{1 + (K_i - 1)V} \quad \text{[EQ 13.16]}$$

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. 11]. The viscosity μ for each phase is given by

$$[(\mu - \mu^\circ) \cdot \xi + 0.0001]^{1/4} = \sum_{i=1}^{5} a_i \cdot b_r^{\,i-1} \qquad \text{[EQ 13.17]}$$

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. 11] . These values can be modified by using the `LBCCOEF` keyword.

The reduced molar densities b_r are given by

$$b_r = \frac{b_p}{b_c} \qquad \text{[EQ 13.18]}$$

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

$$\frac{1}{b_c} = V_c = \sum x_i \cdot V_{c,i} \qquad \text{[EQ 13.19]}$$

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:

$$[(\mu - \mu^\circ) \cdot \xi + 0.0001] = \left(\sum_{i=1}^{5} a_i \cdot b_r^{\,i-1} \right)^4 \cdot \exp\left(\frac{b_r}{b_{max,r} - b_r} \right) \qquad \text{[EQ 13.20]}$$

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 13.17]. 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 $V_{min,i}$

$$\frac{1}{b_{max}} = V_{min} = \sum x_i \cdot V_{min,i} \quad \text{[EQ 13.21]}$$

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

$$b_p < b_{max} \quad \text{[EQ 13.22]}$$

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

$$V_{min,i} = \frac{1}{b_0} \cdot V_{c,i} \quad \text{[EQ 13.23]}$$

In this case the maximum reduced molar density in [EQ 13.20] will be a constant:

$$b_{max,r} = b_0 \quad \text{[EQ 13.24]}$$

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. 79] 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 $V_{\text{mol},\,p}$ is given by:

$$V_{\text{mol},\,p} = V_{\text{mol},\,p}^{EoS} - \sum_{i=1}^{N} z_i c_i \qquad \text{[EQ 13.25]}$$

where

$p = (\text{liquid, vapor})$ represents the phase of the system

$V_{\text{mol},\,p}^{EoS}$ 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:

$$s_i = \frac{c_i}{b_i} \qquad \text{[EQ 13.26]}$$

where:

$$b_i = \frac{\Omega_{b,\,i} R T_{ci}}{p_{ci}} \qquad \text{[EQ 13.27]}$$

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:

$$P = \frac{RT}{V-B} - \frac{A}{V(V+B)+B(V-B)} \qquad \text{[EQ 13.28]}$$

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 to determine 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 timestep 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:

$$K_i = \frac{\exp\left[5.37(1-\omega_i)\left(1-\frac{1}{T_{ri}}\right)\right]}{P_{ri}} \qquad \text{[EQ 13.29]}$$

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.

$$g(V) = \sum_{i=1}^{n_c} \frac{z_i(K_i-1)}{1+V(K_i-1)} = 0 \qquad \text{[EQ 13.30]}$$

ECLIPSE 300 then solves this equation for the mole fractions of each component in the liquid and vapor phases,

$$x_i = z_i/[1 + V(K_i - 1)],\ i = 1, 2, \ldots\ldots n_c \qquad \text{[EQ 13.31]}$$

$$y_i = (K_i z_i)/[1 + V(K_i - 1)],\ i = 1, 2, \ldots\ldots n_c \qquad \text{[EQ 13.32]}$$

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:

$$Z^3 - Z^2(1 - b) + Z(a - 3b^2 - 2b) - (ab - b^2 - b^3) = 0 \qquad \text{[EQ 13.33]}$$

where

$$a = \frac{AP}{R^2 T^2} \qquad \text{[EQ 13.34]}$$

$$b = \frac{BP}{RT} \qquad \text{[EQ 13.35]}$$

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),

$$\sum_{i=1}^{n_c} \left(\frac{f_{iL}}{f_{iV}} - 1 \right)^2 < \varepsilon . \qquad \text{[EQ 13.36]}$$

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:

$$K_i^{NEW} = K_i^{OLD} \cdot \left[\frac{f_{iL}}{f_{iV}} \right]. \qquad \text{[EQ 13.37]}$$

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. 12]. This is carried out anew at each timestep, 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. 13]. 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. 14].

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 14

Introduction

x	ECLIPSE 100 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 ten 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:

- 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:

- EOSS
- TCRITS
- PCRITS
- VCRITS or ZCRITS
- 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 15

Introduction

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

A number of files may be produced by ECLIPSE for each simulation run. These are:

	PRINT file	Main printer output	(RPT... keywords)
	DEBUG file	Debug output	(DEBUG 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)
	RSSPEC file	Restart index file	(RSSPEC keyword in RUNSPEC section)
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, may be controlled by the user with the keywords indicated. 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 is 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 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 16

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-timestep of the full field run.

The `FLUX` file is read during the reduced field run to generate the appropriate boundary conditions. Timesteps 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 16.1 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.

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. 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.

ECLIPSE 300

There is no restriction on the position of LGRs relative to flux boundaries in ECLIPSE 300.

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.

Current limitations

`DUMPFLUX` and `USEFLUX` cannot be used simultaneously in the same run.

ECLIPSE 100

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, the Solvent model and other special features involving additional conservation equations.

ECLIPSE 100

The pressure boundary conditions account for solvent, API and brine tracking.

Gas Field Operations Model

The Flux Boundary option should not be used in conjunction with automatic DCQ reduction in the Gas Field Operations model.

Summary of keywords

RUNSPEC section

	REGDIMS	Item 4 specifies the maximum number of flux regions in the run.

GRID section

	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.

Options keywords

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.

The Foam Model

Chapter 17

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 gas.

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, flow and collapse. In this model we assume the foam is transported with the gas phase, and hence we model the foam by a tracer in the gas phase that accounts for adsorption on to the rock 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 the gas phase. Hence the foam concentration can be thought of as the surfactant concentration existing in foam form.

The foam conservation equations are solved fully implicitly at the end of each time step after the oil, water and gas flows have been computed. The foam is assumed to exist only in the gas phase. Account is taken of the adsorption and decay of the foam. The modification of the gas mobility is treated explicitly, with the mobility modification being applied at the subsequent time step.

Adsorption

The adsorption of foam is assumed to be instantaneous, and the quantity adsorbed is a function of the active foam concentration. You are required to supply an adsorption isotherm as a function of foam concentration (see the FOAMADS keyword). The quantity of foam adsorbed on to the rock is given by:

$$\text{Mass of adsorbed foam} = V \cdot \left(\frac{1-\phi}{\phi}\right) \cdot \rho_r \cdot \mathrm{CA}(C_{\mathrm{foam}}) \qquad \text{[EQ 17.1]}$$

where

V	is the pore volume of the cell
ϕ	is the porosity
ρ_r	is the mass density of the rock (see the FOAMROCK keyword)
$\mathrm{CA}(C_{\mathrm{foam}})$	is the adsorption isotherm as a function of local foam concentration in solution.

There are two adsorption models which can be selected. The first model ensures that each grid block retraces the adsorption isotherm as the foam concentration falls in the cell. The second model assumes that the adsorbed foam concentration on the rock may not decrease with time and hence does not allow for any de-adsorption.

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; 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.

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; the modification associated with the conditions at the end of each time step are applied at the subsequent time step.

Unmodified gas flow: $F_g = \frac{K_{rg}}{\mu_g B_g}(T \cdot \mathrm{DP})$ [EQ 17.2]

Modified flow: $F_{gm} = M(C_{\mathrm{foam}})\frac{K_{rg}}{\mu_g B_g}(T \cdot \mathrm{DP}) = M(C_{\mathrm{foam}})F_g$ [EQ 17.3]

where

K_{rg}	is the gas relative permeability
μ_g	is the gas viscosity
B_g	is the gas formation volume factor
T	is the transmissibility
DP	is the potential difference
$M(C_{\mathrm{foam}})$	is the input mobility reduction factor
C_{foam}	is the foam concentration.

The mobility reduction factor $M(C_{\mathrm{foam}})$ can optionally be influenced by two separate effects, both of which will tend to increase M (that is to increase the gas mobility again). $M(C_{\mathrm{foam}})$ can be varied as a function of both pressure and shear rate.

The mobility reduction factor including the pressure effect is:

$$\mathrm{MP} = (1 - M(C_{\mathrm{foam}}))M_p(P) + M(C_{\mathrm{foam}})$$ [EQ 17.4]

where

MP	is the mobility reduction factor with the pressure effect
$M(C_{\mathrm{foam}})$	is the original reduction factor as a function of foam concentration
$M_p(P)$	is the pressure dependency function
P	is the oil pressure.

The mobility reduction factor including the shear effect is:

$$\mathrm{MF} = (1 - \mathrm{MP})M_s(v) + \mathrm{MP}$$ [EQ 17.5]

where

MF	is the final mobility reduction factor
MP	is the mobility reduction factor after applying the pressure effect
$M_s(v)$	is the shear dependency function
v	is the gas velocity.

The gas velocity is calculated as:

$$v = B_g\left(\frac{F_g}{\phi \cdot A}\right) \qquad \text{[EQ 17.6]}$$

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.

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.

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 5 optional keywords which, if present, invoke the adsorption, decay and additional mobility functionality features of the model.

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)

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 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:

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 the gas mobility reduction includes the pressure dependent effect but **not** the shear effect, as the shear effect is calculated for each flow separately.
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.

Summary of keywords

RUNSPEC section

FOAM	Activates the Foam model

GRID section

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.

PROPS section

FOAMADS	Foam adsorption data.
FOAMDCYO	Foam decay data (as a function of oil saturation).
FOAMDCYW	Foam decay data (as a function of water saturation).
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.
FOAMROCK	Foam-rock properties.
RPTPROPS	Controls output from the PROPS section The FOAM mnemonic outputs the foam properties.

SUMMARY section

The keywords shown in Table 17.1 control output of data specific to the Foam model.

Table 17.1 Foam SUMMARY output keywords

Field	Group	Well	Connection	Region	Block	Information
			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
FTIPTFOA				RTIPTFOA	BTIPTFOA	In Solution

Table 17.1 Foam SUMMARY output keywords (Continued)

Field	Group	Well	Connection	Region	Block	Information
				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

RPTSOL

Controls output from the SOLUTION section.
The FIPFOAM mnemonic outputs the foam fluid in place reports.
The FOAM mnemonic outputs grid block foam concentration.
The FOAMADS mnemonic outputs grid block foam adsorption.
The FOAMDCY mnemonic outputs grid block foam decay.
The FOAMMOB mnemonic outputs the gas mobility reduction factor.

Note that the output of the gas mobility reduction factor 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

RPTSCHED

Controls output from the SCHEDULE section.
The FIPFOAM mnemonic outputs the foam fluid in place reports.
The FOAM mnemonic outputs grid block foam concentration.
The FOAMADS mnemonic outputs grid block foam adsorption.
The FOAMDCY mnemonic outputs grid block foam decay.
The FOAMMOB mnemonic outputs the gas mobility reduction factor.

Note that the output of the gas mobility reduction factor will include the effects of pressure dependence (FOAMMOBP) but not the shear effect (FOAMMOBS), as the shear effect is calculated for each flow separately.

WFOAM

Foam concentration for injection wells.

Example data set

RUNSPEC

```
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 DATA
-- 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 ============================================================
--
--  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  VISCOSIBILITY
--
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
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'
'FIPFOAM=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 18

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 timesteps 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 and `IMPLICIT` for all black oil runs. The default solution procedure may be changed by using one of the keywords `AIM`, `IMPLICIT` 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 timestep is:

$$R_{fl} = \frac{dM}{dt} + F + Q \qquad \text{[EQ 18.1]}$$

where

dM	is the mass, per unit surface density, accumulated during the current timestep, dt
F	is the net flow rate into neighboring grid blocks
Q	is the net flow rate into wells during the timestep
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:

$$R_{fu} = f_l - f_v = 0 \qquad \text{[EQ 18.2]}$$

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;

$$R = \begin{bmatrix} R_o \\ R_w \\ R_g \end{bmatrix} \qquad X = \begin{bmatrix} P_o \\ S_w \\ S_g \text{ or } R_s \text{ or } R_v \end{bmatrix} \qquad \text{[EQ 18.3]}$$

and the Jacobian, $J = \frac{dR}{dX}$, takes the form

$$\frac{dR_i}{dX_j} = \begin{bmatrix} \frac{dR_o}{dP_o} & \frac{dR_o}{dS_w} & \frac{dR_o}{dS_g} \\ \frac{dR_w}{dP_o} & \frac{dR_w}{dS_w} & \frac{dR_w}{dS_g} \\ \frac{dR_g}{dP_o} & \frac{dR_g}{dS_w} & \frac{dR_g}{dS_g} \end{bmatrix}_{ij} \qquad \text{[EQ 18.4]}$$

The mass change during the timestep, dt, is then

$$dM = M_{t+dt} - M_t \qquad \text{[EQ 18.5]}$$

with

$$M = \mathrm{PV}\begin{bmatrix} \dfrac{S_o}{B_o} + \dfrac{R_v S_g}{B_g} \\ \dfrac{S_w}{B_w} \\ \dfrac{S_g}{B_g} + \dfrac{R_s S_o}{B_o} \end{bmatrix} \qquad \text{[EQ 18.6]}$$

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 variables are	$R_s = 0,\ S_g = 1 - S_w,$ $P_o,\ S_w,\ R_v.$
State 2	Gas & oil, variables are	$R_v = R_{v\,\mathrm{sat}},\ R_s = R_{s\,\mathrm{sat}},$ $P_o,\ S_w,\ S_g$
State 3	Oil only, variables are	$R_v = 0,\ S_g = 0,$ $P_o,\ S_w,\ R_s.$

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.

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

$$R_m = PV - FV = 0 \qquad \text{[EQ 18.7]}$$

where PV is the cell pore volume, and FV is the fluid volume, given by

$$FV = PV\left(\frac{m_t L}{b_o^m} + \frac{m_t V}{b_g^m} + \frac{m_w}{b_w^m}\right) \qquad \text{[EQ 18.8]}$$

with the total hydrocarbon molar density given by

$$m_t = \sum_c m_c \qquad \text{[EQ 18.9]}$$

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

$$J = \begin{array}{l} \\ \text{Water} \\ \text{Comp1} \\ \ldots \\ \text{Comp}N_c \\ \text{Volumebal} \end{array} \begin{array}{c} P \;\; S_w \;\; m_1 \;\; \ldots \;\; m_N \\ \begin{bmatrix} x & x & x & \ldots & x \\ x & x & x & \ldots & x \\ \ldots & \ldots & \ldots & \ldots & \ldots \\ x & x & x & \ldots & x \\ x & x & x & \ldots & x \end{bmatrix} \end{array} \qquad \text{[EQ 18.10]}$$

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:

$$R_{fu} = J - H_t + H_v \qquad \text{[EQ 18.11]}$$

where

$$H_l = \ln\left(\frac{f_l}{Px}\right) \qquad \text{[EQ 18.12]}$$

$$H_v = \ln\left(\frac{f_v}{Py}\right) \qquad \text{[EQ 18.13]}$$

An additional normalization residual (the Rachford-Rice equation) exists in the compositional case,

$$R_n = \sum_i \left(z_i \frac{(K_i - 1)}{1 + V(K_i - 1)} \right) - 1 = 0 \qquad \text{[EQ 18.14]}$$

where z_i and K_i are component mole fractions and K-values respectively. These solution variables for the flash calculation are $X_f = (J_1, \ldots, J_N, V)$.

The flash solution X_f is found at each iteration by solving the equilibrium condition that

$$R_{fu}(X_f) = 0. \qquad \text{[EQ 18.15]}$$

In `IMPSAT` mode we have three additional residuals to define the saturation variables X_o, X_g, X_w:

$$R_p = PV(X - S_p(P, m)) = 0 \qquad \text{[EQ 18.16]}$$

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 timestep.

$$R = \frac{M_{t+dt} - M_t}{dt} + F(P_{t+dt}, S_t) + Q(P_{t+dt}, S_t) \qquad \text{[EQ 18.17]}$$

The mass terms M_{t+dt} are evaluated using both pressures and saturations at the end of the timestep. 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 timestep. 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

Timesteps are chosen automatically to limit saturation changes to 5%. However a converged timestep 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

Timesteps are chosen automatically to limit saturation changes to 5%. The default timestep 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 timestep. The flow terms between cells are evaluated assuming the saturations, generalized mobilities and reservoir density terms are all fixed at the previous timestep.

ECLIPSE 300

AIM

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 timesteps, 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

This option is based on the paper "IMPES Stability: The Stable Step" by K.H. Coats [Ref. 86]. 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, timesteps 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 timestep 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 timesteps 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 timesteps 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 timesteps.

Instability in IMPES

The most common instability using IMPES arises when a phase crosses the critical saturation and becomes mobile. In Figure 18.1 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 timestep, *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 timestep, 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 timestep. 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 18.1 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

$$\sum_i (R_o)_i = \sum_i \left(\frac{dM_o}{dt}\right)_i + \sum_i (Q_o)_i$$

$$\sum_i (R_w)_i = \sum_i \left(\frac{dM_w}{dt}\right)_i + \sum_i (Q_w)_i \qquad \text{[EQ 18.18]}$$

$$\sum_i (R_g)_i = \sum_i \left(\frac{dM_g}{dt}\right)_i + \sum_i (Q_g)_i$$

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:

$$\mathrm{MB}_o = \overline{B_o} dt \left\{ \left(\sum_i (R_o)_i \right) / \left(\sum_i (\mathrm{PV})_i \right) \right\}$$

$$\mathrm{MB}_w = \overline{B_w} dt \left\{ \left(\sum_i (R_w)_i \right) / \left(\sum_i (\mathrm{PV})_i \right) \right\} \qquad \text{[EQ 18.19]}$$

$$\mathrm{MB}_g = \overline{B_g} dt \left\{ \left(\sum_i (R_g)_i \right) / \left(\sum_i (\mathrm{PV})_i \right) \right\}$$

where $\overline{B_o}$ is the average oil formation factor etc.

The numerical values of MB_o, MB_w and 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 $1.0\mathrm{E}\text{-}7$. 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

$$\mathrm{CNV}_o = \overline{B_o} dt \cdot \mathrm{MAX}_i \left| \frac{(R_o)_i}{(\mathrm{PV})_i} \right|$$

$$\mathrm{CNV}_w = \overline{B_w} dt \cdot \mathrm{MAX}_i \left| \frac{(R_w)_i}{(\mathrm{PV})_i} \right|$$

$$\mathrm{CNV}_g = \overline{B_g} dt \cdot \mathrm{MAX}_i \left| \frac{(R_g)_i}{(\mathrm{PV})_i} \right| \qquad \text{[EQ 18.20]}$$

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:

$$\Delta S' = \frac{du^T}{dm_c} \cdot \Delta m_c \qquad \text{[EQ 18.21]}$$

where

$$u^T = \frac{m_t L}{b_o^m} + \frac{m_t V}{b_g^m} + \frac{m_w}{b_w^m} \qquad \text{[EQ 18.22]}$$

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 timestep 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 tightened down the convergence.

ECLIPSE 100 Black Oil

The flow rate into cell i from a neighboring cell n, F_{ni}, is

$$F_{ni} = T_{ni}\begin{bmatrix} \frac{k_{ro}}{B_o\mu_o} & 0 & \frac{R_v k_{rg}}{B_g\mu_g} \\ 0 & \frac{k_{rw}}{B_w\mu_w} & 0 \\ \frac{R_s k_{ro}}{B_o\mu_o} & 0 & \frac{k_{rg}}{B_g\mu_g} \end{bmatrix}_u \times \begin{bmatrix} dP_{oni} \\ dP_{wni} \\ dP_{gni} \end{bmatrix} \quad \text{[EQ 18.23]}$$

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

$$Q_i = -T_{wi}(P_{oi} - H_{iw} - P_{bh}) \begin{bmatrix} \frac{k_{ro}}{B_o \mu_o} + \frac{R_v k_{rg}}{B_g \mu_g} \\ \frac{k_{rw}}{B_w \mu_w} \\ \frac{k_{rg}}{B_g \mu_g} + \frac{R_s k_{ro}}{B_o \mu_o} \end{bmatrix}_i \qquad \text{[EQ 18.24]}$$

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 964.

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^c_{pni}, is

$$F^c_{pni} = T_{ni} M^c_p \, dP_{pni} \qquad \text{[EQ 18.25]}$$

where

M^c_p	is the generalized mobility of component c in phase p and is given by

$$M^c_p = x^c_p \, k_{rp}(S_p) \frac{b^m_p}{\mu_p} \qquad \text{[EQ 18.26]}$$

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

$$dP_{pni} = P_{pn} - P_{pi} - \rho_{pni} G(D_n - D_i) \quad \text{[EQ 18.27]}$$

or

$$dP_{pni} = P_n - P_i + P_{cpn} - P_{cpi} - \rho_{pni} G(D_n - D_i) \quad \text{[EQ 18.28]}$$

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

$$M_o^c = x_c k_{ro}(S_w, S_g) \frac{b_o^m}{\mu_o} \quad \text{[EQ 18.29]}$$

and

$$M_g^c = y_c k_{rg}(S_g)\frac{b_g^m}{\mu_g} \quad \text{[EQ 18.30]}$$

The rate of flow of component c embedded in phase p into a production well from cell i is

$$Q_{pi}^c = -T_{wi}(P_{pi} - H_{iw} - P_{bh})M_{pi}^c \quad \text{[EQ 18.31]}$$

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 964 for further information.

Reservoir densities

ECLIPSE 100

In the black oil case, reservoir densities (ρ) of oil and gas are computed from surface densities (ρ_s) using

$$\rho_o = (\rho_{so} + CR_s\rho_{sg})/B_o$$
$$\rho_g = (C\rho_{sg} + R_v\rho_{so})/B_g \qquad \text{[EQ 18.32]}$$

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

$$\rho_{oni} = (\rho_{on} + \rho_{oi})/2$$
$$\rho_{gni} = (\rho_{gn} + \rho_{gi})/2 \qquad \text{[EQ 18.33]}$$

ECLIPSE 300

In the compositional case, the reservoir densities are obtained from

$$\rho_p = MW_p b_p^m \qquad \text{[EQ 18.34]}$$

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

$$\rho_{pni} = \frac{(\rho_{pn}S_{pn} + \rho_{pi}S_{pi})}{(S_{pn} + S_{pi})} \qquad \text{[EQ 18.35]}$$

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

$$J \cdot \Delta x = R(X) \qquad \text{[EQ 18.36]}$$

or

$$\Delta x = J^{-1} R(X) \qquad \text{[EQ 18.37]}$$

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

$$\Delta x = J^{-1} \cdot R'_{fl} \qquad \text{[EQ 18.38]}$$

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

$$J'_D = \begin{array}{l} \\ \text{Water} \\ \text{Comp1} \\ \dots \\ \text{Comp}N_c \\ \text{Volumebal} \end{array} \begin{array}{c} P \;\; S_w \;\; m_1 \;\; \dots \;\; m_N \\ \begin{bmatrix} x & x & 0 & \dots & 0 \\ x & 0 & x & \dots & 0 \\ \dots & \dots & \dots & \dots & \dots \\ x & 0 & 0 & \dots & x \\ x & x & x & \dots & x \end{bmatrix} \end{array} \qquad \text{[EQ 18.39]}$$

$$J'_B = \begin{array}{l} \\ \text{Water} \\ \text{Comp1} \\ \dots \\ \text{Comp}N_c \\ \text{Volumebal} \end{array} \begin{array}{c} P \;\; S_w \;\; m_1 \;\; \dots \;\; m_N \\ \begin{bmatrix} x & 0 & 0 & \dots & 0 \\ x & 0 & 0 & \dots & 0 \\ \dots & \dots & \dots & \dots & \dots \\ x & 0 & 0 & \dots & 0 \\ 0 & 0 & 0 & \dots & 0 \end{bmatrix} \end{array} \qquad \text{[EQ 18.40]}$$

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, $\frac{dR'}{dP}^m$. This is a banded pressure equation, which may be solved using the nested factorization linear solver (see "Solution of the Linear Equations" on page 744).

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

$$J'_D = \begin{array}{l} \\ \text{Water} \\ \text{Oil} \\ \text{Gas} \end{array} \begin{array}{c} \begin{array}{ccc} P & S_w & X_3 \end{array} \\ \begin{bmatrix} x & x & 0 \\ x & 0 & x \\ x & 0 & x \end{bmatrix} \end{array} \qquad \text{[EQ 18.41]}$$

$$J'_B = \begin{array}{l} \\ \text{Water} \\ \text{Oil} \\ \text{Gas} \end{array} \begin{array}{c} \begin{array}{ccc} P & S_w & X_3 \end{array} \\ \begin{bmatrix} x & 0 & 0 \\ x & 0 & 0 \\ x & 0 & 0 \end{bmatrix} \end{array} \qquad \text{[EQ 18.42]}$$

Gas Field Operations Model

Chapter 19

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.

ECLIPSE 100

- Automatic compressor operation.

  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 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:

$$ACQ = 365.25 \times DCQ \qquad \text{[EQ 19.1]}$$

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 timestep, that is, at the end of the previous timestep. Therefore it is compared against the DCQ × swing factor for the previous timestep.)

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, **minus** any gas injection. 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 260).

ECLIPSE 300

In ECLIPSE 300 the sales gas rate is defined as the **gross** gas production from the formation, **minus** the gas consumption, **minus** any gas injection. 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 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 254) 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.

ECLIPSE 100

Automatic compressors

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 538). 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`.

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.

ECLIPSE 100

Automatic compressors with multiple contract groups

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 19.1 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 nodal PRessure in network
x			GPRDC		Group nodal PRessure in network at Delivery Capacity
x	x			NGOPAS	Number of iterations to converge DCQ in first pass

Remember that the mnemonics FGPR, GGPR, FGPT, GGPT refer to the **gross** gas produced from the formation, without allowing for consumption.

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 timestep. 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 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 timestep 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    Conv
-- years     DCQ      req    t/steps    reduc     reduc    DCQ its   tol
      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 19.1 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 19.1 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 207) 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

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 19.2 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

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.

Simulator performance keyword

NGOPAS	Outputs the number of iterations used to converge the DCQ in the first pass of the contract period.

SCHEDULE section

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 552), 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. 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.
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 • Automatic compressor switching • Control of sales gas rate instead of gross production rate.
ECLIPSE 100		

Gas Lift Optimization

Chapter 20

Introduction

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL

The Gas Lift Optimization facility is a Special Extension to ECLIPSE 100; it is not available in 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:

- 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?

- 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?

- 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 20.1). 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 20.1 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 was increased by one increment, and also what it would produce if the rate was reduced by one increment. The production rates are adjusted if necessary to obey any rate or BHP limits the well may have. From this information, ECLIPSE calculates the well's "weighted incremental gradient" (that is the increase in oil rate when an increment is added, multiplied by the well's weighting factor, divided by the size of the lift gas injection rate increment), and its "weighted decremental gradient" (that is the decrease in oil rate when an increment is subtracted, multiplied by the well's weighting factor, divided by the increment size). 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. 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

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 538). 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 20.2). 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 20.2 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.

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.

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.

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 unnecessary 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

Gas lift optimization can be used in combination with the group production rules (see "Group Production Rules" on page 383 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

When there is a production network (see "The Network Option" on page 538), 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 increment size) can also be written to the SUMMARY file by using the mnemonic WOGLR (Well Oil Gas Lift Ratio).

Restrictions

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.

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 20.1 Gas lift optimization SUMMARY output controls

Field	Group	Well	Information
FGLIR	GGLIR	WGLIR	Gas Lift Injection Rate
		WOGLR	Well Oil Gas Lift Ratio (that is the incremental gradient of the field's production rate with respect to the addition of another gas lift increment to that well)

SCHEDULE section

GLIFTOPT	Sets limits on group lift gas supply.
LIFTOPT	Activates Gas Lift Optimization and sets some governing parameters.
VFPPROD	Inputs VFP tables.
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 21

Gas plant tables

x	ECLIPSE 100 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:

$$m_{o_i} = R_{o_i} z_i \qquad \text{[EQ 21.1]}$$

$$m_{\mathrm{NGL}_i} = R_{\mathrm{NGL}_i} z_i \qquad \text{[EQ 21.2]}$$

$$m_{g_i} = R_{g_i} z_i \quad \text{[EQ 21.3]}$$

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:

$$x_i = \frac{m_{o_i}}{\Sigma m_{o_i}} \quad \text{[EQ 21.4]}$$

$$n_i = \frac{m_{\mathrm{NGL}_i}}{\Sigma m_{\mathrm{NGL}_i}} \quad \text{[EQ 21.5]}$$

$$y_i = \frac{m_{g_i}}{\Sigma m_{g_i}} \quad \text{[EQ 21.6]}$$

The molar output rates of oil, NGLs and gas per mole of feed are given by:

$$L = \sum_i m_{o_i} \quad \text{[EQ 21.7]}$$

$$N = \sum_i m_{\mathrm{NGL}_i} \quad \text{[EQ 21.8]}$$

$$V = \sum_i m_{g_i} \quad \text{[EQ 21.9]}$$

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:

$$V_{mix} = \sum_i n_i V_i \quad \text{[EQ 21.10]}$$

The partial molar volumes are obtained from the density values entered with keyword DNGL, using

$$V_i = \frac{Mw_i}{\rho_i} \quad \text{[EQ 21.11]}$$

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 21.1 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

Table 21.1 Keywords to obtain reports of recovered NGLs and gas (Continued)

Keyword	Description
FGRT	gas remaining volume total
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'.

The GASWAT Option

Chapter 22

Introduction

x	ECLIPSE 100 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.

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 keyword. 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.

Equation of state modifications

The conventional Peng-Robinson equation of state

The A -coefficient in the Peng-Robinson equation of state is given by:

$$A = \sum_{jk} x_j x_k A_{jk} \quad \text{[EQ 22.1]}$$

where x are component mole fractions and

$$A_{jk} = (1 - k_{jk})(A_j A_k)^{1/2} \quad \text{[EQ 22.2]}$$

k_{jk} are binary interaction coefficients. A_{jk} is defined as:

$$A_j = \frac{\Omega_j P_{rj}}{T_{rj}^2} \quad \text{[EQ 22.3]}$$

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:

$$A_j = \frac{A_j^r \cdot P}{T^2} \quad \text{[EQ 22.4]}$$

The Peng-Robinson equation of state then defines A_j^r as:

$$A_j^r = \frac{\Omega_{oj} T_{cj}^2 \alpha_j}{P_{cj}} \quad \text{[EQ 22.5]}$$

with

$$\alpha_j^{1/2} = 1 + f(w_j)(1 - T_{rj}^{1/2}) \quad \text{[EQ 22.6]}$$

$f(w_j)$ is a polynomial in the acentric factor, given by:

$$f(w_j) = 0.37464 + 1.54226 w_j - 0.26996 w_j^2 \quad \text{[EQ 22.7]}$$

To account for the temperature dependence in this expression we write it in the form:

$$A_j^r = \frac{\Omega_{oj} T_{cj}^2 (p_j + q_j T^{1/2})^2}{P_{cj}} \quad \text{[EQ 22.8]}$$

where $p_j = 1 + f(w_j)$ and $q_j = -f(w_j)/T_c^{1/2}$.

Then:

$$A_{rjk} = [1 - k_{jk}]\left[\frac{\Omega_{oj}T_{cj}^2}{P_{cj}}\right]^{1/2}\left[\frac{\Omega_{ok}T_{ck}^2}{P_{ck}}\right]^{1/2}[p_j p_k + (p_j q_k + p_k q_j)T^{1/2} + q_j q_k T]$$

[EQ 22.9]

In ECLIPSE 300 we store the three coefficients A_{1jk}, A_{2jk} and A_{3jk}, where:

$$A_{1jk} = (1 - k_{jk})\left(\frac{\Omega_{oj}T_{cj}^2}{P_{cj}}\right)^{1/2}\left(\frac{\Omega_{ok}T_{ck}^2}{P_{ck}}\right)^{1/2}(p_j p_k)$$

[EQ 22.10]

$$A_{2jk} = (1 - k_{jk})\left(\frac{\Omega_{oj}T_{cj}^2}{P_{cj}}\right)^{1/2}\left(\frac{\Omega_{ok}T_{ck}^2}{P_{ck}}\right)^{1/2}(p_j q_k + p_k q_j)$$

[EQ 22.11]

$$A_{3jk} = (1 - k_{jk})\left(\frac{\Omega_{oj}T_{cj}^2}{P_{cj}}\right)^{1/2}\left(\frac{\Omega_{ok}T_{ck}^2}{P_{ck}}\right)^{1/2}[q_j q_k]$$

[EQ 22.12]

This enables the equation of state coefficient for a cell of temperature T to be simply assembled as:

$$A_{jk} = (A_{1jk} + A_{2jk}T^{1/2} + A_{3jk}T)\frac{P}{T^2}$$

[EQ 22.13]

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:

  $$\alpha^{1/2} = 1 + aq_1(1 - aq_2 T_r) + aq_3(T_r^{-3} - 1)$$

  [EQ 22.14]

  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:

$$\alpha^{1/2} = p + qTr^{1/2} + rTr + sTr^{(-3)} \quad \text{[EQ 22.15]}$$

- 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:

$$k_{jw}^{a} = bq_1 + bq_2 T_{rj} + bq_3 T_{rj}^2 \quad \text{[EQ 22.16]}$$

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:

$$k_{N_2 w}^{a} = -1.70235(1 + 0.025587 c_s^{3/4}) + 0.44338(1 + 0.08126 c_s^{3/4}) T_{rN_2}$$

[EQ 22.17]

$$k^a_{CO_2w} = -0.31092(1 + 0.15587c_s^{0.7505}) + (0.23580)(1 + 0.17837c_s^{0.979})T_{rCO_2} - 21.2566\exp(-6.7222T_{rCO_2} - c_s)$$ [EQ 22.18]

$$k^a_{H_2Sw} = -0.20441 + 0.23426T_{rH_2S}$$ [EQ 22.19]

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, 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 keyword. 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:

$$c_s^{ppm} = A \cdot c_s / (1 + B \cdot c_s) \quad \text{[EQ 22.20]}$$

$$c_s = c_s^{ppm} / (1 - c_s^{ppm}) \quad \text{[EQ 22.21]}$$

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 be entered using the `SALINITY` keyword in the `PROPS` section. If this 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:

WEPR	Well energy production rate
WEPT	Well energy production rate
WEIR	Well energy injection rate
WEIT	Well energy injection rate
WGQ	Well gas quality
GEPR	Group energy production rate
GEPT	Group energy production rate
GEIR	Group energy injection rate
GEIT	Group energy injection rate
GGQ	Group gas quality
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.

Gas Calorific Value Control

Chapter 23

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 291) 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 timestep 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

$$Q_{pG} = \sum_{W \in i} Q_{pW} + M_G \sum_{W \in g^+} G_{pW} + M_G \alpha \sum_{W \in g^-} G_{pW} \qquad \text{[EQ 23.1]}$$

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 23.1] by setting

$$Q_{pG} = Q_{pG}^{\text{targ}} \qquad \text{[EQ 23.2]}$$

where

Q_{pG}^{targ}	is group G's target production rate of the controlled phase p.

This gives

$$M_G = \frac{Q_{pG}^{\text{targ}} - \sum_{W \in i} Q_{pW}}{\sum_{W \in g^+} G_{pW} + \alpha \sum_{W \in g^-} G_{pW}} \qquad \text{[EQ 23.3]}$$

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,

$$\frac{\sum_{W} C_{gW} Q_{gW}}{\sum_{W} Q_{gW}} = C_{gG}^{\text{targ}} \qquad \text{[EQ 23.4]}$$

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,

$$Q_{gW} = M_G G_{pW} (Q_g/Q_p)_W \quad \text{for } (W \in g^+)$$

$$Q_{gW} = \alpha M_G G_{pW} (Q_g/Q_p)_W \quad \text{for } (W \in g^-) \qquad \text{[EQ 23.5]}$$

where

$(Q_g/Q_p)_W$	is the well's latest known phase production ratio of gas to the controlled phase p.

Substituting [EQ 23.5] into [EQ 23.4] yields

$$M_G \sum_{W \in g^+} (C_{gW} - C_{gG}^{\text{targ}})\, G_{pW}\, (Q_g/Q_p)_W - \alpha M_G \sum_{W \in g^-} (C_{gG}^{\text{targ}} - C_{gW}) G_{pW} (Q_g/Q_p)_W = \sum_{W \in i} (C_{gG}^{\text{targ}} - C_{gW}) Q_{gW} \qquad \text{[EQ 23.6]}$$

Simultaneous solution of [EQ 23.1] and [EQ 23.6] yields the guide rate adjustment factor α

$$\alpha = \frac{a \sum_{W \in g^+} (C_{gW} - C_{gG}^{\text{targ}}) G_{pW} \left(\frac{Q_g}{Q_p}\right)_W - b \sum_{W \in g^+} G_{pW}}{a \sum_{W \in g^-} (C_{gG}^{\text{targ}} - C_{gW}) G_{pW} \left(\frac{Q_g}{Q_p}\right)_W + b \sum_{W \in g^-} G_{pW}} \qquad \text{[EQ 23.7]}$$

where:

$$a = Q_{pG}^{\text{targ}} - \sum_{W \in i} Q_{pW}$$

$$b = \sum_{W \in i} (C_{gG}^{\text{targ}} - C_{gW}) Q_{gW}$$

[EQ 23.8]

Once α has been obtained from [EQ 23.7], the group's guide rate multiplier M_G can be obtained from [EQ 23.3]. 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 23.7] is zero or negative, there are two actions that ECLIPSE may take:

- 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.

- 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 23.7] is zero or negative, there are two actions that ECLIPSE may take:

- 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 $\Delta Q_{gW}(C_{gG}^{\text{targ}} - C_{gW})$.

- 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:

- 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.

- 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

$$E_G = \sum_{W \in i} E_W + M_G \sum_{W \in g^+} G_{eW} + M_G \, \alpha \sum_{W \in g^-} G_{eW} \qquad \text{[EQ 23.9]}$$

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 23.9] 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

$$M_G = \frac{E_G^{\text{targ}} - \sum_{W \in i} E_W}{\sum_{W \in g} G_{eW}} \qquad \text{[EQ 23.10]}$$

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

$$M_G = \frac{E_G^{\text{targ}} - \sum_{W \in i} E_W}{\sum_{W \in g^+} G_{eW} + \alpha \sum_{W \in g^-} G_{eW}} \qquad \text{[EQ 23.11]}$$

When the controlled quantity is energy, [EQ 23.6] becomes

$$M_G \sum_{W \in g^+} \left(1 - \frac{C_{gG}^{\text{t arg}}}{C_{gW}}\right) G_{eW} - \alpha M_G \sum_{W \in g^-} \left(\frac{C_{gG}^{\text{targ}}}{C_{gW}} - 1\right) G_{eW} \qquad \text{[EQ 23.12]}$$

$$= \sum_{W \in i} (C_{gG}^{\text{targ}} - C_{gW}) Q_{gW}$$

Simultaneous solution of [EQ 23.9] and [EQ 23.12] gives the guide rate adjustment factor required to meet the target mean calorific value

$$\alpha = \frac{e \sum_{W \in g^+} \left(1 - \frac{C_{gG}^{\text{t arg}}}{C_{gW}}\right) G_{eW} - b \sum_{W \in g^+} G_{eW}}{e \sum_{W \in g^-} \left(\frac{C_{gG}^{\text{targ}}}{C_{gW}} - 1\right) G_{eW} + b \sum_{W \in g^-} G_{eW}} \qquad \text{[EQ 23.13]}$$

where:

$$e = E_G^{\text{targ}} - \sum_{W \in i} E_W$$

$$b = \sum_{W \in i} (C_{gG}^{\text{targ}} - C_{gW}) Q_{gW} \qquad \text{[EQ 23.14]}$$

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 305 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 907 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 timestep 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}

$$\beta_W = (1 + AC_{gW})^B \qquad \text{[EQ 23.15]}$$

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 251). 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:

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 653), 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 23.1 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

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.
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 24

Introduction

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL

The Geomechanics option provides improved ability to simulate compaction processes that are not available with the rock compaction option (see "Rock Compaction" on page 687 of this manual). The benefits of coupling the geomechanical rock stress calculation with the fluid flow include:

- Improved ability to predict subsidence and compaction in a reservoir.
- Improved modeling of fluid flow through stress dependent permeability and Biot's constant.
- Flexibility
  - Stress and velocity (rigid) boundary conditions can be set on external as well as internal surfaces.
  - You do not set the grid. The grid for the stress calculation is the same grid that is used when calculating fluid flow.
  - Rock mechanical properties are input on a grid by grid basis.
- 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.

There are two techniques to calculated the coupled stress. Either finite element or finite difference methods can be used. The finite element method retains second order accuracy when the grid is skewed which can arise, for example, when using Corner-Point geometry. However, it may be slower to calculate than the finite difference option which is only accurate when the gridding is Cartesian or orthogonal Corner-Point. Most of the simulator technology is available such as the ability to run in black oil, compositional or thermal modes and the full use of well model features.

A maximum number of 50,000 grid cells has been set for the Geomechanics option. For this release, there are a number of important restrictions in both the finite element and finite difference method options. Radial gridding is not allowed. NNC connections are not supported which prohibits faults and pinch-outs as well. There is no ability to run parallel calculations. Dual porosity is not available. These are discussed in the section "Restrictions" on page 330 of this manual.

A plasticity model is available with the finite element calculation. Either Mohr-Coulomb or Drucker-Praeger failure criteria can be chosen, with or without hardening.

Implementation of geomechanics in ECLIPSE 300

Previous Work

A preliminary publication that outlines some of the basic ideas discussed below can be found in [Ref. 85] of the bibliography. A more recent publication can be found in [Ref. 95].

Elastic Stress Equations

Steady state rock momentum balance equations in the x, y and z directions can be written as:

$$\frac{\partial \sigma_x}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} + F_x = 0$$

$$\frac{\partial \sigma_y}{\partial y} + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{zy}}{\partial z} + F_y = 0 \qquad \text{[EQ 24.1]}$$

$$\frac{\partial \sigma_z}{\partial z} + \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + F_z = 0$$

Here $\vec{F}$ is the body force. The z-direction is the direction of the gravity body force.

The elastic normal stresses σ and shear stresses τ can be expressed in terms of strains, ε and γ, as:

$$\sigma_x = 2G\varepsilon_x + \lambda(\varepsilon_x + \varepsilon_y + \varepsilon_z) - \alpha P$$

$$\sigma_y = 2G\varepsilon_y + \lambda(\varepsilon_x + \varepsilon_y + \varepsilon_z) - \alpha P$$

$$\sigma_z = 2G\varepsilon_z + \lambda(\varepsilon_x + \varepsilon_y + \varepsilon_z) - \alpha P \qquad \text{[EQ 24.2]}$$

$$\tau_{xy} = G\gamma_{xy}$$

$$\tau_{yz} = G\gamma_{yz}$$

$$\tau_{zx} = G\gamma_{zx}$$

Note that stresses as defined in equation [EQ 24.2] include the pore pressure and Biot constant and are, therefore, total stresses.

Constants G, also known as the modulus of rigidity, and λ are Lamé's constants. They are functions of Young's modulus, E, and Poisson's ratio, ν. In particular, $G = \frac{E}{2(1+\nu)}$, $\lambda = \frac{E\nu}{(1-2\nu)(1+\nu)}$. Strains $\varepsilon_{x,y,z}$ are defined in terms of displacements in the x, y, z directions, namely u, v and w. Thus

$$\varepsilon_x = \frac{\partial u}{\partial x}$$

$$\varepsilon_y = \frac{\partial v}{\partial y}$$

$$\varepsilon_z = \frac{\partial w}{\partial z}$$

$$\gamma_{xy} = \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} \qquad \text{[EQ 24.3]}$$

$$\gamma_{yz} = \frac{\partial v}{\partial z} + \frac{\partial w}{\partial y}$$

$$\gamma_{zx} = \frac{\partial w}{\partial x} + \frac{\partial u}{\partial z}$$

Within the simulator, the rock force is calculated in units of Newtons, lbf, or dynes depending on whether you have chosen metric, field or lab units for the simulation.

Gridding

Before returning to a discussion of the fluid component mass conservation and rock mass conservation equations, gridding for the finite difference and finite element methods of calculating the rock force (momentum) balance equations is briefly outlined.

Firstly, the finite difference method of solving the above equations is discussed. Separate control volumes for balancing the force are used in each orthogonal grid direction. These control volumes are staggered in each of the coordinate directions. In Figure 24.1, x-, y- and z-direction rock force balance control volumes are shown.

Figure 24.1 Control volumes (light grey grid) for rock force balance

Also shown in Figure 24.1 is the finite difference discretization used. The grid with the darker black lines in this figure is the standard reservoir simulation grid, which are control volumes for fluid component and rock mass conservation. Rock displacements u, v and w are defined on the faces of this grid. Rock momentum control volumes in a particular coordinate direction are centred on the rock displacement in that direction.

We have found the staggered gridding to be the most natural for differencing purposes and for setting of boundary conditions. It is also the most accurate because it achieves second order accuracy when calculating the normal stresses and mixed first/second order accuracy when calculating the shear stresses, when the gridding is cartesian or orthogonal corner-point geometry. To illustrate, the differenced strains on the x+ side of the x-direction rock force balance control volume in Figure 24.1 can be written

$$\varepsilon_{x_{x+}} = \frac{u_{i+1,j,k} - u_{i,j,k}}{\Delta x_{i,j,k}}$$

$$\varepsilon_{y_{x+}} = \frac{v_{i,j+1,k} - v_{i,j,k}}{\Delta y_{i,j,k}} \qquad \text{[EQ 24.25]}$$

$$\varepsilon_{z_{x+}} = \frac{w_{i,j,k+1} - w_{i,j,k}}{\Delta z_{i,j,k}}$$

Another consequence of the staggered gridding is that the force balance equations [EQ 24.1] give rise to a thirteen-banded jacobian for three-dimensional simulations. In addition to the standard diagonal and six off-diagonal bands, there are six more connecting points in the differencing stencil which contribute the other bands.

The finite element method again uses the same standard simulation grid. A variational form of the above equations is set up where each equation in [EQ 24.1] is multiplied by a component of a test function, then summed over all components and finally integrated over a domain of three dimensional grids. Basis functions are piece-wise tri-linear. All unknown rock displacements are defined at the nodes or corner-points of the grid.

Fluid Component Conservation, Rock Volume Conservation and Volume Balance Equations

Compositional and thermal simulation requires you to specify the number of fluid components. Fluid components are conserved in the standard user-specified grid blocks as noted above.

The default porosity-stress relation is conservation of rock volume in the grid block over the time step:

$$V^{n+1}(1-\phi^{n+1}) = V^{n}(1-\phi^{n}) \qquad \text{[EQ 24.26]}$$

where V is the bulk volume of the grid block, ϕ is the porosity. 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. 17], and also Quandalle and Sabathier [Ref. 7]).

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. 83], and calculate a total stress as follows:

$$\sigma_T = \sigma_{eff} - \alpha P \qquad \text{[EQ 24.27]}$$

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.

Variables and solution of the equations

Variables used in the 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, elastic rock displacements in the x, y, z directions, and porosity. 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.

The linear solver used to solve the coupled system of non-linear equations is an algebraic multi-grid solver, [Ref. 96].

Boundary conditions and initialization

Either stress or displacement boundary conditions can be chosen. Each side of the simulation grid, that is i-, i+, j-, j+ and k-, k+, can have one of the above boundary conditions prescribed. Internal boundaries can also have a load or displacement boundary condition applied.

It is required that one side of the grid be rigid. Usually the bottom of the grid will have a zero displacement condition. Overburden stresses are applied at the top of the grid, which is the k- boundary in the `STRESSBC` keyword, and horizontal stresses are applied on either i-, i+, j- or j+ and the other side is kept rigid. There are several reasons for this. One is that measured or estimated horizontal and vertical boundary stresses need only be applied on one side and are immediately communicated throughout the grid. Applying these stresses on both boundaries would double the applied boundary load. Secondly, the nature of equations [EQ 24.1] is such that the rock displacement variables are translationally invariant. Consequently, the linear solver needs to have the rigid (Dirichlet) boundary conditions in order to fix the solution of the linear system of equations. Failure to provide rigid boundary conditions will usually result in a failure of the solver to find a solution.

Plasticity

The keyword "GEOYLDF Geomechanics yield function parameters" on page 735 of the reference manual allows classical yield functions to be set in specified regions of the grid. Either Drucker-Praeger (default), Mohr-Coulomb or Von-Mises models can be chosen. All of these are defined with a cohesion and angle of internal friction. Arbitrary geomechanical regions in the grid can be set using the keyword "YLDNUM Geomechanics region numbers" on page 2056 of the "ECLIPSE Reference Manual". Parameters and yield functions are then defined in each region. For example, a region can be set around a particular well. This option is only available with the finite element stress calculation.

User Interface

You enter a Young's modulus, Poisson ratio and rock density for each grid block in the `GRID` section. A Biot constant may also be entered but, if not, it is defaulted to 1.0. Stress boundary conditions are entered in the `SCHEDULE` section. If a plastic calculation (finite element only) is required, you enter a cohesion, angle of internal friction and hardening parameters if desired in different regions of the grid in the `PROPS` section. Further discussion of this option is found in the `GEOYLDF` keyword.

Summary of keywords

RUNSPEC section

GEOMECH	Selects the geomechanics option, allows choices on fixed/moving grid, fully/partly coupled solution of the rock stress equations
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

BIOTC	Sets the Biot constant for each grid block.
FAULTS	Can define a geomechanics fault where slippage is allowed.
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 proponent flowback feature
ROCKDEN	Sets the rock density for each grid block.
YOUNGMOD	Sets the elastic Young's modulus for each grid block.

PROPS section

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.
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

YLDNUM	Defines a geomechanics region number for each grid block.

Solution section

OUTSOL	Same as RPTRST, below.
RPTRST	Controls output for GRAF or RESTART files.

Input Keyword Or Mnemonic	Output	E100	E300
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, PRSTRES3		x
ROCKDISP	Geomechanics, rock displacements; this produces output with the names ROCKDISX, ROCKDISY, ROCKDISZ		x

SCHEDULE section

RPTSCHED	Controls output to the print file

Input Keyword or Mnemonic	Output	E100	E300
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, PRSTRES3		x
ROCKDISP	Geomechanics, rock displacements; this produces output with the names ROCKDISX, ROCKDISY, ROCKDISZ		x

STRESSBC	Permits the application of a load or a velocity (rigid) boundary condition on all external and internal surfaces of the simulation grid.
WSCTAB	Links a well to a particular scale damage table as defined with SCDATABG. This feature is part of the PROPFLOW option.

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 fully- or partly-coupled geomechanics calculation in ECLIPSE 300 is selected using the `GEOMECH` keyword in the `RUNSPEC` section.

- The first argument is no longer available.
- The second argument is an option on how the geomechanics rock force balance is coupled to the other component conservation equations.
  - 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 memory is allocated/deallocated and this stress is updated using the latest fluid (pore) pressures.

  The two other possible choices here are:

  - to run the geomechanical force balance fully IMPES coupled
  - to update the rock stress at every time step.

  The fully IMPES coupled choice solves the stress equations, porosity-stress relation and the IMPES pressure equation simultaneously. This is much more computationally expensive and has not proved to be a robust as hoped. Although this option is still available, the use is cautioned that there may be problem when running this due to failure of the non-linear equations to converge.

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.

If the proponent flowback option is selected, parameters can be set with the PROPFLOW keyword.

Modifying permeability, Biot's constant as a function of stress and plasticity

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.

Plasticity can be activated if the finite element plastic calculation is desired. The keyword GEOYLDF defines parameters for the plastic yield functions and hardening laws in each geomechanics region.

Stress boundary conditions

Boundary conditions for the geomechanics rock force (stress) balance are determined with the keyword STRESSBC in the 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 rigid, zero-normal-displacement boundary condition. A user-defined boundary condition is either a constant rock velocity or a load (applied stress) on this surface. Usually a velocity boundary condition with zero velocity is used, which forces the boundary to remain rigid with no displacement.

Note A stress boundary condition cannot be applied on both faces in the same coordinate direction. This restriction arises from the translational invariance of the stress balance equations. If you apply a load on a "+" boundary in a certain direction, then the "-" boundary in the same direction must be rigid or constant velocity. You may mix coordinate directions for applying loads. For example, an overburden stress may be applied on the "K-" face, while the "K+" face or bottom is held rigid. At the same time, the "I+" and "J+" faces may be assigned a load but the opposite "I-" and "J-" faces must be assigned a velocity or held rigid. Any combination works as long as the opposite side to a boundary on which a load is applied is held rigid or, at most, is moving with a constant velocity.

The rest of the input file

The remaining part of the ECLIPSE 300 input file with geomechanics remains exactly the same as before. Note that there are some restrictions - see "Restrictions" on page 330.

Stress and pore pressure equilibration

Equilibration of stress with pressure is the stress profile that is the result of a given pressure distribution, stress boundary conditions and solving the steady state rock momentum balance equations as discussed above. See, for example, [EQ 24.1] in "Elastic Stress Equations" on page 317.

There are several ways to output this equilibrated stress profile.

The initial pressure distribution may be determined in the `SOLUTION` section using the keyword `EQUIL` or non-equilibrium initialization of the pressure and saturations by directly setting these properties. If the pore pressure profile has been calculated from a pore-pressure analysis or has been directly measured, then the keyword `PRESSURE` can be used in the `SOLUTION` section to set this property explicitly. Note that if this method is chosen, then the initial fluid state requires that the oil, water and gas saturations and/or hydrocarbon and water component mole fractions also be specified in order to completely define the fluid state. Keyword `NEI` can be used to set the mole fractions if only the fluid saturations are known. Stress boundary conditions are then specified in the `SCHEDULE` section using the keyword `STRESSBC`. An equilibrated stress profile that is the result of these initial pressures and stress boundary conditions can be obtained directly by specifying a partly coupled stress calculation in the `GEOMECH` keyword, that is by setting the second argument to be either T or R, then taking a small timestep directly after the `STRESSBC` keyword, for example 1 day. Do not specify any injection or production. The simulator will then calculate the single timestep and proceed to calculate the stress profile with the pressure distribution at the end of this timestep. If the initial pressure distribution is not in equilibrium, for example, if the `EQUIL` keyword has not been used and the pressures, saturations and mole fractions that have been directly entered are not in equilibrium, then the

simulator may need several non-linear iterations to converge. If the EQUIL keyword has been used, then the initial pressures are in equilibrium and the simulator should only need 1 iteration. The stress profile that is calculated at the end of the time step is the result of solving [EQ 24.1] and is in equilibrium with the pressures and boundary conditions. It can be displayed with the usual methods.

Another method is available if the pressure is only measured at discrete points. This method then involves running the coupled fluid flow and stress calculation to steady state. At the discrete points in the grid where the initial pressure is known, single production wells can be placed at these points with a limiting bottom hole pressure that is the measured pressure. Stress boundary conditions are set in the SCHEDULE section. Timestepping of the simulator then commences. Application of the stress boundary conditions squeeze the reservoir, which in turn raises the pressure and forces fluid out of the production wells. Fluid production at these points continues until the pressure has dropped to the limiting bottom hole pressure, which is the measured pressure datum. When all production has stopped, then the pressure is in equilibrium because the fluid flow conservation equations have been solved, which includes gravitational equilibrium, the pressure matches the pressure data at all points where the single wells have been defined and the stress profile is in equilibrium with this pressure profile because stresses are the result of solving the conservation of rock momentum equations. In reservoirs with lower permeability, the simulator may need to advance to very large, possibly geological times in order to reach steady state. The simulator reaches steady state faster when more points with measured pressure data are provided, that is more points in the grid where production wells can be defined.

Finally, if the simulator is carrying out a reservoir simulation with injection/production and the stress is being updated as determined by the second argument of the GEOMECH keyword, then the stress is in equilibrium with the pressure at that point in time or during convergence of the time step.

Performance

Running the rock geomechanics stress calculation fully IMPES coupled involves solving the force conservation equations simultaneously with the fluid component conservation equations. This option is only available with the finite difference method and involves much more computing time. To avoid unnecessary computation, it is advised to test whether there are significant differences in a fully or partly coupled geomechanical simulation.

If partly coupled, 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, extra memory is allocated, and 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. Finally, the extra allocated memory is deallocated. The reservoir simulation then proceeds with updated permeabilities and porosities until the next geomechanical stress update.

This partly-coupled calculation is much faster than fully coupled. Larger numbers of grids can be accommodated.

When the finite difference stress calculation is chosen, it is recommended that the SAMG solver, keyword SAMG, is chosen with default parameters. This can be accomplished by placing the keyword SAMG in the RUNSPEC section of the input file and on the following line, a backward slash.

Stability

A partly-coupled simulation should not affect the stability of the simulation. Changes to permeability and porosity due to the additional rock stress calculation are assumed to be small enough for stability not to be affected.

If running a fully IMPES coupled finite difference simulation in which rock stress and IMPES pressure equations are solved simultaneously, some sensitivity has been noted when the primary wetting phase saturation is at or near connate, particularly when wells are present in that block. There may also be problems with single phase flow when the rock stress is in a state of unstable equilibrium (see, for example, [Ref. 87], in the section "Stability of Elastic Systems").

Notes on the partly/fully coupled calculations

When using the partly coupled (default) calculation, porosities are updated when the rock stress calculation takes place, either at report steps or at the end of every time step. The rock compressibility is set to zero during the advancement of the reservoir simulation to a report or time step. However, this can be overidden. To do this, the 71st argument of OPTIONS3 can be set to be greater than 0. Then the rock compressibility can be set with the keyword ROCK. See also the discussion of the second porosity-stress relation that is set in the 4th argument of the GEOMECH keyword.

If there is difficulty in converging the fully coupled case, several possibilities exist as to the reasons and the remediation.

Firstly, the reason may be due to the primary wetting phase, water for example, being at or near a critical saturation. If the nonconvergence is not immediately seen at the beginning of the simulation but develops later, this may be the cause. A primary wetting phase that is changing from mobile to immobile, in turn, causes perturbations in the pressure, which feed back to the stress calculation. Because the porosity is implicit and a direct consequence of the rock displacement, the problem may have difficulty in converging due to the changing pore volume causing the saturations to oscillate around the critical point. A possible remedy is to slightly change the relative permeability endpoints or functions.

Secondly, there are known instabilities to the elastic system of equations, discussed for example in Landau and Lifshitz, [Ref. 87]. If this is the cause of the nonconvergence in the fully IMPES coupled case, then it is also there in the partly coupled case. Also this is evident right from the beginning of the simulation. To test this hypothesis, try the following. Switch the coupling to solve the stress equations at every timestep by setting the second argument of the GEOMECH keyword to T. Then run the simulator for several small timesteps and look at rock displacements as printed in the .prt file. If they oscillate and possibly grow, then this is the likely cause. Two possible remedies exist in this case. One is to use a damping factor. The 30th argument of CVCRIT, which is defaulted to 1, can be set to a value greater than 0 and less than 1, say 0.5, to see if this

stabilizes the rock displacements. If so, then you can continue to run the stress calculation using a damping factor less than 1, but only in the partly coupled case. A second remedy is to put some extra blocks, if possible, somewhere in the region where the oscillations appear to be arising in order to anchor the displacements. Often only one extra block is enough to stabilize the calculation.

Restrictions

ECLIPSE 300

Geomechanics in ECLIPSE 300 has the following restrictions.

- The finite difference method of solving the stress equations when run with corner-point gridding does not contain any correction terms for non-orthogonal, skewed grids. This can lead to unacceptable errors if these types of corner-point grids are present in the simulation. The user is advised to use corner-point gridding that is as close as possible to cartesian, where grid blocks are oriented in the standard x, y and z directions, and without skew. An error condition will be generated if this combination is specified, which can be overidden by setting the 85th position of the `OPTIONS3` keyword to be 1.
- Full IMPES coupling of the geomechanics stress calculation to the reservoir component conservation equations can be computationally very expensive for larger numbers of grids.
- Only the `IMPES` method (see "The IMPES and AIM methods" on page 238 of the chapter "Formulation of the Equations" on page 232 of this manual) of solving the nonlinear equations is available if running the fully-coupled geomechanics stress calculation. If running partly coupled, then any method such as `AIM` or fully implicit will work for solving the standard fluid flow equations.
- Dual porosity calculations are not available.
- Parallel is not available.
- Local Grid Refinement is not available.
- Radial gridding is not available.
- Faults and pinch-outs are not available

Gi Pseudo-compositional Model

Chapter 25

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 timestep 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

$$R_{v,\,\mathrm{inv}} = f_{dr} \cdot R_{v,\,\mathrm{inj}} + (1 - f_{dr}) R_{v,\,\mathrm{cell}} \qquad \text{[EQ 25.1]}$$

which yields the following expression for the fractional injection gas flux into each grid cell:

$$f_{dr} = \frac{R_{v,\,\mathrm{inv}} - R_{v,\,\mathrm{cell}}}{R_{v,\,\mathrm{inj}} - R_{v,\,\mathrm{cell}}} \qquad \text{[EQ 25.2]}$$

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:

$$\mathrm{Gi} = \int \frac{f_{dr} F_g}{\mathrm{HCPV}} dt \qquad \text{[EQ 25.3]}$$

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:

$$f_{dr} \cdot F_g = \left(1 - \frac{R_{v,\ \mathrm{inv}}}{R_{v,\ \mathrm{cell}}}\right) F_g \qquad \text{[EQ 25.4]}$$

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:

$$f_{drw} \cdot Q_g = \left(1 - \frac{R_{v,\ \mathrm{inj}}}{R_{v,\ \mathrm{cell}}}\right) Q_g \qquad \text{[EQ 25.5]}$$

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 25.3] is:

$$\delta \mathrm{Gi} = \frac{f_{dr} F_g + f_{drw} Q_g}{\mathrm{HCPV}} \delta t \qquad \text{[EQ 25.6]}$$

$$\delta \mathrm{Gi} = \frac{(R_v - R_{v,\ \mathrm{up}}) F_g \delta t}{R_v \cdot \phi \cdot V \cdot (1 - S_w)} + \frac{(R_v - R_{v,\ \mathrm{inj}}) Q_g \delta t}{R_v \cdot \phi \cdot V \cdot (1 - S_w)} \qquad \text{[EQ 25.7]}$$

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 ($\mathrm{VOL} \rightarrow 0$, $\delta t \rightarrow 0$) the differential equation for Gi in one dimension becomes:

$$\frac{d}{dt}\mathrm{Gi} = \frac{F_g}{\phi \cdot R_v A(1 - S_w)} \frac{dR_v}{dx} \qquad \text{[EQ 25.8]}$$

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).

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

GIMODEL	Activates the Gi Pseudo-compositional model.

PROPS section

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

GI	Initial Gi values for enumeration.

SUMMARY section

The following keywords control output of data specific to the Gi model:

Table 25.1 Gi pseudo-compositional SUMMARY output controls

Block	Information
BGI	Block Gi value

SCHEDULE section

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

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
==============================================================
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  ============================================================
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 ============================================================
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 ============================================================
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 26

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.

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 GeoQuest 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 timestep 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 GeoQuest 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.

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 'FUNCTION' run in SimOpt.

Current restrictions

ECLIPSE 100 only

- The Gradient option may not at present be used with the Parallel option or with Reservoir Coupling.

ECLIPSE 100 only

- 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 timestep 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

- LGRs employing local timestepping 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

- 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.
- Gradients cannot at present be obtained for multi-segmented injector wells.

ECLIPSE 300 only

- Gradients cannot be used with the Grid Coarsening Option.

Current Uses (ECLIPSE 300 only)

- It is possible to use the Gradient option in both Black Oil and Compositional runs.
- In addition, this option can be used in Fully Implicit and AIM (Adaptive IMplicit) modes.
- The Gradient option may be used with the Parallel option in ECLIPSE 300.

Example (ECLIPSE 100 only)

To set three transmissibility multiplier parameters, with NX=3 NY=3 NZ=3, set the number of parameters using RUNSPEC keyword HMDIMS

```
HMDIMS
  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 using RUNSPEC keyword HMDIMS

```
HMDIMS
  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 timestep 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 timestep 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 201). 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+00
  0.00000000000000D+00   0.00000000000000D+00   0.26393554627632D+00
  0.26588984672034D+00   0.26491269649833D+00   0.26491269649833D+00
'VALUES  '           9 'DOUB'
  0.00000000000000D+00   0.00000000000000D+00   0.00000000000000D+00
  0.00000000000000D+00   0.00000000000000D+00   0.20023871765242D+03
  0.20038969805460D+03   0.20031420785351D+03   0.20031420785351D+03
'NAMES   '           5 'CHAR'
'PRO     ' 'HMTRANX ' '        ' 'PRODUCER' 'OPEN    '
'HMHEADI '           2 'INTE'
          1           1
'DERIVS  '           9 'DOUB'
  0.00000000000000D+00   0.00000000000000D+00   0.00000000000000D+00
  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+00
  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 timestep used

Subsequent records are written in the order shown in Figure 26.1

Figure 26.1 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'	

ECLIPSE 300

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:

ECLIPSE 300

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)
		• UNITS: barsa (METRIC), psia (FIELD), atma (LAB)
	Item 7	4 point pressure average (see WPAVE)
		• UNITS: barsa (METRIC), psia (FIELD), atma (LAB)
	Item 8	5 point pressure average (see WPAVE)
		• UNITS: barsa (METRIC), psia (FIELD), atma (LAB)
	Item 9	9 point pressure average (see WPAVE)
		• 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 GeoQuest software - please contact your GeoQuest 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 GeoQuest software - please contact your GeoQuest 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 timestep, ECLIPSE forces the fully implicit residual in each simulator grid block to be zero (see "Formulation of the Equations" on page 232).

This can be written as

$$R = \frac{M_{t+dt} - M_t}{dt} + F_{t+dt} + Q_{t+dt} = 0 \qquad \text{[EQ 26.1]}$$

where the subscripts t and $t + dt$ denote quantities at the current and advance time, with a simulator timestep 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 timestep.

For any gradient parameter λ, we have, from [EQ 26.1],

$$R(X_{t+dt}, \lambda) = \frac{M_{t+dt}(X_{t+dt}, \lambda) - M_t(\lambda)}{dt} + F_{t+dt}(X_{t+dt}, \lambda) + Q_{t+dt}(X_{t+dt}, \lambda) \qquad \text{[EQ 26.2]}$$

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 26.2] throughout with respect to λ yields

$$\frac{\partial R}{\partial X_{t+dt}} \frac{\partial X_{t+dt}}{\partial \lambda} = -\frac{\partial R}{\partial \lambda} \qquad \text{[EQ 26.3]}$$

which can be inverted as

$$\frac{\partial X_{t+dt}}{\partial \lambda} = -\left(\frac{\partial R}{\partial X_{t+dt}}\right)^{-1} \frac{\partial R}{\partial \lambda} \qquad \text{[EQ 26.4]}$$

For a fully implicit discretization,

$$\frac{\partial R}{\partial \lambda} = \frac{1}{dt}\left(\frac{\partial M_{t+dt}}{\partial \lambda} - \frac{dM_t}{d\lambda}\right) + \frac{\partial F_{t+dt}}{\partial \lambda} + \frac{\partial Q_{t+dt}}{\partial \lambda} \qquad \text{[EQ 26.5]}$$

[EQ 26.4] 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 26.4] at each simulator timestep. When there are several gradient parameters, each requires the solution of the same set of linear equations [EQ 26.4] but with different right hand sides $\partial R / \partial \lambda$ in [EQ 26.5].

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 26.2] can be written

$$R(X_{t+dt}, \lambda) = \frac{\lambda M_{t+dt}(X_{t+dt}) - M_t(\lambda)}{dt} + F_{t+dt}(X_{t+dt}) + Q_{t+dt}(X_{t+dt}) \qquad \text{[EQ 26.6]}$$

and so

$$\frac{\partial R}{\partial \lambda} = \frac{1}{dt}\left(M_{t+dt} - \frac{dM_t}{d\lambda}\right) \qquad \text{[EQ 26.7]}$$

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 26.5] 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`, `KMKRW` etc.). During a gradient-based minimization loop, a sequence of multiplicative modifiers $\lambda_i, i = i, \ldots, \upsilon$ 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.

Summary of keywords

RUNSPEC section

ECLIPSE 300	FMTHMD	Declares that the HMD file is to be formatted.
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

HMFAULTS	Specifies gradients for faults.
HMMULRGT	Specifies inter-region transmissibility parameters.
HMAQUNUM	Specifies gradients for numerical aquifer parameters.

PROPS section

HMROCK	Specifies rock compressibility parameters.
HMROCKT	Specifies rock compaction parameters.

REGIONS section

HMxxxxxx	Specifies gradient parameters.

Note The following options apply to the HMxxxxxx keyword.

	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.
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

HMAQUCT	Specifies gradients for Carter-Tracy aquifers.
HMAQUFET	Specifies gradients for Fetkovich aquifers.

SCHEDULE section

HMWPIMLT	Specifies gradients with respect to well productivity index.

Parameter modifier keywords

GRID section

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

ECLIPSE 300

HMMULTFT	Specifies modifiers for fault transmissibilities.
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.
HMMULTZ-	Specifies negative Z-direction transmissibility modifiers.
HMMULTR	Specifies R-direction transmissibility modifiers.
HMMULTTH	Specifies theta-direction transmissibility modifiers.

PROPS section

HMPROPS	Section heading for property modifiers.

Note The following options apply to the HMPROPS keyword.

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 "HMPROPS Section heading for end-point scaling modifiers" on page 869

SOLUTION section

HMMLCTAQ	Specifies modifiers for Carter-Tracy aquifers.
HMMLFTAQ	Specifies modifiers for Fetkovich aquifers.

SUMMARY keywords

TCPUTSHT	CPU time per timestep for all the gradient calculations.
TCPUHT	Total CPU time for all the gradient calculations.
TCPUTSH	CPU time per timestep 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.

Miscellaneous keywords

ECLIPSE 300

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.
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 DATA
-- 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 27

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:

- Drill new wells from the drilling queue
- Open new connections on AUTO in existing wells
- Reduce each well's THP limit to a specified lower value
- Change VFP table numbers (for example to model a retubing operation)
- Change well ALQ values (that is apply a fixed amount of artificial lift)
- 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 552).

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:

- it is not subordinate to the group that is unable to achieve its production target
- 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
- it is subordinate to a prioritization group (keyword GCONPRI). The drilling queue logic is not activated under prioritization group control
- 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
- its group, or any superior group in the hierarchy, already has its maximum allowed number of open wells (keyword GECON item 9)
- the oil saturation in all its completed grid blocks is below a specified minimum value (keyword WELSOMIN)
- 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:

- it has no connections in its AUTO opening queue
- it is already operating at its own target or limiting rate for the required phase, and so cannot increase its flow further
- it is not subordinate to the group that is unable to achieve its production target
- 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
- 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:

- it has no secondary THP limit, or the THP has already been reduced to its secondary limit
- it is currently operating under a control mode other than THP control
- it is not subordinate to the group that is unable to achieve its production target
- 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 538), 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:

- it has no secondary VFP table number, or it has already switched to its secondary table
- it is currently operating under a control mode other than THP control
- it is not subordinate to the group that is unable to achieve its production target
- 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:

- it has no secondary ALQ value set in keyword WLIFT, or it has already switched fully to this ALQ value
- it is currently operating under a control mode other than THP control
- it is not subordinate to the group that is unable to achieve its production target
- 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
- 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 276). 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 552). 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 538), 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 28

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. 17], Killough [Ref. 18], Carlson [Ref. 19], Land [Ref. 20] 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-, etc., 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.

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. 18].

ECLIPSE 100

Relative permeability hysteresis may be modeled by the Carlson [Ref. 19] or Killough [Ref. 18] methods.

ECLIPSE 300

Relative permeability hysteresis may be modeled by the Carlson [Ref. 19] or Killough [Ref. 18] 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 28.1. 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 28.1 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. 19] or Killough's method [Ref. 18]. 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. 19] or Killough's method [Ref. 18] 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

$$S_{ncrt} = S_{ncrd} + \frac{S_{hy} - S_{ncrd}}{1 + C(S_{hy} - S_{ncrd})} \qquad \text{[EQ 28.1]}$$

where

$$C = \frac{1}{S_{ncri} - S_{ncrd}} - \frac{1}{S_{n\,\max} - S_{ncrd}} \qquad \text{[EQ 28.2]}$$

(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

$$K_{rn}(S_n) = \frac{K_{rni}(S_{norm})K_{rnd}(S_{hy})}{K_{rnd}(S_{n\,\max})} \qquad \text{[EQ 28.3]}$$

where K_{rni} and K_{rnd} represent the relative permeability values on the bounding imbibition and drainage curves respectively, and

$$S_{norm} = S_{ncri} + \frac{(S_n - S_{ncrt})(S_{n\,\max} - S_{ncri})}{S_{hy} - S_{ncrt}} \qquad \text{[EQ 28.4]}$$

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 unphysical behavior when $\boldsymbol{S}_{hy}$ is small. As $\boldsymbol{S}_{hy}$ tends to $\boldsymbol{S}_{ncrd}$ in [EQ 28.1], $\boldsymbol{S}_{ncrt}$ tends to $\boldsymbol{S}_{ncrd}$, 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 28.1] has been modified as follows:

$$S_{ncrt} = S_{ncrd} + \frac{S_{hy} - S_{ncrd}}{A + C(S_{hy} - S_{ncrd})} \qquad \text{[EQ 28.5]}$$

where

$$A = 1 + a \cdot (S_{n\,\max} - S_{hy}) \qquad \text{[EQ 28.6]}$$

The parameter a is input through the fourth data item in the EHYSTR keyword and defaults to 0.1.

ECLIPSE 300

Jargon's method for relative permeability hysteresis

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:

$$S_{ncrt} = S_{ncrd} + \frac{(S_{ncri} - S_{ncrd})(S_{nhy} - S_{ncrd})}{(S_{nmax} - S_{ncrd})} \qquad \text{[EQ 28.7]}$$

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:

$$R(X) = \frac{K_{ri}(S_n)}{K_{rd}(S_n)} \quad \text{[EQ 28.8]}$$

where X is defined from the non-wetting saturation S_n by

$$X = \frac{S_n - S_{ncrd}}{S_{nmax} - S_{ncrd}} \quad \text{[EQ 28.9]}$$

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:

$$K_{rn}(S_n) = R(X_s)K_{rnd}(S_n) \quad \text{[EQ 28.10]}$$

$$X_s = \frac{S_n - S_{ncrd}}{S_{hy} - S_{ncrd}} \quad \text{[EQ 28.11]}$$

This can also be written as

$$K_{rn}(S_n) = \frac{K_{ri}(\hat{S})}{K_{rd}(\hat{S})}K_{rnd}(S_n), \text{ where}$$

$$\hat{S} = S_{ncrd} + (S_n - S_{ncrd})\frac{(S_{nmax} - S_{ncrd})}{(S_{hy} - S_{ncrd})}$$

ECLIPSE 300

Differences in hysteresis treatment in three-phase cases

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 it 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 $\boldsymbol{S}_{ncri} > \boldsymbol{S}_{ncrd}$ 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 28.2. 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 28.2 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,

$$K_{rw}(1-S_{ncrt}) = K_{rwd}(1-S_{ncrt}) + (K_{rwi}(1-S_{ncri}) - K_{rwd}(1-S_{ncri}))\left(\frac{S_{ncrt}-S_{ncrd}}{S_{ncri}-S_{ncrd}}\right)^{A} \quad \text{[EQ 28.12]}$$

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

$$K_{rw}(S_w) = K_{rwd}(1-S_{hy}) + \frac{(K_{rw}(1-S_{ncrt}) - K_{rwd}(1-S_{hy}))K_{rwi}(1-S_{norm})}{K_{rwi}(1-S_{ncri})} \quad \text{[EQ 28.13]}$$

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 28.3.

Figure 28.3 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:

$$P_c = P_{cd} + F(P_{ci} - P_{cd}) \qquad \text{[EQ 28.14]}$$

with

$$F = \left(\frac{1}{S_w - S_{why} + E} - \frac{1}{E}\right) \Big/ \left(\frac{1}{S_{wma} - S_{why} + E} - \frac{1}{E}\right) \qquad \text{[EQ 28.15]}$$

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 $P_{cw} \rightarrow P_{cwi}$.

ECLIPSE 100

The curvature parameter E can be entered in Item 1 of the `EHYSTR` keyword.

ECLIPSE 300

The curvature parameter is fixed at 0.1.

The imbibition capillary pressure P_{ci} in [EQ12.9] 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:

$$P_c = P_{ci} + G(P_{cd} - P_{ci}) \qquad \text{[EQ 28.16]}$$

with

$$G = \left(\frac{1}{S_{dep} - S_w + E} - \frac{1}{E}\right) \Big/ \left(\frac{1}{S_{dep} - S_{why} + E} - \frac{1}{E}\right) \qquad \text{[EQ 28.17]}$$

in which

S_{why}	is 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 [EQ12.9] to be **re-traversed** when this reversal back to drainage occurs.

ECLIPSE 100

To obtain the **new** scanning curve 't' described by [EQ12.11] 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

$$P_c = P_{ci} + F(P_{cd} - P_{ci}) \qquad \text{[EQ 28.18]}$$

with

$$F = \left(\frac{1}{S_{why} - S_w + E} - \frac{1}{E}\right) / \left(\frac{1}{S_{why} - S_{wcrd} + E} - \frac{1}{E}\right) \quad \text{[EQ 28.19]}$$

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 28.4.

Figure 28.4 The gas capillary pressure case

Analogously to the case of Pcw,

$$P_c = P_{cd} + F(P_{ci} - P_{cd}) \quad \text{[EQ 28.20]}$$

with

$$F = \left(\frac{1}{S_{ghy} - S_g + E} - \frac{1}{E}\right) / \left(\frac{1}{S_{ghy} - S_{gcr} + E} - \frac{1}{E}\right) \quad \text{[EQ 28.21]}$$

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 29

Introduction

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL

This chapter describes the WAG (Water Alternating Gas) Hysteresis option.

The hysteresis models described in "Hysteresis" on page 394 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. 19] and [Ref. 20].

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 29.1).

Figure 29.1 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

$$Sg_{trap} = Sg_{cr} + \frac{(Sg_m - Sg_{cr})}{(1 + C \cdot (Sg_m - Sg_{cr}))} \qquad \text{[EQ 29.1]}$$

where

C is Land's parameter input with the WAGHYSTR keyword.

The gas relative permeability on the scanning curve is given by

$$K_{rg}(Sg) = K_{rg}^{drain}(Sg_f) \qquad \text{[EQ 29.2]}$$

where

$$Sg_f = Sg_{cr} + \frac{1}{2}\left\{(Sg - Sg_{trap}) + \sqrt{(Sg - Sg_{trap})^2 + \frac{4}{C}(Sg - Sg_{trap})}\right\} \qquad \text{[EQ 29.3]}$$

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.

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 29.3] can be interpreted as the free gas saturation for a given saturation state. Hence the trapped gas saturation is given by

$$Sg_{td} = Sg - Sg_f \qquad \text{[EQ 29.4]}$$

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 29.2.

Figure 29.2 Schematic diagram showing a typical secondary drainage curve

These secondary drainage curves are calculated using the following equation

$$K_{rg}^{drain} = [K_{rg}^{input} - K_{rg}^{input}(Sg^{start})] \cdot \left[\frac{Sw_{co}}{Sw^{start}}\right]^{\alpha} + [K_{rg}^{imb}(Sg^{start})] \qquad \text{[EQ 29.5]}$$

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 29.3] 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.

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 703) 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.

$$SOM_{mod} = SOM - (a \cdot Sg_{td}) \qquad \text{[EQ 29.6]}$$

where

SOM	is the minimum residual oil saturation
Sg_{td}	is the dynamic trapped gas saturation given by [EQ 29.4]
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 29.3.

Figure 29.3 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

$$K_{rw}^{imb} = K_{rw2} \cdot \left(1 - \frac{Sg^{start}}{Sg_{max}}\right) + K_{rw3} \cdot \left(\frac{Sg^{start}}{Sg_{max}}\right) \qquad \text{[EQ 29.7]}$$

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 29.4.

Figure 29.4 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

$$K_{rw}^{drain} = K_{rw}^{imb} \cdot \left(1 - \frac{Sg - Sg^{sdrain}}{Sg_{max} - Sg^{sdrain}}\right) + K_{rw3} \cdot \left(\frac{Sg - Sg^{sdrain}}{Sg_{max} - Sg^{sdrain}}\right) \quad \text{[EQ 29.8]}$$

If $Sg < Sg^{drain}$ then

$$K_{rw}^{drain} = K_{rw}^{imb} \cdot \left(1 - \frac{Sg^{sdrain} - Sg}{Sg^{sdrain}}\right) + K_{rw2} \cdot \left(\frac{Sg^{sdrain} - Sg}{Sg^{drain}}\right) \quad \text{[EQ 29.9]}$$

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 a 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 30

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.

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 is treated as inactive.

   The ISOLNUM data can be written out for an existing model by using the RPTISOL keyword in the GRID section. The ISOLNUM array is written out to the debug file, and 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 GRID. 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 419.

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 420.

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:

COORD....    dummy values for the entire grid.
ZCORN....

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 /

COORD....    dummy values for the entire grid (both grids).
ZCORN....

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 31

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 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 below in "Non-equilibrium initialization (ECLIPSE 300)" on page 431

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

$$P_{cow}(S_w) = P_o - P_w \qquad \text{[EQ 31.1]}$$

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

$$P_{cog}(S_g) = P_g - P_o \qquad \text{[EQ 31.2]}$$

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 693.

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

$$P_{cow}(S_w) + P_{cog}(S_g = 1 - S_w) = P_g - P_w \qquad \text{[EQ 31.3]}$$

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

ECLIPSE 100

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.

ECLIPSE 100

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).

ECLIPSE 100

The Quiescence option

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

$$P_{wat} = P - P_{cow} + PMODW \qquad \text{[EQ 31.4]}$$

and

$$P_{gas} = P + P_{cog} + PMODG \qquad \text{[EQ 31.5]}$$

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

$$P = max(P, p_{wat} - (\tilde{\rho}_{wat} - \tilde{\rho}_{oil})H/2, p_{gas} - (\tilde{\rho}_o - \tilde{\rho}_g)H/2)$$ [EQ 31.6]

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

$$PMODW = sign(p_{wat} - P + P_{cow}) \times min(|p_{wat} - P + P_{cow}|, (\rho_w - \rho_o)H/2)$$ [EQ 31.7]

and

$$PMODG = sign(P + P_{cog} - p_{gas}) \times min(|P + P_{cog} - p_{gas}|, (\tilde{\rho}_o - \tilde{\rho}_g)H/2)$$ [EQ 31.8]

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:

$$f_{iL}(P, x) = f_{iV}(P, y) \qquad \text{[EQ 31.9]}$$

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:

$$z_i = x_i L + y_i V \qquad \text{[EQ 31.10]}$$

with

$$L = \frac{S_o b_o^m}{S_o b_o^m + S_g b_g^m} \qquad \text{[EQ 31.11]}$$

where b_o^m and b_g^m are the oil and gas molar densities. As long as [EQ 31.9] 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 31.9] 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:

- 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.
- 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:

- Specify the liquid and vapor compositions explicitly using the XMF and YMF keywords. These enter values for every cell and for every component.

- 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.

Local Grid Refinement and Coarsening

Chapter 32

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 32.1.

Figure 32.1 Single column radial grid refinement

Alternatively it is possible to use a two-dimensional model with NTHETA=1 that looks like Figure 32.2.

Figure 32.2 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 32.3.

Figure 32.3 Radial refinement in a four column box

Alternatively, setting NTHETA=4 creates a refinement which looks like Figure 32.4.

Figure 32.4 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 442).

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 32.5 (in the areal direction).

Figure 32.5 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.

ECLIPSE 100

Note When using the REFINE keyword in parallel runs, it is necessary to insert an ENDFIN keyword after reading the required local grid data.

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 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 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, or MULTIPLY keywords) without previously defining the property for those cells.

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. 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 timestep 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 'LGR1' and 'LGR2'. The GRID section might look like:

```
CARFIN
-- NAME    I1-I2  J1-J2  K1-K2 NX NY NZ NWMAX--
  'WEST'    4 5    3 3    3 4   4  4  4   1  /

CARFIN
-- NAME    I1-I2  J1-J2  K1-K2 NX NY NZ NWMAX--
  'EAST'    6 7    4 5    4 5   4  4  4   1  /

AMALGAM
  'EAST' 'WEST' /
/
```

This generates a composite refinement which looks like Figure 32.6 (in the areal direction).

Figure 32.6 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'  3  2  2   2   'OPEN'   0     -1    0.3    3*           'X' /
  'DEV'   'WEST'  4  2  3   3   'OPEN'   0     -1    0.3    3*           'X' /
  'DEV'   'EAST'  1  2  1   1   'OPEN'   0     -1    0.3    3*           'X' /
  'DEV'   'EAST'  2  2  2   2   '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 timestepping 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 porosity

In dual porosity runs the global coordinates specified in CARFIN and RADFIN refer to the matrix blocks only. For example, 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.

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 timestep, 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 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 timestep. 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 "Timesteps" on page 450), 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 893) 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

$$\mathrm{TRANN}_{lg} = \frac{\mathrm{CDARCY} \cdot \mathrm{TMLTX}_l \cdot \mathrm{DIPC}}{B} \qquad \text{[EQ 32.1]}$$

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

$$B = \frac{1}{2A_l} \cdot \left(\frac{\mathrm{DX}_g}{\mathrm{PERMX}_g} + \frac{\mathrm{DX}_l}{\mathrm{PERMX}_l} \right) \qquad \text{[EQ 32.2]}$$

The area is given by

$$A_l = \mathrm{DY}_l \cdot \mathrm{DZ}_l \cdot \mathrm{NTG}_l \qquad \text{[EQ 32.3]}$$

The dip correction is given by

$$\mathrm{DIPC} = \frac{\mathrm{DHS}}{\mathrm{DHS} + \mathrm{DVS}} \qquad \text{[EQ 32.4]}$$

with

$$\mathrm{DHS} = \left(\frac{\mathrm{DX}_l + \mathrm{DX}_g}{2} \right)^2 \qquad \text{[EQ 32.5]}$$

and

$$\mathrm{DVS} = (\mathrm{DEPTH}_l - \mathrm{DEPTH}_g)^2 \qquad \text{[EQ 32.6]}$$

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.

Timesteps

Local timestepping (ECLIPSE 100)

Efficiency is achieved by solving each local grid (or amalgamation of local grids) individually using “local timestepping”. This allows the local grids to have shorter timesteps 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 timesteps to reach the end of a global timestep. This is very efficient in cases where the local grids naturally require much shorter timesteps than the global grid.

Local timestepping 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 timestepping can cause group flow rates to drift significantly away from their group control targets. The approach of local timestepping is the default method for solving local grids and was the only method available in versions of ECLIPSE 100 prior to 97A.

When local timestepping is used, the sequence for calculating each global timestep is as follows:

1 The global model is advanced through a new timestep 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 timesteps that are often smaller than the global timestep. The local timesteps are chosen to synchronize with the end of each global timestep.

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 timestep). 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 timestep.

5 After the completion of a global timestep the material for each phase in each host cell is compared with the material sum for the corresponding local cells. Subsequent global timesteps 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 timestep 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 timestepping. 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 timestepping. However, the coupled system chooses a timestep that is the minimum of natural timestep over the global and all local grids. In cases where one LGR requires a timestep 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 timestep 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 timestepping. 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 timestep as the global grid. There is no local timestepping 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.

ECLIPSE 100

3. 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.
4. It is not currently possible to coarsen within a local grid.

ECLIPSE 100 only

5. 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

6. Adjoining radial LGRs are allowed.
7. 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`).
8. 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.
9. 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`).
10. Aquifers cannot be attached to local grids. ECLIPSE ignores any aquifer connections to refined cells.
11. The diffusion options (Molecular Diffusion and Environmental Tracers) are not available in local grid runs.
12. The `NNC` or `EDITNNC` keywords for the global grid should not refer to non-neighbor connections connecting refined cells in the global grid.

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 timestep. When using local timestepping, 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 timestep, 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 timesteps, 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 timestepping is employed, economic limits applied to local wells and their connections (WECON and CECON) are checked at the end of each local timestep 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 timestep. 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.

3 In parallel runs LGRs cannot be switched on (LGRON) after they have been switched off

4 In parallel runs wells cannot be declared inside an LGR (WELSPECL) once that LGR has 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 timestepping 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 timestep, 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 timesteps, 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.

Radial refinements

- 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:
  - 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.
  - 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.
  - 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.)
- 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.
- 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.
- The sides of the inner radial cells are assumed to be vertical.
- The innermost radius must be defined by INRAD.
- 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.
- The user may provide some or all DR values. Any missing radii are completed as a geometric series, as follows:

  No of radii = NR
  No of radii defined by user = NU
  Radius of inside of outer local block = 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 $\mathrm{FAC} = \left(\frac{\mathrm{RREF}}{\mathrm{RU}}\right)^{1/(\mathrm{NR}-\mathrm{NU}-1)}$

- 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.

- Figure 32.7 shows the numbering convention used for connecting local radial grids to the host when RADFIN is used:

Figure 32.7 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 32.8

Figure 32.8 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.

- 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 457, and host cells which are approximately square in cross-section, this method corresponds to the approach described by Pedrosa and Aziz [Ref. 21], and Ewing et al. [Ref. 22].

Alphabetical list of GRID section keywords allowed within a local grid

Refer to the "ECLIPSE Reference Manual" for a further description of each keyword.

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.
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.
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.
THCONR	Specifies rock 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

$$DX_c = \frac{\sum_f DX_f}{(J2 - J1 + 1)(K2 - K1 + 1)},$$

with similar expressions for DY_c and DZ_C.

Permx, Permy, Permz

$$Kx_c = \frac{\sum_f PV_f Kx_f}{PV_c}.$$

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.

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 32.9 in the case NRX = NRY = 3:

Figure 32.9 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 ninepoint schemes are incompatible with autorefinement, and it is not possible to re-specify transmissibility multipliers within the SCHEDULE section when using autorefinement.
5. It is not possible to use the grid coarsening option when restarting a run.

Summary of keywords

RUNSPEC section

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

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.
	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

ENDFIN	terminates data for a local grid.
REFINE	indicates that subsequent data refers to a named local grid.

PROPS section

ENDFIN	terminates data for a local grid.
REFINE	indicates that subsequent data refers to a named local grid.

REGIONS section

ENDFIN	terminates data for a local grid.
REFINE	indicates that subsequent data refers to a named local grid.

SOLUTION section

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

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.
	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 timestep.
	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

ECLIPSE 100 only	OPTIONS	Switch 32 controls timestep 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 timestep. 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

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        ==========================================================
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     ==========================================================
FOPR
FGPR
FWPR
FOPT
FGPT
FWPT
FWCT
FGOR
LWBHP
'LGR1' 'PROD' /
/
LCPR
'LGR1' 'PROD' 3 2 2 /
/
FPR
FWIR
FWIT
RUNSUM
```

SCHEDULE

```
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  /
/
```

Timesteps 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 33

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. 23]).

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 763 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 790 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

$$S_o^* = \mathrm{MAX}(S_o - S_{or}, 0.0) \qquad \text{[EQ 33.1]}$$

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,

$$K_{ro} = \frac{S_o}{S_o + S_s} K_{rn}(S_n) \qquad \text{[EQ 33.2]}$$

$$K_{rs} = \frac{S_s}{S_o + S_s} K_{rn}(S_n) \qquad \text{[EQ 33.3]}$$

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:

$$K_{ro} = M_{kro} \frac{S_o}{S_o + S_s} K_{rn}(S_n) \qquad \text{[EQ 33.4]}$$

$$K_{rs} = M_{krs} \frac{S_s}{S_o + S_s} K_{rn}(S_n) \qquad \text{[EQ 33.5]}$$

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

$$P_s = P_o \qquad \text{[EQ 33.6]}$$

$$P_w = P_o - P_{cnw}(S_w) \qquad \text{[EQ 33.7]}$$

Note that equations [EQ 33.1], [EQ 33.3] and [EQ 33.2] 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:

$$\mu_{o\ \mathrm{eff}} = \mu_o^{1-\omega}\mu_m^{\omega} \qquad \text{[EQ 33.8]}$$

$$\mu_{s\ \mathrm{eff}} = \mu_s^{1-\omega}\mu_m^{\omega} \qquad \text{[EQ 33.9]}$$

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

$$\frac{1}{\mu_m}^{1/4} = \frac{S_s}{S_n}\left(\frac{1}{\mu_s}\right)^{1/4} + \frac{S_o}{S_n}\left(\frac{1}{\mu_o}\right)^{1/4} \qquad \text{[EQ 33.10]}$$

$$\mu_m = \frac{\mu_o\mu_s}{\left(\frac{S_s}{S_n}\cdot\mu_o^{1/4} + \frac{S_o}{S_n}\cdot\mu_s^{1/4}\right)^4} \qquad \text{[EQ 33.11]}$$

$$S_n = S_o + S_s \qquad \text{[EQ 33.12]}$$

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 33.8] and [EQ 33.9] and [EQ 33.10], [EQ 33.11] and [EQ 33.12] above. The value of each effective component viscosity is then used in turn in [EQ 33.10], [EQ 33.11] and [EQ 33.12] to yield an effective saturation fraction to be used in oil and solvent density calculations. The effective saturation fractions used for density calculations are

$$\left(\frac{S_o}{S_n}\right)_{oe} = \frac{\mu_{o\,\mathrm{eff}}^{1/4} \cdot \mu_o^{1/4} - \mu_s^{1/4} \cdot \mu_o^{1/4}}{\mu_{o\,\mathrm{eff}}^{1/4} \cdot (\mu_o^{1/4} - \mu_s^{1/4})} \qquad \text{[EQ 33.13]}$$

$$\left(\frac{S_o}{S_n}\right)_{se} = \frac{\mu_{s\,\mathrm{eff}}^{1/4} \cdot \mu_o^{1/4} - \mu_s^{1/4} \cdot \mu_o^{1/4}}{\mu_{s\,\mathrm{eff}}^{1/4} \cdot (\mu_o^{1/4} - \mu_s^{1/4})} \qquad \text{[EQ 33.14]}$$

The effective oil and solvent densities (ρ_{oeff}, ρ_{seff}) are now computed from the effective saturations fractions in expressions [EQ 33.13] and [EQ 33.14] and the pure component densities (ρ_o , ρ_s) using the following formulae:

$$\rho_{o\,\mathrm{eff}} = \rho_o\left(\frac{S_o}{S_n}\right)_{oe} + \rho_s\left[1 - \left(\frac{S_o}{S_n}\right)_{oe}\right] \qquad \text{[EQ 33.15]}$$

$$\rho_{s\,\mathrm{eff}} = \rho_o\left(\frac{S_o}{S_n}\right)_{se} + \rho_s\left[1 - \left(\frac{S_o}{S_n}\right)_{se}\right] \qquad \text{[EQ 33.16]}$$

The ECLIPSE model allows you to specify a different surface density for the solvent gas to that for the solution gas. In equations [EQ 33.15] and [EQ 33.16], 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 33.13] and [EQ 33.14] 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 effects in 2-D studies by setting $\omega=2/3$ independent 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 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):

$$\frac{1}{B_g} = \frac{1}{B_{gm}} M_p + \frac{1}{B_{gi}}(1 - M_p) \qquad \text{[EQ 33.17]}$$

$$\frac{1}{B\mu_g} = \frac{1}{B\mu_{gm}} M_p + \frac{1}{B\mu_{gi}}(1 - M_p) \qquad \text{[EQ 33.18]}$$

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:

$$B_{gm} = \frac{\mathrm{SD}_s}{R_{s\ \mathrm{eff}}} \qquad \text{[EQ 33.19]}$$

where

SD_s	= surface density of solvent
$R_{s\ \mathrm{eff}}$	= effective solvent density in [EQ 33.16].

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:

$$P_{cog} = P_{cog\ (\text{misc})} M_p + P_{cog\ (\text{immisc})}(1 - M_p) \quad \text{[EQ 33.20]}$$

$$P_{cog\ (\text{misc})} = P_{cog}(S_g)$$

$$P_{cog\ (\text{immisc})} = P_{cog}(S_g + S_s) \quad \text{[EQ 33.21]}$$

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 33.3] and [EQ 33.2] or [EQ 33.4] and [EQ 33.5]. 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 33.1.

Figure 33.1 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:

$$F_{ij,c} = T_{ij}\frac{K_{ci}}{B_{ci}\,\mu_{ci}}\mathrm{DP}_{ij,c} \qquad \text{[EQ 33.22]}$$

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 33.22] by a linearly extrapolated relative permeability, K_{ci}^*, obtained from cells i and k,

$$K_{ci}^* = \frac{2\mathrm{DX}_i + \mathrm{DX}_k}{\mathrm{DX}_i + \mathrm{DX}_k}K_{ci} - \frac{\mathrm{DX}_i}{\mathrm{DX}_i + \mathrm{DX}_k}K_{ck} \qquad \text{[EQ 33.23]}$$

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,

$$\mathrm{MIN}(K_{ci}, K_{cj}) < K_{ci}* < \mathrm{MAX}(K_{ci}, K_{cj}) \qquad \text{[EQ 33.24]}$$

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 33.3] and [EQ 33.2]). 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 33.6] and [EQ 33.7] with end points implied from equations [EQ 33.3] and [EQ 33.2] or [EQ 33.4] and [EQ 33.5].

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.

Multi-phase flash

Chapter 34

Introduction

x	ECLIPSE 100 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

MULTIPHA	Requests Multiphase option

PROPS section keywords

HYDRO	Define multi-phase components

SUMMARY section keywords

Solid properties can be output to the summary file. Keywords are given in Table 34.1

Table 34.1 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 34.2 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 MULTIPHA 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:

- the heaviest hydrocarbon is an aromatic (see the HYDRO keyword), and
- 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 MULTIPHA keyword is used, and if the heaviest hydrocarbon is an aromatic (see the HYDRO keyword).

Multi-segment Wells

Chapter 35

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. 80].

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:

- 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.
- 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.
- 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.
- 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:
    - Variable pressure loss multipliers can be used to represent 'smart' devices which react to isolate high GOR or WOR regions.
    - 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.
    - Other advanced well components such as sub-critical valves, 'labyrinth' devices and down-hole separators can also be modeled.
- 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.
- 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 35.1). 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—no loops are allowed.

Figure 35.1 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 35.2). 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.

Figure 35.2 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 35.3). 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.

Figure 35.3 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

$$q_{pj} = T_{wj} M_{pj} (P_j + H_{cj} - P_n - H_{nc}) \qquad \text{[EQ 35.1]}$$

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 35.4). 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 35.4 Hydrostatic head components

The pressure loss calculation

The following options are available for calculating the pressure drop:

- 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 941). 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 514).

- 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).

- 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.

- 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.

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.

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`.

- 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 504), 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.

- 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.

- 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

$$v_g = C_0 j + v_d \qquad \text{[EQ 35.2]}$$

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

$$j = v_{sg} + v_{sl} = \alpha_g v_g + (1 - \alpha_g) v_l \qquad \text{[EQ 35.3]}$$

where α_g is the gas volume fraction, averaged across the pipe area.

The average liquid flow velocity is thus

$$v_l = \frac{1 - \alpha_g C_0}{1 - \alpha_g} j - \frac{\alpha_g}{1 - \alpha_g} v_d \qquad \text{[EQ 35.4]}$$

Several of the one-dimensional slip correlations take the general form of [EQ 35.2]. 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. 28]. The relationships contain some adjustable parameters that can be tuned to fit observations. In [Ref. 28] the parameter values are optimized to fit a series of experiments performed in 15 cm (6 in) diameter tubing. 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 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. 24] 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. 26], 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

$$C_0 = \frac{A}{[1 + (A - 1)\beta^{*2}]} \qquad \text{[EQ 35.5]}$$

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

$$\beta^* = \frac{(\beta - B)}{(1 - B)} \text{ subject to } 0 \le \beta^* \le 1 \qquad \text{[EQ 35.6]}$$

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 35.12]. We thus choose the following expression for β

$$\beta = \max\left(\alpha_g, F_v \frac{\alpha_g |j|}{v_{sgf}}\right) \qquad \text{[EQ 35.7]}$$

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, but if the optimized parameter set is requested in keyword `WSEGDFMD` it defaults to 0.6. 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.

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

$$\frac{\partial}{\partial \alpha_g}(\alpha_g C_0) > 0 \quad \text{and} \quad \frac{\partial}{\partial j}(j C_0) > 0 \qquad \text{[EQ 35.8]}$$

Both these criteria will be satisfied if

$$B < \frac{2 - A}{A} \qquad \text{[EQ 35.9]}$$

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. 28]. 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

$$v_g(v_l = 0) = \frac{v_d}{1 - \alpha_g C_0} \qquad \text{[EQ 35.10]}$$

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

$$v_c = \left[\frac{\sigma_{gl}\, g\, (\rho_l - \rho_g)}{\rho_l^{\,2}}\right]^{1/4} \qquad \text{[EQ 35.11]}$$

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

$$v_g(v_l = 0) = K_u \left(\frac{\rho_l}{\rho_g}\right)^{1/2} v_c \qquad \text{[EQ 35.12]}$$

where K_u is the 'critical Kutateladze number', which is related to the dimensionless pipe diameter

$$D^* = \left[\frac{g(\rho_l - \rho_g)}{\sigma_{gl}}\right]^{1/2} D \qquad \text{[EQ 35.13]}$$

by the function in Table 35.1.

Table 35.1 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

$$v_d = \frac{(1 - \alpha_g C_0) C_0 K(\alpha_g) v_c}{\sqrt{\frac{\rho_g}{\rho_l}}\,\alpha_g C_0 + 1 - \alpha_g C_0} \qquad \text{[EQ 35.14]}$$

where

$$K(\alpha_g) = 1.53/C_0 \qquad \text{when } \alpha_g \le a_1$$

$$K(\alpha_g) = K_u \qquad \text{when } \alpha_g \ge a_2$$

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. If the optimized parameter set is requested in keyword `WSEGDFMD` the values are reset to 0.05 and 0.13 respectively.

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

$$v_d(\theta) = m(\theta)\; v_d\big|_{\theta = 0} \qquad \text{[EQ 35.15]}$$

where θ is the angle of inclination from the vertical and $v_d\big|_{\theta = 0}$ is given by [EQ 35.14].

By default ECLIPSE uses the inclination factor given by Hasan and Kabir [Ref. 32]

$$m(\theta) = (\cos\theta)^{0.5}(1 + \sin\theta)^2 \qquad \text{[EQ 35.16]}$$

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.

You can over-ride the default formula for the inclination factor by requesting 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. (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 is optimized to fit the experimental results described in [Ref. 28]. This is the recommended option if you select the optimized set of model parameters in 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.

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

$$\sigma_{gl} = \frac{\alpha_o \sigma_{go} + \alpha_w \sigma_{wg}}{\alpha_o + \alpha_w} \qquad \text{[EQ 35.17]}$$

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. 33]. 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. 32] to model the slip between oil and water in the liquid phase. This takes the form of

$$v_o = C_0' v_l + v_d' \qquad \text{[EQ 35.18]}$$

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, $\alpha_{ol} = \alpha_o/(\alpha_o + \alpha_w)$

$$C_0' = A \quad \text{when } (\alpha_{ol} \le B1)$$
$$C_0' = 1 \quad \text{when } (\alpha_{ol} \ge B2)$$
$$C_0' = A - (A-1)\left(\frac{\alpha_{ol} - B1}{B2 - B1}\right) \quad \text{when } (B1 < \alpha_{ol} < B2)$$

[EQ 35.19]

The values of the coefficients default to $A = 1.2$, $B1 = 0.4$ and $B2 = 0.7$. A condition equivalent to [EQ 35.8], that the oil superficial velocity should increase with the oil holdup fraction, requires

$$B1 < (2 - A)\, B2$$

[EQ 35.20]

The oil-liquid drift velocity is calculated as

$$v_d' = 1.53 v_c' (1 - \alpha_{ol})^2$$

[EQ 35.21]

where

$$v_c' = \left[\frac{\sigma_{ow}\, g\, (\rho_w - \rho_o)}{\rho_w^{\,2}}\right]^{1/4}$$

[EQ 35.22]

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 35.21] 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 optimized set of model parameters in keyword WSEGDFMD, ECLIPSE will use a different formulation for oil-water slip based on the results described in [Ref. 28]. In these experiments no profile slip was observed between the oil and water phases, and accordingly the parameter A in [EQ 35.19] is set equal to 1.0. The parameters $B1$ and $B2$ are redundant since C_0' is always 1.0. In place of [EQ 35.21] the oil-liquid drift velocity is calculated as

$$v_d' = 1.53 v_c' (1 - \alpha_{ol})^{n'}$$

[EQ 35.23]

where n' is set equal to 0.95.

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 35.15]. By default ECLIPSE uses Hasan and Kabir's relation [Ref. 32] as shown in [EQ 35.16]. 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. 28]. 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.

Friction

The calculation of the friction pressure loss is based on the formulation used in the correlation of Hagedorn and Brown [Ref. 34]

$$\delta P_f = \frac{C_f \, f \, L \, w^2}{A^2 \, D \, \rho} \qquad \text{[EQ 35.24]}$$

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 941), but with the Reynolds number formulated as

$$R_e = \frac{C_r \, D \, w}{A \, \mu} \qquad \text{[EQ 35.25]}$$

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).

$$\delta P_a = H_{v\text{out}} - \sum_{\text{inlets}} H_{v\text{in}} \qquad \text{[EQ 35.26]}$$

The velocity head of the mixture flowing through a junction is

$$H_v = \frac{0.5\, C_f\, w^2}{A^2\, \rho} \qquad \text{[EQ 35.27]}$$

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:

- An inlet segment, from which the wellstream enters the separator.
- A water or gas offtake segment, which should be the beginning of the lateral branch that injects the separated fluid back into the formation.
- 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:

- 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`).
- 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 35.5 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 35.5 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

$$\alpha_w^{of} = E + (1 - E)\alpha_w^{sep}$$

$$\alpha_g^{of} = (1-E)\alpha_g^{sep} \qquad \text{[EQ 35.36]}$$

$$\alpha_o^{of} = (1-E)\alpha_o^{sep}$$

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},

$$E = E_{max} \qquad \text{when } \alpha_w^{sep} > \alpha_{lim}$$

$$E = \frac{\alpha_w^{sep}}{\alpha_{lim}} E_{max} \qquad \text{when } \alpha_w^{sep} < \alpha_{lim} \qquad \text{[EQ 35.37]}$$

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

$$Q_o^{of} = A\,j\,(\alpha_o^{of} b_o^{sep} + R_v^{sep} \alpha_g^{of} b_g^{sep})$$

$$Q_w^{of} = A\,j\,\alpha_w^{of} b_w^{sep} \qquad \text{[EQ 35.38]}$$

$$Q_g^{of} = A\,j\,(\alpha_g^{of} b_g^{sep} + R_s^{sep} \alpha_o^{of} b_o^{sep})$$

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

$$\alpha_g^{of} = E + (1 - E)\alpha_g^{sep}$$

$$\alpha_w^{of} = (1-E)\alpha_w^{sep} \qquad \text{[EQ 35.39]}$$

$$\alpha_o^{of} = (1-E)\alpha_o^{sep}$$

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 35.40]

Behavior of the model

An oil/water separation system powered by electric submersible pumps is described in [Ref. 81]. 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:

- 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.
- 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.
- 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
/
```

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 528).

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. 82].

Energy inflow

The flow of energy between a grid block and its associated segment's node is given by the inflow performance relationship

$$q_{ej} = T_{wj} \cdot \sum_{p} M_{epj} \cdot (P_j + H_{cj} - P_n - H_{nc}) \qquad \text{[EQ 35.41]}$$

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 35.1].

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

$$Q = -K \cdot A \cdot \frac{\partial T}{\partial X} \qquad \text{[EQ 35.42]}$$

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 35.42] can be integrated over the radius to give

$$Q = \frac{L}{R} \cdot (T_i - T_o) \qquad \text{[EQ 35.43]}$$

where the specific thermal resistance of the annulus is given by

$$R = \frac{1}{2\pi} \cdot \frac{1}{K} \cdot \ln\frac{r_o}{r_i}, \qquad \text{[EQ 35.44]}$$

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 a thin annulus is calculated by substituting

$$r_o = r_i + \Delta r$$

into [EQ 35.44] to give

$$R = \frac{1}{2\pi} \cdot \frac{\Delta r}{K} \cdot \frac{1}{r_i} \qquad \text{[EQ 35.45]}$$

Heat transfer along the well

[EQ 35.42] 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 35.43] 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 35.6 Heat conduction between a well an the formation

The thermal resistance for these cases is defined by summing terms similar to [EQ 35.44] and [EQ 35.45], see Prats, Chapter10:

$$R_h = \frac{1}{2\pi}\left[\frac{1}{h_f \cdot r_i} + \frac{1}{h_{P_i} \cdot r_i} + \frac{\ln(r_o/r_i)}{\lambda_P} + \ldots\right.$$

$$\ldots + \frac{1}{h_{P_o} \cdot r_o} + \frac{\ln(r_{ins}/r_o)}{\lambda_{ins}} + \frac{1}{h_{rc_{an}} \cdot r_{ins}} + \ldots$$

$$\ldots + \frac{\ln(r_{co}/r_{ci})}{\lambda_P} + \frac{\ln(r_w/r_{co})}{\lambda_{cem}} + \frac{\ln(r_{ea}/r_w)}{\lambda_{ea}} + \frac{f(t_D)}{\lambda_E}\Bigg]$$

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 35.43] 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 35.7 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 35.8), since the flow is modeled as entering the well segment at its node.

Figure 35.8 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 498).

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.

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 506). You can change the default values of these parameters for all well segments using keyword `WSEGDFPA`. 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 511).

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 503). 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. 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, i.e. "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.

ECLIPSE 300 Thermal Option

Heat loss

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 35.9 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 950). 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 950). 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

ECLIPSE 100

Multi-segment wells can require substantially more well iterations to converge them than standard wells require, 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 501.

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 timestep. 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:

- The Wellbore Friction option
- The Polymer Flood model
- Brine Tracking
- The Temperature option
- The Solvent model
- The Gi model
- Automatic Refinement in the LGR option
- The `COMPFLSH` keyword.

ECLIPSE 300

Multi-segment wells in ECLIPSE 300 cannot at present be used in conjunction with the following facilities:

- velocity dependent gas relative permeability contribution to D-factor (`VDKRG`)
- changing well dimensions in the `SCHEDULE` section (`CHANDIMS`)
- k-value tables in compositional mode (`KVALUES`)
- 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 941) 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:

- There is an enforced one-to-one correspondence between segments and grid block connections. This may result in an excessively large number of segments.
- 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.
- 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.
- 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.

To convert 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.

- The maximum number of multi-segment wells in the run is the same as the maximum number of friction wells.
- The maximum number of segments per well is be 1 + the maximum number of grid block connections per well.
- 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

WSEGDIMS	Sets array dimensions for multi-segment wells

PROPS section

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 35.2 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

Table 35.2 Multi-segment wells SUMMARY output controls (Continued)

E100	E300	Segment	Information
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)
	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)

Table 35.2 Multi-segment wells SUMMARY output controls (Continued)

E100	E300	Segment	Information
	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)

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     /
/
```

SCHEDULE section

	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.
ECLIPSE 300 only	OPTIONS3	Switch 34 may help convergence in difficult situations involving crossflow.
	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.
	WSEGDFIN	Inputs a table of inclination scaling factors versus deviation angle for the Drift Flux slip model
	WSEGDFMD	Defines the default values for the set of parameters in 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.
	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.
	WSEGTABL	Designates segments to have their pressure drops calculated from a VFP table.
	WSEGVALV	Defines segments to represent a sub-critical valve.

The Network Option

Chapter 36

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 36.1). 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 36.2 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 36.2. The actual manifold pressure and flow rate of the group will be given by the intersection of these two curves.

Figure 36.1 common wellhead manifold and pipeline

Figure 36.2 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 36.3. 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 36.3 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 552).

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 555.) 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 555.) 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 "IAutomatic compressors" on page 554).

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 36.3, 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 552).

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 555). 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.

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 36.3, for example, is constructed by the following records

```
GRUPTREE
 'GR-A1' 'PLAT-A' /
 'GR-A2' 'PLAT-A' /
 'GR-B1' 'PLAT-B' /
 'GR-B2' 'PLAT-B' /
/
```

GRUPNET Define the production network. For example, in Figure 36.3, 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

```
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 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 Define an injection network. For example, in Figure 36.3, 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

```
GNETINJE
-- GROUP   NETWORK   FIXED      VFP
-- NAME    PHASE     PRESSURE   TAB
 'PLAT-B' 'WATER'   14.7           /
 'GR-B2'  'WATER'   1*          2  /
/
```

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

```
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' /
  /
```

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:

```
GNETPUMP
-- GROUP    RATE    PHASE    NEW VFP   NEW LIFT  NEW GAS
-- NAME     LIMIT            TAB NO    QUANTITY  CONSUMPTION
   'GR-B1' 500     'OIL'    1*        50                  /
   /
```

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)

```
NETBALAN
-1.0 /
```

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)

```
WAITBAL
'YES' /
```

WNETDP

Optional keyword to apply an additional fixed pressure drop between a well's tubing head and its group's node in the network

```
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:

- 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

- 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.
- 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.
- It provides more flexibility in handling well and group downtime.
- 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 36.4. 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 36.4 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 36.5, which is different from the network structure.

Figure 36.5 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 36.4 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.

IAutomatic compressors

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 540).

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 36.4 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 36.4 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 36.4, 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'    /
/
```

ECLIPSE 100

Define any gas consumption or import with either GCONSUMP or NCONSUMP

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.

ECLIPSE 300

Remove any fuel gas from the network with NGASREM

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 36.4 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 then 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 555). 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

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 36.1 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.

Table 36.2 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

Table 36.2 Network SUMMARY output controls (ECLIPSE 300) (Continued)

Group	Information
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

	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.
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 37

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 (the discretization stencil uses 9 cells in a 3*3 patch in two dimensions, or 27 cells in a 3*3*3 patch in three dimensions).

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 819 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. 37]. 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 37.1 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 37.2 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. 39] 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. However, by using the NINENUM keyword in the GRID section, you can choose to apply the scheme only in nominated regions of the reservoir.

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 38

Introduction

x	ECLIPSE 100 ECLIPSE 300 SPECIAL

Velocity-dependent flow is a non-Darcy effect. The usual Darcy flow condition is:

$$q = \left(\frac{KA}{\mu}\right)\frac{dP}{dx} \qquad \text{[EQ 38.1]}$$

or

$$\frac{dP}{dx} = \left(\frac{\mu}{KA}\right)q \qquad \text{[EQ 38.2]}$$

where:

q	is the volumetric flow rate
K	is the permeability
A	is the area through which flow occurs
μ	is the viscosity
dP/dx	is the pressure gradient normal to the area

The above form is correct in cgs units - that is if K is measured in Darcys, q in cm^3/s, A in cm^2, μ in cP, dP/dx in atm/cm.

$$1D = 1\ cm^3.s^{-1}.cP/(atm.cm^{-1}.cm^2) \qquad \text{[EQ 38.3]}$$

$$= 1\ cP.cm^2.atm^{-1}s^{-1}$$

$$= 0.001\ Pa.s.cm^2/(101325\ Pa.s)$$

$$= 9.86923.10^{-9}\ cm^2$$

Permeability has the dimensions of [length]^2, viscosity the dimensions of [pressure.time] with 1 cP = 0.001 Pa.s, and 1 atm = 1.01325.10^5 Pa.

If we wish to use another set of units, such as one of those used in ECLIPSE 300, we must include a suitable constant:

$$q = C_1\left(\frac{KA}{\mu}\right)\frac{dP}{dx} \qquad \text{[EQ 38.4]}$$

For example, if we use K in mD, then C_1 would be (D/mD)=1000, effectively converting back to Darcys.

In the `FIELD` unit set we have q in RB/D, K in mD, A in ft.^{2}, dP/dx in psi/ft. Then C_1 is:

C_1 = (D/mD) (cm^2/ft.^{2}) (atm/psi) (ft./cm) (RB/cm^3)(s/day)

= (D/mD) (atm/psi) (RB/ft^3) (ft^2/cm^2)(s/day)

= (0.001) (1/14.696) (0.1781076)(0.001076391)(86400.0)

= 0.00112712

This is the form used in normal flow calculations in ECLIPSE 300.

When we consider non-Darcy flow, we follow Holditch and Morse in adding an extra velocity-dependent term to Darcy's law. Returning to cgs units we have:

$$\frac{dP}{dx} = \left(\frac{\mu}{KA}\right)q + \beta\rho\left(\frac{q}{A}\right)^2 \qquad \text{[EQ 38.5]}$$

where:

ρ	is the fluid mass density
(q/A)	is the fluid velocity.

We express this in the form

$$\frac{dP}{dx} = \left(\frac{\mu}{KA}\right)q\left(1 + \frac{\beta\rho qK}{A\mu}\right) \qquad \text{[EQ 38.6]}$$

$$q = \left(\frac{KA}{\mu}\right)\left(\frac{dP}{dx}\right)\Big/\left(1 + \frac{\beta\rho qK}{A\mu}\right) = \left(\frac{KA}{\mu}\right)\left(\frac{dP}{dx}\right)F_{ND} \qquad \text{[EQ 38.7]}$$

with the *non-Darcy flow* factor given by:

$$F_{ND} = \frac{1}{1 + \frac{\beta\rho qK}{A\mu}} \qquad \text{[EQ 38.8]}$$

where β is the user input non-Darcy flow coefficient, in units of atm.s^2.g^{-1} = 1 F (Forchheimer). The units of the group are then:
Atm.s^2 .g^{-1} g.cm^{-3} cm^3.s^{-1} cP.cm^2.Atm^{-1}.s^{-1} cm^{-2}cP^{-1}, which is dimensionless, as it should be.

An alternative unit for β can be obtained by noting that:

1F = 1 Atm.s2.g-1

$= 1.01325.10^6$ dyne.cm^{-2}.s^2.g^{-1}

$= 1.01325.10^6$ g.cm.s^{-2}.cm^{-2}.s^2.g^{-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. 5] shows a typical value of $\beta = 10^7$cm^{-1} at $K = 60mD$; that is, 9.86 F.

As for Darcy flow, we must modify the expression for F_{ND} in the unit sets used by ECLIPSE 300.

- In `FIELD` units, we have:

$$F_{ND} = \frac{1}{1 + \frac{C_2 \beta \rho q K}{A \mu}}$$

where:

ρ	is the phase density lb/ft^3
q	is the flow rate in RB/day
K	is the permeability in mD
A	is the area in ft^2

Thus the constant C_2 is:

C_2 = (g/cm^3)(lb/ft.^{3})(cm^3/s) (RB/day) (D/mD) (ft.^{2}/cm^2)

= (day/s) (g/lb) (ft.^{3}/RB) (ft.^{2}/cm^2) (D/mD)

= 1/(24.3600)(453.592)(5.614583)(0.001076391)(0.001)

$= 3.172774512.10^{-8}$

- In METRIC and PVT-METRIC units, we have:

$$
\begin{aligned}
C_2 &= \frac{(\mathrm{g}/\mathrm{cm}^3)}{(\mathrm{kg}/\mathrm{m}^3)}\frac{(\mathrm{cm}^3/s)}{(\mathrm{m}^3/\mathrm{day})}(\mathrm{D}/\mathrm{mD})(\mathrm{m}^2/\mathrm{cm}^2) \\
&= (\mathrm{day}/\mathrm{s})(\mathrm{g}/\mathrm{kg})(\mathrm{m}^2/\mathrm{cm}^2)(\mathrm{D}/\mathrm{mD}) \\
&= (1/(24 \cdot 3600))(0.001)(10000)(1000) \\
&= 1.1574074 \cdot 10^{-9}
\end{aligned}
$$

- In LAB units:

$$
\begin{aligned}
C_2 &= \frac{(\mathrm{cm}^3/\mathrm{s})}{(\mathrm{cm}^3/\mathrm{hr})}(\mathrm{D}/\mathrm{mD}) = (\mathrm{hr}/\mathrm{s})(\mathrm{D}/\mathrm{mD}) \\
&= \left(\frac{1}{3600}\right)(0.001) = 2.77777778 \cdot 10^{-7}
\end{aligned}
$$

Construction of flows

As velocity is a two-point quantity, we must construct this outside domains. As flow velocity is a function of F_{ND} and as F_{ND} is a function of flow velocity, we have two coupled equations to solve for phase p:

$$F_{ND} = \frac{1}{1 + \frac{C_2 \beta \rho_p q K}{A \mu_p}} \quad \text{[EQ 38.9]}$$

$$q = F_{ND} T \left(\frac{K_{rp}}{\mu_p} \right) \Delta\phi_p \quad \text{[EQ 38.10]}$$

Substitute for q from the second equation:

$$F_{ND} = \frac{1}{1 + \frac{C_2 \beta \rho F_{ND} T (K_{rp} / \mu_p) \Delta\phi_p K}{A \mu_p}} \quad \text{[EQ 38.11]}$$

Let:

$$B = \left(\frac{C_2 \beta T K}{A} \right) (\Delta\phi_p) \left(\frac{\rho_p K_{rp}}{\mu_p^2} \right) \quad \text{[EQ 38.12]}$$

The factor $(C_2 \beta T K / A)$ is fixed, the remainder depends upon the phase p and the state of the fluids during the simulation.

This is a quadratic equation for F_{ND}, which we can solve:

$$F_{ND} = \frac{1}{1 + B F_{ND}} \quad \text{[EQ 38.13]}$$

$$B F_{ND}^2 + F_{ND} - 1 = 0 \quad \text{[EQ 38.14]}$$

$$F_{ND} = \frac{-1 + \sqrt{(1 + 4B)}}{2B} \quad \text{[EQ 38.15]}$$

The non-Darcy flow coefficient can be input by using either the `VDFLOW` or `VDFLOWR` keywords in the `PROPS` section.

The program will print out the maximum deviation of the non-Darcy factor from unity at the end of each time step.

Velocity-dependent relative permeabilities

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 and VDKRG respectively.

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 modes 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 j = (oil, gas), the model (1) CN is calculated from:

$$N_{cj}^{(1)} = \frac{v_g \mu_g}{\sigma} \qquad \text{[EQ 38.16]}$$

whereas the model (2) CN is calculated from:

$$N_{cj}^{(2)} = \frac{K K_{rvj} \Delta P_j}{\sigma L} \qquad \text{[EQ 38.17]}$$

and the model (3) CN from:

$$N_{cj}^{(3)} = (2 \phi S_j K K_{rvj})^{1/2} \frac{\Delta P_j}{\sigma} \qquad \text{[EQ 38.18]}$$

where:

K is the absolute permeability

L is the cell-length in the direction of flow

ϕ is the cell porosity

S_j is the normalized phase saturation (fraction of HydroCarbon Pore Volume, or HCPV)

ΔP_j is the pressure drop of the jth 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_{rvj} is the CN-modified relative permeability of the jth phase in the direction of flow at the previous timestep. We will discuss this last point in more detail shortly.

Note that the model (1) CN for the oil and gas phases depends on the gas velocity and viscosity: this apparent discrepancy has been verified by experiment.

Given a phase CN calculated from [EQ 38.16], [EQ 38.17] or [EQ 38.18], we now calculate a phase Normalized Capillary Number (NCN) from:

$$N_{cnj} = \frac{N_{cbj}}{N_{cj}} \qquad \text{[EQ 38.19]}$$

where N_{cbj} 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 38.16] 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_j, where:

$$X_j = 1 - \exp(-m_j N_{cnj}) \quad \text{[EQ 38.20]}$$

where m_j is, **and must be**, an experimentally determined parameter. Therefore, given a phase residual saturation S_{rbj} determined from routine Special Core AnaLysis (SCAL), the effect of the CN dependency is to change the residual saturation to:

$$S_{rbj} \rightarrow X_j S_{rbj} \quad \text{[EQ 38.21]}$$

Note if the m_j-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:

$$K_{rmj} = \frac{(S_j - X_j S_{rbj})}{(1 - X_j S_{rbj})} \quad \text{[EQ 38.22]}$$

where S_j is the normalized or HCPV saturation of the jth phase, which is related to the true saturation $S_j^{(3P)}$ by:

$$S_j = \frac{S_j^{(3P)}}{1 - S_w} \quad \text{[EQ 38.23]}$$

where S_w is the water saturation.

The user-input base relative permeability curves, K_{rbj}, are modified because of the change in residual saturation through [EQ 38.21]. The CN-modified relative permeability curve for phase j is now calculated from:

$$K_{rvj} = N_{cnj}^{1/n_j} K_{rbj} + \left(1 - N_{cnj}^{1/n_j}\right) K_{rmj} \quad \text{[EQ 38.24]}$$

where n_j is given by:

$$n_j = n_{1j} S_j^{n_{2j}} \quad \text{[EQ 38.25]}$$

and (n_{1j}, n_{2j}) are, **and must be**, experimentally determined parameters.

Explicit calculation of capillary number dependency

From [EQ 38.16], [EQ 38.17] and [EQ 38.18], we see that the CN is a function of the phase relative permeability K_{rvj}: note [EQ 38.16] and [EQ 38.17] are effectively equivalent assuming the 2nd expression is only applied to the gas phase. However, from [EQ 38.20], [EQ 38.22] and [EQ 38.24], we see K_{rvj} 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 timestep. 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.

Forchheimer model

At high gas velocities, in addition to the benefit derived from the CN effect, there is the detrimental effect due to turbulence, expressed through the Forchheimer parameter, β_j. The Forchheimer parameter is applied as per the basic non-Darcy flow model discussed previously. The only difference here is the mechanism by which the β_j parameters are calculated.

Two models for calculating β_j are available. The model (1) expression is:

$$\beta_j = \frac{a_j}{\phi^{b_j} S_j^{c_j} (KK_{rvj})^{d_j}} \qquad \text{[EQ 38.26]}$$

whereas the model (2) expression is:

$$\beta_j = \beta_{dj} S_j^{c_j} (KK_{rvj})^{d_j} \qquad \text{[EQ 38.27]}$$

where the set of parameters, (a_j, b_j, c_j, d_j) or (β_{dj}, c_j, d_j) are, **and must be**, experimentally determined.

There appears to be no real physical basis for the porosity and saturation dependency in [EQ 38.26] and saturation dependency in [EQ 38.27] other than in the original work done on the Forchheimer effect. Due account was taken of the HydroCarbon Pore Space. Therefore, it would seem appropriate to re-define these expressions as:

$$\beta_j = \frac{\tilde{a}_j}{(KK_{rvj})^{d_j}} \qquad \text{[EQ 38.28]}$$

and:

$$\beta_j = \tilde{\beta}_{dj} (KK_{rvj})^{d_j} \qquad \text{[EQ 38.29]}$$

where:

$$\tilde{a}_j = \frac{a_j}{\phi_0^{b_j}(1 - S_{w0})^{c_j}} \qquad \text{[EQ 38.30]}$$

and:

$$\tilde{\beta}_{dj} = \beta_j(1 - S_{w0})^{c_j} \qquad \text{[EQ 38.31]}$$

where (ϕ_0, S_{w0}) are the initial porosity and water saturation: appropriate values for (b_j, c_j) would appear to be $(5.5, 5.5)$, respectively. If the forms [EQ 38.30] and [EQ 38.31] are used, the values of the (b_j, c_j) entered through the VDKRO and/or VDKRG keywords in the PROPS section should both be zero.

Well D-factors

The Forchheimer parameter for the gas phase is also used to calculate a well D-factor for each well connection from

$$D = \alpha \cdot \beta \cdot \frac{k}{h} \cdot \frac{1}{r_w} \cdot \frac{\gamma_g}{\mu_g} \qquad \text{[EQ 38.32]}$$

where

α	is a units conversion factor
β_g	is the Forchheimer parameter for gas
k	is the effective permeability of the connected grid block
h	length of the connection
r_w	wellbore radius
γ_g	gas mole weight
μ_g	gas viscosity

This calculated value overrides D-factors supplied via COMPDAT or WDFAC. If the WDFACCOR keyword is specified however, then the value calculated from its correlation will override the value calculated from [EQ 38.32]. The calculated D-factor for a well connection may be written to the summary file using the mnemonic CDFAC.

Parallel option

Chapter 39

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 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.

- 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.
- 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.
- 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 39.1]

Theoretical parallel CPU time: $\left(\frac{1-S}{NP} + \frac{S}{NP-1}\log_2(NP)\right)CPU_{serial}$ [EQ 39.2]

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:

- Load balancing, where the amount of work on each processor is not exactly equal.
- 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 39.1. Note that this keyword also contains partitions for the 2 and 3 processor cases.

Table 39.1 The division of cells between processors using the PSPLITX keyword.

Process	Cells
1	1 to 40
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.

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.

If there is only a small number of LGRs, or one LGR is particularly large, you may optionally specify that one or more of the local grids is to be processed in parallel. This can only be done for Cartesian local grid refinements. This facility is activated using item 13 of the `CARFIN` keyword. It can be used in conjunction with amalgamations.

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.

- 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.
- 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.
- 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.
- 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.
- 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.
- 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 timesteps. 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 timestep. 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.

- 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

The MEMORY keyword specifies the memory requirements of each processor. These 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.

If Parallel ECLIPSE 300 is being run on a distributed memory machine, then for any parallel performance to be achieved, all the processes within the parallel job will have dedicated processors. In this case a legitimate option is to set the memory requirements to the maximum available per node. In contrast, on a shared memory machine it is advisable to set the memory requirements to something close to the minimum required, so as not to consume resources that could be available to other jobs.

Normally ECLIPSE 100 will not require the use of the MEMORY keyword. If it is used it behaves in the same way as for ECLIPSE 300.

Two arguments are needed for the MEMORY keyword: the first requests real memory in MB, and the second requests character storage in K CHAR*8.

Example

This run requires 200 Megabytes of real storage and 20 thousand CHAR*8 strings.

```
MEMORY
200 20 /
```

Summary of keywords

RUNSPEC section

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

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

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 40

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:

- 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).
- The rock is isotropic.
- There is a homogeneous mineral modulus.
- 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. 88].

The Petro-elastic model can be decomposed into four levels. These are:

- "Level 1: Impedances and velocities" on page 595
- "Level 2: Effective bulk/shear modulus and density" on page 596
- "Level 3: Rock properties and fluid properties" on page 597
- "Level 4: Individual fluid phase properties" on page 600

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:

- I_P - the impedance to pressure waves for a particular simulation gridcell
- I_S - the impedance to shear waves for a particular simulation gridcell
- ν - the Poisson's ratio for a particular simulation gridcell

These are defined as

$$I_P = \rho V_P \qquad \text{[EQ 40.1]}$$

$$I_S = \rho V_S \qquad \text{[EQ 40.2]}$$

$$\nu = \frac{\frac{1}{2}V_P^2 - V_S^2}{V_P^2 - V_S^2} \qquad \text{[EQ 40.3]}$$

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

$$V_P = \sqrt{\frac{K + (4G/3)}{\rho}} \qquad \text{[EQ 40.4]}$$

$$V_S = \sqrt{\frac{G}{\rho}} \qquad \text{[EQ 40.5]}$$

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 594.

The following form of Gassmann's equation is used in the Petro-elastic model:

$$K = K_{frame} + \frac{\left(1 - \frac{K_{frame}}{K_{minerals}}\right)^2}{\frac{\phi_{eff}}{K_{fluid}} + \frac{(1-\phi_{eff})}{K_{minerals}} - \frac{K_{frame}}{(K_{minerals})^2}} \qquad \text{[EQ 40.6]}$$

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

$$G = G_{frame} \qquad \text{[EQ 40.7]}$$

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

$$\rho = \phi_{eff} \cdot \rho_{fluid} + (1 - \phi_{eff}) \cdot \rho_{minerals}$$ [EQ 40.8]

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.

$$\phi_{eff} = \phi_0\left(1 + X + \frac{X^2}{2}\right)$$ [EQ 40.9]

where

$$X = C(P - P_{ref})$$ [EQ 40.10]

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,

$$\rho_{fluid} = S_o\rho_o + S_w\rho_w + S_g\rho_g \qquad \text{[EQ 40.11]}$$

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

$$\frac{1}{K_{fluid}} = \frac{S_o}{K_o} + \frac{S_w}{K_w} + \frac{S_g}{K_g} \qquad \text{[EQ 40.12]}$$

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

$$K_{fluid} = (K_{liquid} - K_g)(1 - S_g)^e + K_g \qquad \text{[EQ 40.13]}$$

where

- e is the "patchy saturation" exponent
- K_{liquid} is the combined bulk modulus of the liquid phases, given by

$$\frac{S_o + S_w}{K_{liquid}} = \frac{S_o}{K_o} + \frac{S_w}{K_w} \qquad \text{[EQ 40.14]}$$

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. 89].

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

$$K_{minerals} = C_1 \phi_{eff} + C_0 \qquad \text{[EQ 40.15]}$$

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

$$(K_{frame})^{\alpha} = C_K (K_{minerals})^{\alpha} + \sum_{i=0}^{7} \left\{ C_{K,i}(P) \cdot \phi_{eff}^{i} \right\} \qquad \text{[EQ 40.16]}$$

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

$$(G_{frame})^{\alpha} = C_G (G_{minerals})^{\alpha} + \sum_{i=0}^{7} \left\{ C_{G,i}(P) \cdot \phi_{eff}^{i} \right\} \qquad \text{[EQ 40.17]}$$

where

- α is the same exponent as specified in [EQ 40.16]
- 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. 88].

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

$$K_o = \rho_o V_o^2 \qquad \text{[EQ 40.18]}$$

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

$$V_o = 2096\sqrt{\frac{\rho_{o,R}}{2.6-\rho_{o,R}}} - (3.7\cdot T) + (4.64\cdot P)$$

$$+ 0.0115\cdot\left\{\left(4.12\sqrt{\frac{1.08}{\rho_{o,R}}-1}\right)-1\right\}\cdot T\cdot P \qquad \text{[EQ 40.19]}$$

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

$$\rho_{o,P} = \rho_o\{0.972 + 3.81\times 10^{-4}(T+17.78)^{1.175}\} \qquad \text{[EQ 40.20]}$$

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

$$K_w = \rho_w V_w^2 \qquad \text{[EQ 40.21]}$$

where

- V_w is the velocity of sound waves in water in the gridcell given (in m/s) by

$$V_w = V_{pw} + C_S \cdot S + (780 - 10P + 0.16P^2) \cdot S^{1.5} - 1820S^2 \qquad \text{[EQ 40.22]}$$

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

$$C_S = 1170 - 9.6T + 0.055T^2 - 8.5\times10^{-5}T^3 + 2.6P - 0.0029TP - 0.0476P^2$$

where T is in ^{o}C, and

$$V_{pw} = \sum_{i=0}^{4} \left\{ \sum_{j=0}^{3} (w_{ij}T^iP^j) \right\} \qquad \text{[EQ 40.23]}$$

where

$$
\begin{aligned}
w_{00} &= 1402.85 & w_{02} &= 3.437\times10^{-3} \\
w_{10} &= 4.871 & w_{12} &= 1.739\times10^{-4} \\
w_{20} &= -0.04783 & w_{22} &= -2.135\times10^{-6} \\
w_{30} &= 1.487\times10^{-4} & w_{32} &= -1.455\times10^{-8} \\
w_{40} &= -2.197\times10^{-7} & w_{42} &= 5.230\times10^{-11} \\
w_{01} &= 1.524 & w_{03} &= -1.197\times10^{-5} \\
w_{11} &= -0.0111 & w_{13} &= -1.628\times10^{-6} \\
w_{21} &= 2.747\times10^{-4} & w_{23} &= 1.237\times10^{-8} \\
w_{31} &= -6.503\times10^{-7} & w_{33} &= 1.327\times10^{-10} \\
w_{41} &= 7.987\times10^{-10} & w_{34} &= -4.614\times10^{-13}
\end{aligned}
\qquad \text{[EQ 40.24]}
$$

Bulk modulus of gas

The bulk modulus of gas in a gridcell is given (in MPa) by

$$K_g = \frac{P\gamma}{1-\left(\frac{P_r}{Z}\cdot f\right)} \qquad \text{[EQ 40.25]}$$

where

$$GRAV = \frac{\rho_{gas|surface}}{\rho_{air|surface}}$$

$$P_r = \frac{P}{4.892-(04048\cdot GRAV)}$$

$$T_r = \frac{T+273.15}{94.72+(170.75\cdot GRAV)}$$

$$f = c\cdot d\cdot m + a$$

$$Z = (a\cdot P_r)+b+(c\cdot d)$$

$$a = 0.03+0.00527(3.5-T_r)^3$$

$$b = 0.642T_r - 0.007T_r^4 - 0.52$$

$$c = 0.109(3.85 - T_r)^2$$

$$d = \exp\left\{-\left[0.45 + 8\left(0.56 - \frac{1}{T_r}\right)^2\right]\frac{P_r^{1.2}}{T_r}\right\}$$

$$m = -1.2\left\{0.45 + 8\left(0.56 - \frac{1}{T_r}\right)^2\right\}\frac{P_r^{0.2}}{T_r}$$

$$\gamma = 0.85 + \frac{5.6}{P_r + 2} + \frac{27.1}{(P_r + 3.5)^2} - 8.7\exp\{-0.65(P_r + 1)\}$$

$GRAV$ 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 40.22]
- T - the temperature in [EQ 40.19], [EQ 40.22], and [EQ 40.25] (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 40.8]
- C_1 - the first order coefficient of effective porosity in the calculation of the effective bulk modulus of the minerals in [EQ 40.15]
- C_0 - the zeroth order coefficient of effective porosity in the calculation of the effective bulk modulus of the minerals in [EQ 40.15]
- C_K is the coefficient of the $K_{minerals}$ term in the calculation of K_{frame} in [EQ 40.16]
- C_G - the coefficient of the $G_{minerals}$ term in the calculation of G_{frame} in [EQ 40.17]
- $G_{minerals}$ - the effective shear modulus of the minerals in [EQ 40.7]
- α - the exponent in the calculation of the effective bulk/shear modulus of the dry rock frame in [EQ 40.16] and [EQ 40.17]
- e - the "patchy saturation" exponent in [EQ 40.13] (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 40.16]
- $C_{G,i}(P)$ - the tabular functions of pressure in the calculation of the effective shear modulus of the dry rock frame in [EQ 40.17]

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 605) and request gradients. For further information on requesting gradients, see "The Gradient Option" on page 344.

Summary of keywords

RUNSPEC section

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

PENUM	Specifies the petro-elastic region number for each gridcell in the simulation model.

PROPS section

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

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 the table below.

Note For each of these properties, you must also request the DEN item in the RPTRST keyword.

.

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	

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 the table below

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 41

Introduction

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL

Polymer flood processes

The main objective of polymer injection during water flooding of oil reservoirs is to decrease the mobility of the injected water. This decreases 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 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. ECLIPSE assumes that the dead pore space is constant for each rock type and does not exceed the corresponding irreducible saturation.

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 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:

$$\frac{d}{dt}\left(\frac{VS_w}{B_r B_w}\right) = \sum\left[\frac{Tk_{rw}}{B_w\ \mu_{w\ \mathrm{eff}} R_k}(\delta P_w - \rho_w g D_z)\right] + Q_w \qquad \text{[EQ 41.1]}$$

$$\frac{d}{dt}\left(\frac{VS_w{}^* C_p}{B_r B_w}\right) + \frac{d}{dt}\left(V\rho_r C_a \frac{1-\phi}{\phi}\right) = \sum\left[\frac{Tk_{rw} C_p}{B_w\ \mu_{p\ \mathrm{eff}} R_k}(\delta P_w - \rho_w g D_z)\right] + Q_w C_p \qquad \text{[EQ 41.2]}$$

$$\frac{d}{dt}\left(\frac{VS_w C_n}{B_r B_w}\right) = \sum\left[\frac{Tk_{rw} C_n}{B_w\ \mu_{s\ \mathrm{eff}} R_k}(\delta P_w - \rho_w g D_z)\right] + Q_w C_n \qquad \text{[EQ 41.3]}$$

$$S_w{}^* = S_w - S_{\mathrm{dpv}} \qquad \text{[EQ 41.4]}$$

where

S_{dpv}	denotes the dead pore space within each grid cell
C_a	denotes the adsorption isotherm which is a function of the local polymer solution 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 local concentration of polymer and sodium chloride in the aqueous phase
μ_{eff}	denotes the effective viscosity of the water, polymer and salt components.
D_z	is the cell center depth.

The remaining symbols have their usual meanings.

The model makes the assumption that the density and formation volume factor of the aqueous phase are independent of the local polymer and sodium chloride concentrations. The polymer solution, reservoir brine and the injected water are represented in the model as miscible components of the aqueous phase, where the degree of mixing is specified through the viscosity terms in the conservation equations.

The principal effects of polymer and brine on the flow of the aqueous phase are represented by equations [EQ 41.1] to [EQ 41.4] 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 41.2]. The adsorption term requires the user to specify the adsorption isotherm, C_a as a function of the local polymer concentration for each rock species. 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 by the ECLIPSE polymer model are a discretized form of the differential equations [EQ 41.1] - [EQ 41.4]. In order to avoid numerical stability problems which could be triggered by strong changes in the aqueous phase properties over a timestep (resulting from large changes in the local polymer/sodium chloride concentrations) a fully implicit time discretization is used. The ECLIPSE model 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. You are required to enter the viscosity of a fully mixed polymer solution as an increasing function of the polymer concentration in solution ($\mu_m(C_p)$). 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

$$\mu_{p,\ \mathrm{eff}} = \mu_m(C_p)^{\omega} \cdot \mu_p^{1-\omega} \qquad \text{[EQ 41.5]}$$

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),

$$\mu_{w,e} = \mu_m(C_p)^{\omega} \cdot \mu_w^{1-\omega} \qquad \text{[EQ 41.6]}$$

In order to calculate the effective water viscosity to be inserted into [EQ 41.1], 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 41.1]:

$$\frac{1}{\mu_{w,\,\mathrm{eff}}} = \frac{1-\bar{C}}{\mu_{w,e}} + \frac{\bar{C}}{\mu_{p,\,\mathrm{eff}}} \qquad \text{[EQ 41.7]}$$

$$\bar{C} = \frac{C_p}{C_{p,\,\max}} \qquad \text{[EQ 41.8]}$$

where

$\bar{C}$ is the effective saturation for the injected polymer solution within the total aqueous phase in the cell.

If the salt-sensitive 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 41.3] 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 adsorption model can handle both stripping and desorption effects. The user specifies an adsorption isotherm which tabulates the saturated rock adsorbed concentration versus the local polymer concentration in solution.

There are currently two adsorption models which can be selected. The first model ensures that each grid cell retraces the adsorption isotherm as the polymer concentration rises and falls in the cell. The second model assumes that the adsorbed polymer concentration on the rock may not decrease with time and hence does not allow for any desorption. More complex models of the desorption process could be implemented if required.

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 with the adsorbed polymer concentration. In order to compute the reduction in rock permeability, the user is required to specify the residual resistance factor (RRF) for each rock type. The actual resistance factor can then be calculated:

$$R_k = 1.0 + (\mathrm{RRF} - 1.0)\frac{C_a}{C_{a\,\max}} \qquad \text{[EQ 41.9]}$$

The value of the maximum adsorbed concentration, $C_{a\ \max}$, depends on the rock type and needs to be specified by the user. Alternative expressions for the resistance factor could also be implemented if required.

You specify the dead pore space for each rock type. It represents the amount of total pore space in each grid cell that is inaccessible to the polymer solution. The effect of the dead pore space within each cell is to cause the polymer solution to travel at a greater velocity than inactive tracers embedded in the water. The ECLIPSE model assumes that the dead pore space for each rock type does not exceed the corresponding irreducible water saturation.

Treatment of the shear thinning effect

The shear thinning of polymer has the effect of reducing the polymer viscosity at higher flow rates. ECLIPSE 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.

The flow velocity is calculated as:

$$v = B_w \cdot \frac{F_w}{\phi A} \qquad \text{[EQ 41.10]}$$

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 the polymer viscosity is assumed to be reversible, and is given by:

$$\mu = \mu_w[(P-1)M+1] \qquad \text{[EQ 41.11]}$$

where

μ_w	is the viscosity of water with no polymer present
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.

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:

$$R_r = e^{(\ln(R_w) + \ln(R_a))/2} \quad \text{[EQ 41.12]}$$

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. A further RUNSPEC section keyword BRINE can be used to specify that salt is an active component in the model (that is the polymer properties are salt-sensitive). By default the Polymer model is not salt-sensitive. Note that the IMPES solution algorithm cannot currently be used with the Polymer option.

The mixing parameter data is obligatory and should be defined using the keyword TLMIXPAR. The maximum number of mixing parameter regions is set using the parameter NTMISC in RUNSPEC 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.

For systems with BRINE specified, 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, then the keywords SALTNODE and PLYVISCS should be used to the enter the two-dimensional table for the viscosity of the solution.

The polymer adsorption data should be entered using the keyword PLYADS in the PROPS section. Other polymer-rock parameters such as the rock mass density used in the adsorption calculation, the dead pore space, the residual resistance factor are input using the keyword PLYROCK.

The shear thinning model is activated if the PLYSHEAR keyword is present in the PROPS section. 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 (38th switch) in the RPTGRID keyword. The representative radius used in the well inflow calculation is output with the well connection data. In models in which shear thinning is not an important effect the PLYSHEAR keyword should be omitted as the calculation of the velocity dependent viscosity can significantly increase the CPU requirements for the run.

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 solution polymer concentration and the initial adsorbed polymer concentration are both automatically set to zero. If the model contains any 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 solution polymer concentration is assumed to be always zero - any polymer entering the aquifer due to backflow is adsorbed by the aquifer rock.

The list of SUMMARY file keywords has been extended for the polymer/brine flood model for use by the GRAF program. The new keywords are included in the SUMMARY section documentation.

The definition of the polymer/salt well injection streams should be set using the keyword WPOLYMER in the SCHEDULE section. Note that wells are not allowed to crossflow when the Polymer Flood model is in use. A field / group / well / connection summary can be obtained at each report time in addition to polymer / salt balance reports on a field/region basis. Grid arrays reports can also be produced for polymer concentration, salt concentration, adsorbed polymer concentration and the permeability reduction factor for the aqueous phase at each report time.

Summary of keywords

RUNSPEC section

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

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

PLYADS	Polymer adsorption functions.
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.
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

MISCNUM	Miscibility region numbers.
RPTREGS	Controls output from the REGIONS section. Mnemonic MISCNUM outputs mixing region numbers.

SOLUTION section

RPTSOL	Controls output from the SOLUTION section. Mnemonic FIPPLY outputs polymer/salt fluid in place reports. Mnemonic PBLK outputs initial grid block polymer concentration values. Mnemonic PLYADS outputs the current adsorbed polymer concentration on the formation rock. Mnemonic RK outputs the water relative permeability reduction factor. Mnemonic SALT outputs the initial grid block salt concentration values. Mnemonic SALTVD outputs the tables of initial salt concentration versus depth.
SALT	Initial salt concentrations.
SALTVD	Salt concentration versus depth for equilibration.

SUMMARY section

The following items are relevant to the Polymer Flood model:

Table 41.1 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

SCHEDULE section

RPTSCHED	Controls output from SCHEDULE section. Mnemonic FIPSALT outputs polymer/salt fluid in place reports. Mnemonic PBLK outputs grid block polymer concentration values. Mnemonic PLYADS outputs the current adsorbed polymer concentration on the formation rock. Mnemonic RK outputs the water relative permeability reduction factor. Mnemonic SALT outputs the grid block salt concentration values.
WPOLYMER	Sets polymer/salt concentrations for injection wells.

Example problems

Problem 1

Polymer flood model test problem.
This is a 1-D problem modeling water injection into oil at adverse mobility ratio. 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 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. 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 timestep. In this case, we are writing field water-cuts, polymer production rates, cumulative polymer total and the cumulative oil production total at each timestep.

```
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  0.3 /
 /
TSTEP
1.0 99.0 2*100.0  /
```

Polymer slug in fresh water

Specify polymer concentration in the injection stream of well 'I'.

```
WPOLYMER
 'I'  5.0  0.0   /
 /
TSTEP
 5*100.0 /
```

Chase water period

Specify polymer concentration in the injection stream of well 'I'.

```
WPOLYMER
 'I'  0.0  0.3   /
 /
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 timestep. In this case, we are writing field water-cuts, polymer production rates, salt production rates and cumulative oil production totals at each timestep.

```
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 42

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,

$$\begin{aligned} \Phi_{\text{oil}} &= P_{\text{oil}} - \rho_o(D - \text{DATUM}) \\ \Phi_{\text{wat}} &= P_{\text{wat}} - \rho_w(D - \text{DATUM}) \\ \Phi_{\text{gas}} &= P_{\text{gas}} - \rho_g(D - \text{DATUM}) \end{aligned} \qquad \text{[EQ 42.1]}$$

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

$$\bar{\Phi}_p = \frac{\sum \Phi_p \cdot PV \cdot S_p}{\sum PV \cdot S_p} \qquad \text{[EQ 42.2]}$$

ECLIPSE 100

Setting `OPTIONS` keyword switch 31 nonzero 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

$$\bar{\rho}_p = \frac{\sum PV \cdot S_p \cdot \rho_p}{\sum PV \cdot S_p} \qquad \text{[EQ 42.3]}$$

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 42.1 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 538) 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 FOPI	GOPP GOPI	WOPP WOPI	COPP COPI	Oil Potential Production rate Oil Potential Injection rate
FWPP FWPI	GWPP GWPI	WWPP WWPI	CWPP CWPI	Water Potential Production rate Water Potential Injection rate
FGPP FGPI	GGPP GGPI	WGPP WGPI	CGPP CGPI	Gas Potential Production rate Gas Potential Injection rate

ECLIPSE 300

Well potentials can be written to the SUMMARY file, using the mnemonics

WOPP	Oil Potential Production rate
WWPP WWIP	Water Potential Production rate Water Potential Injection rate
WGPP WGIP	Gas Potential Production rate 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.

PVM

Chapter 42

PVM introduction

PVM (Parallel Virtual Machine) is a software package that permits a heterogeneous collection of computers connected by a network to be used as a single large computer. Thus large computational problems can be solved more cost effectively by using the aggregate power and memory of many computers. The software is very portable.

PVM enables you to exploit your existing computer hardware to solve much larger problems at minimal additional cost. Hundreds of sites around the world are using PVM to solve important scientific, industrial and medical problems, in addition to use of PVM as an educational tool to teach parallel programming. With thousands of users, PVM has become the de facto standard for distributed computing, world-wide.

See the Web address `www.epm.ornl.gov/pvm/` for full details on PVM.

GeoQuest applications

GeoQuest applications that support PVM allow direct communication between themselves and the simulator they are running, providing the ability to control the simulator and obtain “live” results as they are produced.

Installing PVM

PVM is controlled by several Environment variables, all or some of which may need to be set depending on the hardware to be used and the machines to be added to the virtual machine. The main ones are:

- `PVM_ARCH`

  This specifies the architecture type that the program is running on. If the program is being run using a GeoQuest macro then this is set for you. Otherwise you need to set it manually in the login scripts or some other form of script. The value to set it to can be obtained by running PVM and typing `conf`. Alternatively, use the supplied macro `@pvmgetarch`.

Note The architecture for a PC is always `WIN32`.

- `PVM_ROOT`

  This needs to be set to the location of the installation of PVM. For example, in a standard 2004A installation this is `/ecl/2004a/pvm3`.

If PVM is to be used to spawn processes across a network to other machines then `PVM_ROOT` and `PVM_ARCH` must be set in the `.cshrc` file on the machines hosting the spawned processes. Note that PVM only really works on machines where your login shell is the c-shell, as the k-shell does not execute a script when logging in using the rsh and therefore these parameters cannot be set except by `root` in the `/etc/profile`.

The PC installation requires two more variables to be set through Control Panel | System | Environment or Control Panel | System Properties: Advanced | Environment Variables on Windows PCs. These are:

- `PVM_TMP`

  This is the location for the daemon files created by PVM and is normally set to `C:\temp`. On UNIX systems these files are always found in the `/tmp` directory and currently there is no way of reassigning them.

Caution On Windows PCs, the temporary directory must be "visible" to the outside world. Set the file sharing permissions to (at least) read-only.

`PVM_RSH`

This is the RSH executable to be used when spawning processes on other boxes. On Windows PCs this is traditionally set to `C:\WINNT\SYSTEM32\RSH.EXE`.

On **heterogeneous** networks, that is ones that include UNIX and PC machines, a fifth variable `PVMXDR` must be set to `TRUE`. This indicates to PVM that message data must be converted between platforms as the different systems use different byte structures. This variable must be set on all systems participating in the virtual machine, preferably in the `.cshrc` file. On **homogeneous** networks, the variable should not be set as it slows down the message passing.

UNIX platforms

PVM is automatically installed when you install from the CD. It is installed in the `$ECLPATH/$ECLVER/pvm3` directory.

PC platforms

PVM can be provided in two forms on the PC. It is provided as part of the standard installation suite with the Tools, and also as a separate installable program which provides the source and manuals.

Starting PVM

When installation has been successfully completed (including installation of the standard macros supplied), PVM may easily be started by typing:

`@pvm` on a UNIX workstation

or

`$pvm` on a PC

at the command prompt.

A PVM console prompt will appear thus:

```
pvm>
```

Following the PVM console prompt you can enter commands such as:

`add <remote host name>`

adds another machine to the coupled system, or "virtual machine". You must add all the machines on which you wish to run the coupled applications. There is no need to "add" the local host machine, as this is automatically made available to PVM. If you wish to add a host with a particular path to the PVM daemon it should be added in quotes.

For example:

```
add 'bloggs dx=/ecl/2004a/pvm3/ARCH/pvmd3'
```

where `ARCH` is the target machine's architecture.

`conf` lists the machines on the coupled system. This is useful to check that you have all the necessary remote hosts.

`version` displays the version of PVM that is currently running.

`quit` quits the PVM console program, **leaving the PVM daemon running**. This is how you should exit the PVM console program when you require PVM to continue running so that you can run coupled programs.

`halt` kills the PVM daemon and exits the PVM console program. PVM is no longer running.

`ps` shows which processes are currently running under PVM. This is useful to check whether a job is still running and has all its slaves, or if a job has crashed to see if it has left any slaves running.

`reset` kills all processes running under PVM. This may be used if a process is left hanging. However, it is preferable to use `halt` to kill everything and then restart PVM.

`help` lists the commands you can give at the PVM console.

When PVM is started it creates two files in the `tmp` directory called `pvmd.uid` and `pvml.uid`. Here, `uid` is the users's ID obtained from the system. The `pvmd` file just indicates that PVM is running. If PVM is killed other that by halting it, this file may be left and **it must be deleted manually** otherwise PVM does not run the next time it is started. The `pvml` file contains the output from the PVM daemon and the processes it spawns when they write to `stdout` or `stderr`. It is a useful file to examine for debugging purposes.

Note In multi-user systems, each user may have their own PVM daemon running. These will operate independently of each other.

The paths that PVM uses to search for executables and the remote hosts to be added to the virtual machine are controlled by a file called `eclpvmhost.xxx` in your home directory. Here `xxx` stands for the release version number, for example 2003A, 2004A, etc. Information on the other options that can be set in the PVM hostfile may be obtained, on a full PVM installation under UNIX, by typing `man pvmd`. The important options are:

`ep`	the search path for PVM to find executables. This could consist of several paths separated by a colon. The first executable found will be used.
`dx`	the path to `pvmd3` on the remote machine. This is required for a PC-to-PC network. UNIX requires `PVM_ROOT` to be set at start-up.
`lo`	to log on as a different user on the remote machine.
`so`	password for a remote user (there may be security implications here).
`wd`	the directory to run all remote jobs on the remote machine.

This file is created the first time PVM runs. Every subsequent time PVM runs it checks if the host machine is in the file and adds it if it is not. The paths can be modified after the file has been created, if for some reason the search paths need to be changed from the standard.

On Windows PCs, the situation is somewhat different. PVM can still be run using a hostfile, but it is safest to use the complete path to the hostfile, for example:

```
pvm D:\ecl\home\hostfile.2004a
```

If you do not do this and a hostfile exists in the `PVM_ROOT` directory, PVM uses that one first.

Caution On the PC, if you are using a hostfile with the search path (`ep`) set, the executable to be used by PVM and the PVM installation must be on the same drive (as PVM does not appear to be able to change drives). If this situation causes problems, the only way to get PVM to run the executables is to copy the files (`eclipse.exe`, `e300.exe` `etc`.) to the `pvm3\bin\win32` directory and start PVM with no hostfile.

PVM architecture

It maybe useful to know the architecture under which PVM is running on a particular machine (that is the `PVM_ARCH` environment variable).

There are two methods to obtain this:

- Type `@pvmgetarch` at the command prompt on a UNIX machine. The architecture type is output to the screen.

or

- Start the PVM console and type `conf` at the `pvm>` prompt. This method would have to be used on the PC as there is currently no equivalent command to `@pvmgetarch`.

Note The architecture for a PC is `WIN32`.

Adding remote hosts to PVM

To run the simulator on a remote host, you must add the remote host to PVM.

You cannot add a remote host unless you have `remsh` (or `rsh` - remote shell) capability to that host. A simple test to see whether you have the capability is to type a command of the form `rsh <remote host name> -1 <userid> -n <command>`. For example,

```
rsh sun-machine -1 abc -n ls
```

attempts to run the `ls` command on host sun-machine with userid `abc`. If this is successful you should be able to add the host `sun-machine` to PVM. If this does not work contact your systems administrator to assist you. (This involves creating/updating a `.rhosts` file in your home directory on the target machine.)

If you are running under NT, ensure that you are using the correct `RSH.EXE` which is usually found in the `C:\WINNT\SYSTEM32` directory.

In addition to the above, the `PVM_ROOT` and `PVM_ARCH` environment variables **must** be set in your `.cshrc` file on the remote machine so that PVM knows where to start communications. The easiest way to check this is to log on normally to the remote machine (or use `rlogin`, which should work if `rsh` works) and check that the environment variables are set.

Note You cannot add a remote host to PVM if PVM is already running on the remote host under your userid. If it is, log on to the remote host and halt PVM. See "Starting PVM" on page 636.

You should now be in position to add the remote host to PVM as described below.

UNIX operating system

At the PVM console prompt `pvm>` enter:

```
add <remote host name>
```

If successful, the remote host is added and an appropriate message displayed.

NT operating system

At the PVM console prompt `pvm>` enter:

```
add '<remote host name> dx=/ecl/2004a/pvm3/lib/<arch>/pvmd3'
```

If successful, the remote host is added and an appropriate message displayed.

Note If communications are to be made between a PC and a UNIX workstation then both the PC and the UNIX workstation **must** have the environment variable `PVMXDR` set to `TRUE` for communications to work.

Troubleshooting

There are a number of common error messages that PVM can return, all of which have simple explanations. These are identitified by their error codes:

-6	No host. This could mean either the host you have requested is not in the virtual machine or `PVM_ARCH` is not set to the correct type.
-7	PVM cannot find the executable you have requested. Check the search paths in the `eclpvmhost` file.
-10	No data file. PVM cannot find the data file on a remote host.
-12	FLEX License problem. Usually returned if the FLEX license manager is not running or it cannot find the FLEX license file on a remote machine.

If a host cannot be added through the PVM console, check the following:

- Do you have `rsh` ability between the two hosts? Try using `rsh host ls` to see if you can communicate. Is it the correct `rsh`?
- Is `PVM_ROOT` set on the remote host in its `.cshrc` file?
- Is PVM are already running on the remote machine?
- If PVM is not running, ensure that the PVM temporary files `pvmd.uid` and `pvml.uid` do not still exist in the temporary directory on the remote machine.
- If commnuicating with a PC, ensure that the `PVM_TMP` directory share permissions are set to at least read-only.

If the PVM controller program and the version of ECLIPSE are mismatched, then the usual indication is that ECLIPSE hangs after printing out its build number and date. If this happens, check the PVM version numbers.

PC specific problems

If using PVM on a PC then there are a number of things you have to be aware of.

- The variable `LM_LICENSE_FILE` must be set in the window that PVM is started from. This could be done by setting it in the System Environment. Use Control Panel | System | Environment or Control Panel | System | System Properties Advanced | Environment Variables and set it in your environment space. You need to restart any command prompts after this has been done.
- The variable `HOME` must be set. Usually this would be set to `C:\` but if not then it will be need to be set in a similar fashion to `LM_LICENSE_FILE`. The diagnostic of this not being set is a line in the `pvml` file saying "`HOME` not set, using `/`". This may or may not be a fatal error but it is worth correcting. It can be checked by typing `set` in a command window and seeing if `HOME` is there.

- On the PC, after getting a link to another host and then halting PVM, it may be necessary to wait a few minutes before restarting PVM. This is because Windows does not free the TCP/IP socket immediately and thus a socket busy error may occur.

PVM versions used by GeoQuest releases

As different versions of PVM have been used by different GeoQuest software releases, and they are mutually incompatible, the macros check which versions are running to avoid conflicts. If an Open-ECLIPSE controller has been written, it too must use the appropriate version of PVM to avoid conflicts. Table 42.1 shows the PVM versions used with the various releases of the software.

Table 42.1 PVM versions used by GeoQuest releases

Release	PVM version	
	PC	UNIX
96A	3.3.6	3.3.6
97A	3.3.11	3.3.11
98A	3.4 (beta) 6	3.4 (beta) 6
99A	3.4.0	3.4.0
2000A	3.4.2	3.4.2
2001A	3.4.2	3.4.2
2002A	3.4.2	3.4.2
2003A	3.4.2	3.4.4
2004A	3.4.2	3.4.4

Limitations of PVM

PVM is not supported by load balancing software. If your company uses load balancing software to balance to the use of each machine, then PVM may not be appropriate. It will use the specified host directly, avoiding any load balancing issues.

Caution Do not halt PVM while any program is running, as there is a risk of crashing the program.

Recovery Mechanisms

Chapter 43

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:

$$\text{Total oil in place} = \text{PV} \cdot S_o \cdot \frac{1}{B_o} \qquad \text{[EQ 43.1]}$$

$$S_o = 1 - S_w - S_g$$

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:

$$
\begin{aligned}
\mathrm{DC} &= -d(\mathrm{PV}) \cdot S_o \frac{1}{B_o} \\
\mathrm{DW} &= \mathrm{PV} \cdot d(S_w) \frac{1}{B_o} \\
\mathrm{DG} &= \mathrm{PV} \cdot d(S_g) \frac{1}{B_o} \\
\mathrm{DE} &= \mathrm{PV} \cdot S_o \cdot d\left(\frac{1}{B_o}\right)
\end{aligned}
\qquad \text{[EQ 43.2]}
$$

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:

$$
\begin{aligned}
\mathrm{DG} &= \mathrm{DG} - \mathrm{DE} \\
\mathrm{DE} &= 0
\end{aligned}
\qquad \text{[EQ 43.3]}
$$

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:

$$\mathrm{DS} = \mathrm{PV} \cdot F_s \cdot d(S_g) \cdot \frac{1}{B_o}$$

$$\mathrm{DF} = \mathrm{PV} \cdot (1 - F_s) \cdot d(S_g) \cdot \frac{1}{B_o}$$

[EQ 43.4]

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:

$$\mathrm{DWT} = \mathrm{PV} \cdot F_t \cdot d(S_w) \cdot \frac{1}{B_o}$$

$$\mathrm{DWR} = \mathrm{PV} \cdot (1 - F_t) \cdot d(S_w) \cdot \frac{1}{B_o}$$

[EQ 43.5]

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 43.1 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:

$$d(\mathrm{OIP}) = \mathrm{PV}^{t+dt} \cdot S_o^{t+dt} \cdot \left[\frac{1}{B_o}\right]^{t+dt} - \mathrm{PV}^t \cdot S_o^t \cdot \left[\frac{1}{B_o}\right]^t \qquad \text{[EQ 43.6]}$$

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:

$$\begin{aligned}
\mathrm{PV}^{t+dt} &= \mathrm{PV}^t + d(\mathrm{PV}) \\
S_o^{t+dt} &= S_o^t + d(S_o) \\
\left[\frac{1}{B_o}\right]^{t+dt} &= \left[\frac{1}{B_o}\right]^t + d\left[\frac{1}{B_o}\right] \\
d(\mathrm{OIP}) &= [\mathrm{PV} + d(\mathrm{PV})][S_o + d(S_o)]\left[\frac{1}{B_o} + d\left(\frac{1}{B_o}\right)\right] - \left[\mathrm{PV} \cdot S_o \cdot \frac{1}{B_o}\right]
\end{aligned} \qquad \text{[EQ 43.7]}$$

Expanding the conservation equation,

$$\begin{aligned}
d(\mathrm{OIP}) = {} & d(\mathrm{PV}) \cdot S_o^t \cdot \left[\frac{1}{B_o}\right]^t + d(\mathrm{PV}) \cdot S_o^t \cdot d\left[\frac{1}{B_o}\right] \\
& + \mathrm{PV}^t \cdot d(S_o) \cdot \left[\frac{1}{B_o}\right]^t + d(\mathrm{PV}) \cdot d(S_o) \cdot \left[\frac{1}{B_o}\right]^t \\
& + \mathrm{PV}^t \cdot S_o^t \cdot d\left[\frac{1}{B_o}\right] + \mathrm{PV}^t \cdot d(S_o) \cdot d\left[\frac{1}{B_o}\right] \\
& + d(\mathrm{PV}) \cdot d(S_o) \cdot d\left[\frac{1}{B_o}\right]
\end{aligned} \qquad \text{[EQ 43.8]}$$

The terms in this equation that are associated with the various production mechanisms are:

$$\begin{aligned}
\text{Rock Compaction} &= \mathrm{DC} = -d(\mathrm{PV}) \cdot S_o^t \cdot \left[\frac{1}{B_o}\right]^{t+dt} \\
\text{Saturation Change} &= \mathrm{DS} = -\mathrm{PV}^{t+dt} \cdot d(S_o) \cdot \left[\frac{1}{B_o}\right]^t \\
\text{Oil Expansion} &= \mathrm{DE} = -\mathrm{PV}^t \cdot S_o^{t+dt} \cdot d\left[\frac{1}{B_o}\right] - d(\mathrm{PV}) \cdot d(S_o) \cdot d\left[\frac{1}{B_o}\right]
\end{aligned} \qquad \text{[EQ 43.9]}$$

Now, $d(S_o) = -d(S_w) - d(S_g)$, giving the final breakdown of the production:

$$\text{Rock Compaction} = \text{DC} = -d(\text{PV}) \cdot S_o^t \cdot \left[\frac{1}{B_o}\right]^{t+dt}$$

$$\text{Water Influx} = \text{DW} = \text{PV}^{t+dt} \cdot d(S_w) \cdot \left[\frac{1}{B_o}\right]^{t}$$

$$\text{Gas Influx} = \text{DG} = \text{PV}^{t+dt} \cdot d(S_g) \cdot \left[\frac{1}{B_o}\right]^{t}$$

$$\text{Oil Expansion} = \text{DE} = -\text{PV}^{t} \cdot S_o^{t+dt} \cdot d\left[\frac{1}{B_o}\right] - d(\text{PV}) \cdot d(S_o) \cdot d\left[\frac{1}{B_o}\right]$$

[EQ 43.10]

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:

$$\text{DG} = \text{DG} + \text{DE}$$

$$\text{DE} = 0.0$$

[EQ 43.11]

Reservoir Coupling

Chapter 44

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 PVM, 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 44.1 Three coupled reservoirs; one is the master

Figure 44.2 Three reservoirs coupled by a “dummy reservoir” master

Figure 44.1 and Figure 44.2 illustrate these two cases applied to three coupled reservoirs. In Figure 44.1, 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 44.2 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:
   - 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.
   - 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 538). 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:

- 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).
- 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 552). 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:

- 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.
- 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:

- The master sends the nodal pressures of the master groups to the respective slave groups in the slave reservoirs.
- The slave reservoirs balance their own networks, using the nodal pressures passed from the master as the "fixed" pressures of the slave groups.
- The slave processes send back to the master the flows of the slave groups at the current conditions.
- 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:

- 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.
- 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.
- 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 PVM 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. PVM 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. (See "SUMMARY section" on page 676.)

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 276). 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 662).

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 may use either the PVM or the MPI message-passing system to send messages and data between the master and the slave runs.

Reservoir coupling using PVM

In order to run a coupled reservoir simulation, PVM must be installed on your system. If the slaves are to be run on different workstations, PVM must be installed on each of them. Refer to "PVM" on page 633 for more information, and to the PVM manual, available from public-domain sources detailed in "PVM" on page 633, for installation instructions.

Before running the coupled reservoir simulation, PVM must be started on your workstation. This is usually done by typing "`pvm`", and gives the prompt "`pvm>`". Following the prompt, you can enter commands such as:

`add sun-sparc1`	Adds another machine (e.g. sun-sparc1) to the coupled system. You must add all machines on which you want to run a coupled simulation, using its hostname on your network.
`conf`	lists the machines on the coupled system. This is useful to check that you have added all the necessary machines.
`quit`	quits the `pvm>` prompt, but leaves PVM running so that you can perform coupled simulations.
`halt`	kills PVM.
`help`	lists the commands you can give under the `pvm>` prompt.

Add the machines on which you want to run the coupled simulation, then type "`quit`" to return to the UNIX prompt while leaving PVM running.

Reservoir coupling using MPI

To run a coupled reservoir simulation MPI must first be installed on your system. Refer to "Installing and configuring Parallel ECLIPSE" on page 56 of the "GSS Software 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 44.1:

```
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 44.1:

```
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 44.1, 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
limits
   '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 44.1 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. PVM 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 44.1 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
limits

   '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 251) 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

The FLEXlm license checking procedure in Reservoir Coupling is set up to enable all slaves running on the same host as the master to share the master's licenses for ECLIPSE Special Extensions. Thus for example, if the master and one or more of the slaves are using the Network option, and they are all running on the same host, then the coupled system will only use up a single Network option license. However, slaves that are run on other hosts will each require a separate license.

To take advantage of the license sharing facility, it is important to ensure that the licenses for the required Special Extensions are taken by the master before any of the slaves check them out. There are two areas where precautions should be taken in view of this requirement:

- Certain Special Extensions are not activated by a RUNSPEC section keyword. For example, the Standard Network option is activated by the GRUPNET keyword in the SCHEDULE section. When the master reads the GRUPNET keyword it will check out a Network option license. However, if the SLAVES keyword is entered in the master's SCHEDULE section before the GRUPNET keyword, the slaves will be started before the master has checked out a Network option license. The slaves will therefore not be aware that the master will be using a Network option license, and each slave will ask for a separate license for itself. A similar difficulty could be encountered with the Gas Field Operations model. Unless the GASFIELD keyword in the RUNSPEC section is used to turn on the special options, the master will not check out a Gas Field Operations model license until it reads the SWINGFAC keyword.
- Some of the Special Extensions may not actually be used by the master. For example, if the master is a 'dummy reservoir' containing the group hierarchy but no wells, there would seem little point in defining any local grid refinements in its dataset. Nevertheless, the master must take out a Local Grid Refinement license if this is to be shared with the slaves running on the same host. Otherwise each slave that contains an LGR will use a separate license for itself.

These pitfalls can be avoided by using the LICENSES keyword in the RUNSPEC section of the master's dataset. The keyword tells the master to check out a license for a specified set of options, irrespective of whether or not the master will actually make use of these options. Nevertheless, when the master has checked out a license for a certain option, it will be available to share with all the slaves running on the same host. Furthermore, since the LICENSES keyword is entered in the RUNSPEC section, the master will have checked out the required licenses before any of the slaves are started. For example, the keyword

```
LICENSES
  network /
  lgr /
/
```

will cause the master to check out a license for the Network model and Local Grid Refinement, which can be shared by any slaves running on the same host.

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 201. 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

RUNSPEC section

`LICENSES` Instructs the Reservoir Coupling Master to check out licenses for specified ECLIPSE Special options, which can then be shared by any slaves running on the same host.

SUMMARY section

The following keywords control output of data relevant to the Reservoir Coupling Facility:

Table 44.1 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

DEBUG	Switch 39 turns on debug from the Systems Annex for PVM 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
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 45

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 679.

For ECLIPSE 300, see "Restarts in ECLIPSE 300" on page 685.

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:

- 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.
- 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.

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.

- 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 201 for more details on file formats.

To create RESTART files from which a later run can restart, the procedure is:

- Set either the mnemonic 'BASIC' in the RPTRST keyword (switch 1) > 0, or set the mnemonic 'RESTART' in the RPTSCHED keyword (switch 7) > 0.
- 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).
- 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.
- Giving the mnemonic 'BASIC' in the RPTRST keyword a value > 2 results in the automatic generation of RESTART files at specific intervals of time.
- 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 TSTEP or DATES keyword present to generate the time zero RESTART file.
- 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.
- 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 254 for further information.

Reservoir Coupling Only

- The keywords SLAVES, DUMPCUPL and USECUPL in the Reservoir Coupling Facility. See "Reservoir Coupling" on page 653 for further information.
- Well PI Scaling tables (keyword PIMULTAB).
- Scale deposition and damage tables (keywords SCDPTAB and SCDATAB). See "Scale deposition model" on page 987 for further information.

Notes on RESTARTS in ECLIPSE 100

1 The following data cannot be changed at a restart:

- The grid dimensions NDIVIX, NDIVIY, NDIVIZ, and the number of active cells.
- The radial/Cartesian geometry switch.
- The phases present and the permitted mass transfers (OIL, WATER, GAS, dissolved GAS, vaporized OIL).
- The unit convention.

Vertical Equilibrium Model only

- The Vertical Equilibrium options, keyword VE in the RUNSPEC section.
- 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 at the start of the RESTART file. 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 timestep 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.

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.

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.

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:

- Delete all equilibration or enumeration keywords, and any analytic aquifer keywords.
- Insert the RESTART keyword to specify the RESTART file and report number.

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 (but keep any VFP tables, as these are not passed over in the restart file).
- 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:

- The grid dimensions N_x, N_y, N_z, and the number of active cells.
- The radial/cartesian switch.
- The number of components present.
- 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).

Rock Compaction

Chapter 46

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:

- A tabular description of the compaction as a function of pressure, which can be either reversible or irreversible.
- The ability to modify the transmissibility as a function of pressure.
- A hysteretic model to allow partial reflation.
- *ECLIPSE 300* A cyclic hysteretic model to allow compaction and dilation.
- *ECLIPSE 100* A water-induced compaction model.

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 46.1 Rock Compaction: Hysteresis option

ECLIPSE 300

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 46.2 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.

ECLIPSE 300

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 46.3 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 timestep.

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, i.e. 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.

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.

Saturation Functions

Chapter 47

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 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.

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)

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)

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.
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.
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 over-ride 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.)

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 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.

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.

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 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 **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 three 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. 17].

ECLIPSE 300

A further method called the IKU method is also available.

The default model assumed by ECLIPSE is shown in Figure 47.1. 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

$$k_{ro} = \frac{S_g k_{rog} + (S_w - S_{wco}) k_{row}}{S_g + S_w - S_{wco}} \qquad \text{[EQ 47.1]}$$

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 47.1 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. 40] and is available with keyword STONE1 in the PROPS section. The formula is

$$k_{ro} = k_{rocw} SS_o F_w F_g \qquad \text{[EQ 47.2]}$$

where

k_{rocw} is the value of the oil relative permeability in the presence of connate water only

$$SS_o = (S_o - S_{om})/(1 - S_{wco} - S_{om}) \text{ when } S_o > S_{om}$$

$$F_w = k_{row}/(k_{rocw} \cdot (1 - SS_w))$$

$$F_g = k_{rog}/(k_{rocw} \cdot (1 - SS_g))$$

where

$$SS_w = (S_w - S_{wco})/(1 - S_{wco} - S_{om}) \text{ when } S_w > S_{wco}$$

$$SS_g = S_g/(1 - S_{wco} - S_{om})$$

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_{row} 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 47.2:

Figure 47.2 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 47.3:

Figure 47.3 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 47.4 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:

$$S_{om} = S_{om}^{(i)} + F \cdot (S_{om}^{(d)} - S_{om}^{(i)}) \quad \text{[EQ 47.3]}$$

$$F = F_{sw}F_w + (1 - F_{sw})F_g \quad \text{[EQ 47.4]}$$

$$F_{sw} = \frac{S_w - S_{wco}}{1 - S_{wco} - S_{ocrw}^{(h)}} \quad \text{[EQ 47.5]}$$

$$F_w = \frac{S_{ocrw}^{(i)} - S_{ocrw}^{(h)}}{S_{ocrw}^{(i)} - S_{ocrw}^{(d)}}$$

$$F_g = \frac{S_{ocrg}^{(i)} - S_{ocrg}^{(h)}}{S_{ocrg}^{(i)} - S_{ocrg}^{(d)}}$$

[EQ 47.6]

$S_{ocrw}^{(i)}$	is the critical oil-to-water imbibition
$S_{ocrw}^{(d)}$	is the critical oil-to-water drainage
$S_{ocrw}^{(h)}$	is the critical oil-to-water actual
$S_{ocrg}^{(i)}$	is the critical oil-to-gas imbibition
$S_{ocrg}^{(d)}$	is the critical oil-to-gas drainage
$S_{ocrg}^{(h)}$	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:

$$S_{om} = S_{ogcr}X_g^A + S_{owcr}(1 - X_g)^A + BS_{gm}^2 X_g(1 - X_g)$$

[EQ 47.7]

where:

$$S_{gm} = 1 - S_{wco} - S_{ogcr}$$

[EQ 47.8]

$$X_g = \frac{S_g}{S_{gm}}$$

[EQ 47.9]

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. 41]. The formula is

$$k_{ro} = k_{rocw}\left[\left(\frac{k_{row}}{k_{rocw}} + k_{rw}\right)\left(\frac{k_{rog}}{k_{rocw}} + k_{rg}\right) - k_{rw} - k_{rg}\right]$$

[EQ 47.10]

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.

ECLIPSE 300

IKU method

A further relative permeability option is the IKU method, obtained with the IKU3P keyword in the PROPS section. This applies to all three phases symmetrically and data is entered for water and gas with respect to the other two phases using the SWF3 and SGF3 keywords. Considering the case of the oil relative permeability we first interpolate the endpoints with respect to gas and water to form lower and upper oil saturations.

This linear interpolation may be shown geometrically on a ternary diagram (Figure 47.5):

Figure 47.5 Ternary diagram showing the linear interpolation used to calculate S_{min} and S_{max}

A normalized oil saturation may then be defined in this range:

$$X_o = \frac{S_o - S_o^{\min}}{S_o^{\max} - S_o^{\min}} \qquad \text{[EQ 47.11]}$$

The K_{row} and K_{rog} are then looked up with respect to this variable, which is scaled into the corresponding two-phase mobile regions:

- For K_{row} define a scaling S'_{ow} such that:
  - $X_o = 0$ is mapped into S_{owcr}
  - $X_o = 1$ is mapped into $1 - S_{wcr}$

  Then look up $K_{row}(S'_{ow}(X_o))$
- For K_{rog} define a scaling S'_{og} such that:
  - $X_o = 0$ is mapped into S_{ogcr}
  - $X_o = 1$ is mapped into $1 - S_{gcr}$

Then look up $K_{rog}(S''_{og}(X_o))$

These two contributions are then combined to form the required relative permeability:

$$K_{ro} = f_w K_{row} + f_g K_{rog} \quad \text{[EQ 47.12]}$$

$$f_w = \frac{S_w}{S_w + S_g} \quad \text{[EQ 47.13]}$$

$$f_g = \frac{S_g}{S_w + S_g} \quad \text{[EQ 47.14]}$$

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 47.6.

Figure 47.6 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	$Sg = Sgco$ $Sw = Sw\ max$ $So = 1 - Sgco - Sw\ max$

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 $1 - S_{wco}$

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 $1 - S_{oco}$.

ECLIPSE 100

Mobile fluid requirements

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 $S_{ocrw} + S_{wcr} < 1$

2 $S_{ocrg} + S_{gcr} + S_{wco} < 1$

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 $1 - S_{ocrw}$ that is a null range if the first condition is not satisfied.

Figure 47.7 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.

As a first instance, 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 47.8 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}, given by the Li correlation as:

$$T^{\text{crit}} = \frac{\sum_j T_{cj} \cdot V_{cj} \cdot z_j}{\sum_j V_{cj} z_j} \qquad \text{[EQ 47.15]}$$

Then define f as:

$$f = \frac{T^{\text{crit}}}{T^{\text{res}}} \qquad \text{[EQ 47.16]}$$

f will clearly be 1 if the reservoir temperature 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

$$E = \frac{f - f_g}{f_o - f_g} \qquad \text{[EQ 47.17]}$$

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

$$K_{rhw} = EK^u_{row} + (1 - E)K^u_{rgw} \qquad \text{[EQ 47.18]}$$

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:

$$K_{ro} = gK_{rhw}(S_w) + (1 - g)K_{rog}(S_o) \qquad \text{[EQ 47.19]}$$

$$g = \frac{S_w - S_{wc}}{S_w - S_{wc} + S_g} \quad \text{[EQ 47.20]}$$

$$K_{rg} = hK_{rhw}(S_w) + (1-h)K_{rgo}(S_g) \quad \text{[EQ 47.21]}$$

$$h = \frac{S_w - S_{wc}}{S_w - S_{wc} + S_o} \quad \text{[EQ 47.22]}$$

This interpolation is not used or required when `ISGAS` is specified. It will be disabled if `NOMIX` is used.

Saturation Table Scaling

Chapter 48

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:

`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.
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 critical water saturation is $SWCR$, scaled maximum water saturation is SWU, which belongs to a saturation region whose unscaled critical water is S_{wcr} and whose unscaled maximum water saturation is S_{wmax}, the water capillary pressure P_{cow} is evaluated in the saturation table at the saturation, Sw, where

$$S_w' = S_{wco} + \frac{(SW - SWCR)(S_{wmax} - S_{wcr})}{SWU - SWCR}$$

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:

$$P_c = P_{ct} \frac{\mathrm{PCW}}{P_{cm}} \qquad \text{[EQ 48.1]}$$

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 SW', where

$$S_w' = S_{wcr} + \frac{(SW - SWCR)(S_{wmax} - S_{wcr})}{SWU - SWCR} \quad \text{[EQ 48.2]}$$

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, then if SR is the displacing critical saturation of the associated phase saturation (SR) of the associated phase.

SR = 1 – SOWCR-SGL	in water/oil or gas/oil/water runs
SR = 1 – 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:

$SWCR \le SW \le SR$

$$S_w' = S_{wcr} + \frac{(SW - SWCR)(S_r - S_{wcr})}{SR - SWCR} \quad \text{[EQ 48.3]}$$

$SR \le SW \le SWU$

$$S_w' = S_r + \frac{(SW - SR)(S_{wmax} - S_r)}{SWU - SR} \quad \text{[EQ 48.4]}$$

and also for $SW \le SWCR$ then $K_{rw}(SW) = 0$ and for $SW \ge SWU$ then $K_{rw}(SW) = K_{rwmax}(table)$. [EQ 48.5]

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:

$$K_{rw} = K_{rw}(\text{table})\left(\frac{\text{KRW(grid block)}}{K_{rw\ \max}(\text{table})}\right) \qquad \text{[EQ 48.6]}$$

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 $SWCR \le SW \le SR$

$$K_{rw}(S) = K_{rw}(S')\frac{\text{KRWR(grid block)}}{K_{rw}(S_r)(\text{table})}$$

[EQ 48.7]

2 $SR \le SW \le SWU$

$$K_{rw}(S) = \text{KRWR} + \frac{(K_{rw}(S')(table) - K_{rw}(S_r)(table)))}{(K_{rw\max}(table) - K_{rw}(S_r)(table))}(KRW - KRWR)$$

[EQ 48.8]

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 48.1:

Figure 48.1 Two-point scaling

and the three-point scaling is exemplified in Figure 48.2:

Figure 48.2 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 both 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.

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 48.3.

Figure 48.3 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

  $S'_{p,cr,i} = S'_p - M'_p$.

  [EQ 48.9]

- The remaining cells in the grid contain either mobile or immobile fluid throughout their depth. In this case

  $S'_{p,cr,i} = (\Sigma_i V_i S_{p,\,cr,i})/(\Sigma_i V_i)$.

  [EQ 48.10]

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 48.4). 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 48.4 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 $S^*_{w,\,\mathrm{crit}}$. 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.

$$J(S_w) = \frac{P_c}{\mathrm{ST}}\left(\frac{K}{\phi}\right)^{\frac{1}{2}} \qquad \text{[EQ 48.11]}$$

where

P_c	capillary pressure
ST	surface tension
K	rock permeability
ϕ	rock porosity.

If the JFUNC keyword is used in the GRID section then the capillary pressure is calculated as follows:

The capillary pressure is then calculated as follows:

$$P_c = J(S) \cdot \mathrm{ST} \cdot \left(\frac{\phi}{K}\right)^{\frac{1}{2}} \cdot U_{\mathrm{const}} \qquad \text{[EQ 48.12]}$$

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 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.

$$\text{Scaling Factor} = F = \text{ST} \cdot \left(\frac{\phi}{K}\right)^{\frac{1}{2}} \cdot U_{\text{const}} \qquad \text{[EQ 48.13]}$$

The JFUNC 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:

$$P_c = J(S) \cdot \text{ST} \cdot \frac{\text{ST}(P)}{\text{ST}_{\text{ref}}} \cdot \left(\frac{\phi}{K}\right)^{\frac{1}{2}} \cdot U_{\text{const}} \qquad \text{[EQ 48.14]}$$

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:

$$P_{cw}(S_{w\ \text{init}}) = P_{\text{oil}} - P_{\text{wat}} \qquad \text{[EQ 48.15]}$$

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.

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 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 711. 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 SOWCR + SWCR < 1.0

4 SOGCR + SGCR + 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 /
```

Solid phase

Chapter 49

Introduction

x	ECLIPSE 100 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

SOLID	Requests Solids option

GRID section keywords

ROCKDEN	Rock density

PROPS section keywords

	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

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 49.1 and Table 49.2.

Table 49.1 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 49.2 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 49.3 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:

$$\rho^c = \frac{\rho^c_{ref}}{(1 + C_{T1}(T - T_{ref}))(1 - C_p(P - P_{ref}))}$$

The reference density ρ^c_{ref} 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.

Thermal only

Trapped gas

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 863). This can be done using keywords CCTYPE. In this case the component molar solid volume is given by the gas law

$$V_{sol}^c = \frac{Z^c RT}{P} \qquad \text{[EQ 49.1]}$$

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.

Thermal only

Specific heats

The specific heat C^c_{sol} of a component c in the solid phase is given by

$$C^c_{sol} = C^c_{sol_1} + C^c_{sol_2} \cdot (T - T_{st})$$

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

$$C_s = f_s \cdot W_s \qquad \text{[EQ 49.2]}$$

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:

$$C_a = (1 - f_s) \cdot W_s / W_R \qquad \text{[EQ 49.3]}$$

where W_R is the weight of rock per unit pore volume:

$$W_R = \rho_R \cdot (1 - \theta)/\theta \qquad \text{[EQ 49.4]}$$

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:

$$C_a = C_a(C_s) \qquad \text{[EQ 49.5]}$$

which allows [EQ 49.2], [EQ 49.3] and [EQ 49.4] 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, $f_s = 1$.
- If the SOLIDADS keyword is used, but the rock density is zero, then it is assumed that all solids are in suspension, $f_s = 0$.

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

$$m^c = (1 - \hat{S}_s) \cdot m_{fl}^c + \hat{S}_s \cdot b_s \cdot x_s^c \qquad \text{[EQ 49.6]}$$

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 49.6] becomes:

$$m^{c} = m_{fl}^{c}$$

Initial porosity

The terms $(1-\hat{S}_s)$ and $\hat{S}_s$ in [EQ 49.6] 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.

Thermal only

Initial energy

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 74. 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 49.2.

Solution of the Linear Equations

Chapter 50

Introduction

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL

The linear equations are solved iteratively using Orthomin [Ref. 42] 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

$$r = b - Ay = (B - A)y \qquad \text{sum} = \sum_{i=1}^{\text{ncells}} r_i \qquad \text{[EQ 50.1]}$$

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. 44].

Nested factorization preconditioning

A typical matrix for a simple three-dimensional system is shown in Figure 50.1.

Figure 50.1 The structure of the Jacobian matrix for a simple 3*2*3 problem.

$$A = \begin{bmatrix}
D & U1 & . & U2 & . & . & U3 & . & . & . & . & . & . & . & . & . & . & . \\
L1 & D & U1 & . & U2 & . & . & U3 & . & . & . & . & . & . & . & . & . & . \\
. & L1 & D & . & . & U2 & . & . & U3 & . & . & . & . & . & . & . & . & . \\
L2 & . & . & D & U1 & . & . & . & . & U3 & . & . & . & . & . & . & . & . \\
. & L2 & . & L1 & D & U1 & . & . & . & . & U3 & . & . & . & . & . & . & . \\
. & . & L2 & . & L1 & D & . & . & . & . & . & U3 & . & . & . & . & . & . \\
L3 & . & . & . & . & . & D & U1 & . & U2 & . & . & U3 & . & . & . & . & . \\
. & L3 & . & . & . & . & L1 & D & U1 & . & U2 & . & . & U3 & . & . & . & . \\
. & . & L3 & . & . & . & . & L1 & D & U1 & . & U2 & . & . & U3 & . & . & . \\
. & . & . & L3 & . & . & L2 & . & L1 & D & U1 & . & . & . & . & U3 & . & . \\
. & . & . & . & L3 & . & . & L2 & . & L1 & D & U1 & . & . & . & . & U3 & . \\
. & . & . & . & . & L3 & . & . & L2 & . & L1 & D & . & . & . & . & . & U3 \\
. & . & . & . & . & . & L3 & . & . & . & . & . & D & U1 & . & U2 & . & . \\
. & . & . & . & . & . & . & L3 & . & . & . & . & L1 & D & U1 & . & U2 & . \\
. & . & . & . & . & . & . & . & L3 & . & . & . & . & L1 & D & . & . & U2 \\
. & . & . & . & . & . & . & . & . & L3 & . & . & L2 & . & . & D & U1 & . \\
. & . & . & . & . & . & . & . & . & . & L3 & . & . & L2 & . & L1 & D & U1 \\
. & . & . & . & . & . & . & . & . & . & . & L3 & . & . & L2 & . & L1 & D
\end{bmatrix} \qquad \text{[EQ 50.2]}$$

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 50.1) such as

$$A = D + \mathrm{L1} + \mathrm{U1} + \mathrm{L2} + \mathrm{U2} + \mathrm{L3} + \mathrm{U3} \qquad \text{[EQ 50.3]}$$

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

$$B = (P + \mathrm{L3})P^{-1}(P + \mathrm{U3}) = P + \mathrm{L3} + \mathrm{U3} + \mathrm{L3} \cdot P^{-1} \cdot \mathrm{U3}$$

$$P = (T + \mathrm{L2})T^{-1}(T + \mathrm{U2}) = T + \mathrm{L2} + \mathrm{U2} + \mathrm{L2} \cdot T^{-1} \cdot \mathrm{U2}$$

$$T = (G + \mathrm{L1})G^{-1}(G + \mathrm{U1}) = G + \mathrm{L1} + \mathrm{U1} + \mathrm{L1} \cdot G^{-1} \cdot \mathrm{U1} \qquad \text{[EQ 50.4]}$$

where G is a diagonal matrix and P^{-1} is the inverse of P etc. It follows that

$$\begin{aligned} B &= G + \mathrm{L1} + \mathrm{U1} + \mathrm{L2} + \mathrm{U2} + \mathrm{L3} + \mathrm{U3} + \mathrm{L1}G^{-1}\mathrm{U1} + \mathrm{L2}T^{-1}\mathrm{U2} + \mathrm{L3}P^{-1}\mathrm{U3} \\ &= A + G - D + \mathrm{L1}G^{-1}\mathrm{U1} + \mathrm{L2}T^{-1}\mathrm{U2} + \mathrm{L3}P^{-1}\mathrm{U3} \end{aligned} \qquad \text{[EQ 50.5]}$$

The diagonal matrix G is computed using

$$0 = G - D + \mathrm{L1}G^{-1}\mathrm{U1} + \mathrm{Colsum}(\mathrm{L2}T^{-1}\mathrm{U2} + \mathrm{L3}P^{-1}\mathrm{U3}) \qquad \text{[EQ 50.6]}$$

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. 44].

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

$$G = D - \mathrm{L1}G^{-1}\mathrm{U1} - \mathrm{colsum}(\mathrm{L2}T^{-1}\mathrm{U2}) - \mathrm{colsum}(\mathrm{L3}P^{-1}\mathrm{U3}) \qquad \text{[EQ 50.7]}$$

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

$$(P + \mathrm{L3})(I + P^{-1}\mathrm{U3})y = r \qquad \text{[EQ 50.8]}$$

$$\text{using } y = P^{-1}(r - \mathrm{L3}y) \qquad \text{[EQ 50.9]}$$

$$\text{and } y = y - P^{-1}\mathrm{U3}y \qquad \text{[EQ 50.10]}$$

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 50.38] 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 50.38] we must compute vectors of the form $z = P^{-1}q$ on each plane. This involves solving equations like

$$(T + \mathrm{L2})(I + T^{-1}\mathrm{U2})z = q \qquad \text{[EQ 50.11]}$$

$$\text{using } z = T^{-1}(q - \mathrm{L2}z) \qquad \text{[EQ 50.12]}$$

$$\text{and } z = z - T^{-1}\mathrm{U2}z \qquad \text{[EQ 50.13]}$$

[EQ 50.40] is solved in a forward sweep through the plane, one line at a time. Similarly equation is solved by sweeping backwards one line at a time.

Finally, we note that during the solution of [EQ 50.40] and we must compute vectors of the form $w = T^{-1}v$ on each plane. This involves solving the tri-diagonal equation

$$(G + \mathrm{L1})(I + G^{-1}\mathrm{U1})w = v \qquad \text{[EQ 50.14]}$$

$$\text{using } w = G^{-1}(v - \mathrm{L1}w) \qquad \text{[EQ 50.15]}$$

$$\text{and } w = w - G^{-1}\mathrm{U1}w \qquad \text{[EQ 50.16]}$$

and 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 50.17]

Compute the initial residual, $r = b - Ay$ [EQ 50.18]

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.

4 Test for convergence and escape if converged

Find a new search direction, $y = B^{-1}b$ [EQ 50.19]

5 Orthogonalize γ to the stack of previous search directions (z)

$$y = y - \frac{(y \cdot z)}{(z \cdot z)z}$$

[EQ 50.20]

6 Find the optimum step length, s, which minimizes the saturation normalized sum of the squares of the residuals $(r \cdot r)$:

$$s = \frac{r \cdot q}{q \cdot q}$$

[EQ 50.21]

$$q = A \cdot y$$

[EQ 50.22]

7 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 orthogonalization 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

$$\frac{||R||_2}{||R^o||_2} < TOL \qquad \text{[EQ 50.23]}$$

where

$$||R||_2 = \sqrt{\frac{\sum_{i=1}^{N} \sum_{c=1}^{Nci} \left(\frac{R_{i,c}}{PV_i}\right)^2}{N}} \qquad \text{[EQ 50.24]}$$

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

$$\begin{aligned} Ax + Cw &= r \\ Rx + Dw &= u \end{aligned} \qquad \text{[EQ 50.25]}$$

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 50.1),
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

$$(A - CD^{-1}R)x = r - CD^{-1}u \qquad \text{[EQ 50.26]}$$

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 50.25]. 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:

$$J_w = \frac{dQ_w}{dX} \qquad \text{[EQ 50.27]}$$

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 50.26].

See [Ref. 45], and [Ref. 47] for further information.

Nested factorization for LGRs

The linear equations to be solved take the form

$$Ax = b \qquad \text{[EQ 50.28]}$$

where A is a banded matrix. The banded nature can be expressed as

$$A = D + L1 + U1 + L2 + U2 + L3 + U3 + L4 + U4 \qquad \text{[EQ 50.29]}$$

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:

$$M = (B + \mathrm{L4})B^{-1}(B + \mathrm{U4}) = B + \mathrm{L4} + \mathrm{U4} + \mathrm{L4} \cdot B^{-1} \cdot \mathrm{U4} \qquad \text{[EQ 50.30]}$$

$$B = (P + \mathrm{L3})P^{-1}(P + \mathrm{U3}) = P + \mathrm{L3} + \mathrm{U3} + \mathrm{L3} \cdot P^{-1} \cdot \mathrm{U3} \qquad \text{[EQ 50.31]}$$

$$P = (T + \mathrm{L2})T^{-1}(T + \mathrm{U2}) = T + \mathrm{L2} + \mathrm{U2} + \mathrm{L2} \cdot T^{-1} \cdot \mathrm{U2} \qquad \text{[EQ 50.32]}$$

$$T = (G + \mathrm{L1})G^{-1}(G + \mathrm{U1}) = G + \mathrm{L1} + \mathrm{U1} + \mathrm{L1} \cdot G^{-1} \cdot \mathrm{U1} \qquad \text{[EQ 50.33]}$$

It follows that

$$\begin{aligned} M = {} & G + \mathrm{L1} + \mathrm{U1} + \mathrm{L2} + \mathrm{U2} + \mathrm{L3} + \mathrm{U3} + \mathrm{L4} + \mathrm{U4} \\ & + \mathrm{L1}G^{-1}\mathrm{U1} + \mathrm{L2}T^{-1}\mathrm{U2} + \mathrm{L3}P^{-1}U3 + \quad L4M^{-1}U4 \end{aligned} \qquad \text{[EQ 50.34]}$$

that results in

$$M = A + G - D + \mathrm{L1}G^{-1}\mathrm{U1} + \mathrm{L2}T^{-1}\mathrm{U2} + \mathrm{L3}P^{-1}\mathrm{U3} + L4M^{-1}U4 \qquad \text{[EQ 50.35]}$$

The diagonal matrix G is defined as

$$G = D - L1G^{-1}U1 - rowsum(L2T^{-1}U2 + L3P^{-1}U3 + L4P^{-1}U4) \qquad \text{[EQ 50.36]}$$

This choice ensures that

$$rowsum(A) = rowsum(M) \qquad \text{[EQ 50.37]}$$

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 .

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. It can be activated 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`

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 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 50.2 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 50.3 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 50.4 shows a different sub-grid configuration for the same coloring as in Figure 50.2. 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 50.2 Two color (red-blue) split suitable for a 4-processors parallel run

Figure 50.3 The sub-grids from with amalgamation of LGRs and base grid

Figure 50.4 The sub-grids from 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

$$A = \begin{bmatrix} 0 & 0 \\ L_{\alpha\beta} & 0 \end{bmatrix} + \begin{bmatrix} S_\alpha & 0 \\ 0 & S_\beta \end{bmatrix} + \begin{bmatrix} 0 & U_{\alpha\beta} \\ 0 & 0 \end{bmatrix} \qquad \text{[EQ 50.38]}$$

The procedure is to perform nested factorization on this block system, i.e. the pre-conditioner is

$$\tilde{M} = \begin{bmatrix} Q_\alpha & 0 \\ L_{\alpha\beta} & Q_\beta \end{bmatrix} \begin{bmatrix} Q_\alpha & 0 \\ 0 & Q_\beta \end{bmatrix}^{-1} \begin{bmatrix} Q_\alpha & U_{\alpha\beta} \\ 0 & Q_\beta \end{bmatrix} \qquad \text{[EQ 50.39]}$$

thus

$$\tilde{M} = \begin{bmatrix} Q_\alpha & U_{\alpha\beta} \\ L_{\alpha\beta} & (L_{\alpha\beta} Q_\beta^{-1} + Q_\beta) \end{bmatrix} \qquad \text{[EQ 50.40]}$$

The matrix Q is chosen to preserve rowsum (of pressure only) so

$$Q_\alpha = G_\alpha \qquad \text{[EQ 50.41]}$$

and

$$Q_\beta = G_\beta - rowsum(L_{\alpha\beta} Q_\beta^{-1} U_{\alpha\beta}) \qquad \text{[EQ 50.42]}$$

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

$$A = \begin{bmatrix} \begin{bmatrix} S_\alpha & U_{\alpha\beta} \\ L_{\alpha\beta} & S_\beta \end{bmatrix} & U_{\{\alpha\beta,\gamma\delta\}} \\ L_{\{\alpha\beta,\gamma\delta\}} & \begin{bmatrix} S_\gamma & U_{\gamma\delta} \\ L_{\gamma\delta} & S_\delta \end{bmatrix} \end{bmatrix} \qquad \text{[EQ 50.43]}$$

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 50.44]

Iteration

For k = 1,2,... until convergence, do

Search direction $u_k = M^{-1}\upsilon_k$ [EQ 50.45]

Matrix multiply: $w = Au_k$ [EQ 50.46]

Orthogonalize: $h_{i,k} = (w \cdot \upsilon_i)$ [EQ 50.47]

$$w = w - \sum_{i=1}^{k} h_{i,k}\upsilon_i$$

Normalize: $h_{k+1,k} = \|w\|_2$ [EQ 50.48]

$$\upsilon_{k+1} = \frac{w}{h_{k+1,k}}$$

Solution: $x_k = \sum_{i=1}^{k} \alpha_i u_i$ [EQ 50.49]

where the α_i are chosen to minimize the residual $\|b - Ax_k\|_2$

For complete implementation details see [Ref. 49].

The Solvent Model

Chapter 51

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 produces 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. For schemes that maintain pressure above the bubble point, you can run ECLIPSE in 3-phase mode and save on computer costs.

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,

$$K_{rg} = K_{rg}(S_{gas} + S_{solvent}) \quad \text{[EQ 51.1]}$$

Then the relative permeability of either gas component is taken as a function of the local solvent fraction within the gas phase,

$$\text{Fraction of solvent} = F_{sol} = \frac{S_{solvent}}{S_{gas} + S_{solvent}} \quad \text{[EQ 51.2]}$$

$$\text{Fraction of reservoir gas} = F_{gas} = \frac{S_{gas}}{S_{gas} + S_{solvent}} \quad \text{[EQ 51.3]}$$

$$\begin{aligned} K_{r\,\mathrm{sol}} &= K_{rg} \cdot K_{rfs}(F_{\mathrm{sol}}) \\ K_{r\,\mathrm{gas}} &= K_{rg} \cdot K_{rfg}(F_{\mathrm{gas}}) \end{aligned} \quad \text{[EQ 51.4]}$$

Typically the functions $K_{rfs} = K_{r\,\mathrm{sol}}/K_{rg}$ and $K_{rfg} = K_{r\,\mathrm{gas}}/K_{rg}$ (Equation [EQ 51.4]) will be 'straight line' functions such that

$K_{rfs}(0.0) = 0.0$ $K_{rfg}(0.0) = 0.0$

$K_{rfs}(1.0) = 1.0$ $K_{rfg}(1.0) = 1.0$

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:

$$K_{r\ \mathrm{oil}} = \frac{S_{\mathrm{oil}}}{S_n} K_{rn}(S_n) \qquad \text{[EQ 51.5]}$$

$$K_{rsg} = \frac{S_{\mathrm{solvent}} + S_{\mathrm{gas}}}{S_n} K_{rn}(S_n) \qquad \text{[EQ 51.6]}$$

where

K_{rsg}	is the total relative permeability of the gas and solvent
S_n	$= S_{\mathrm{gas}} + S_{\mathrm{solvent}} + S_{\mathrm{oil}}$
$K_{rn}(S_n)$	is the relative permeability of hydrocarbon to water.

Then:

$$K_{r\ \mathrm{sol}} = K_{rsg} \cdot K_{rfs}(F_{\mathrm{sol}})$$

$$K_{r\ \mathrm{gas}} = K_{rsg} \cdot K_{rfg}(F_{\mathrm{gas}}) \qquad \text{[EQ 51.7]}$$

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:

$$K_{r\ \mathrm{oil}} = M_{Kro} \frac{S_{\mathrm{oil}}}{S_n} K_{rn}(S_n) \qquad \text{[EQ 51.8]}$$

$$K_{rsg} = M_{Krsg} \frac{S_{\mathrm{solvent}} + S_{\mathrm{gas}}}{S_n} K_{rn}(S_n) \qquad \text{[EQ 51.9]}$$

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

$$S_{or} = S_{orm} M + S_{ori}(1 - M)$$

[EQ 51.10]

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

$$S_{sgr} = S_{sgrm} M + S_{sgri}(1 - M)$$

[EQ 51.11]

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:

$$K_r = K_{rm} M + K_{ri}(1 - M)$$

[EQ 51.12]

where

K_{rm}	is 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:

$$K_{r\,\mathrm{sol}} = \frac{S_{\mathrm{solvent}}}{S_n} K_{rn}(S_n)$$

[EQ 51.13]

$$K_{r\,\mathrm{oil}} = \frac{S_{\mathrm{oil}}}{S_n} K_{rn}(S_n) \qquad \text{[EQ 51.14]}$$

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

$$S_o{}^* = \mathrm{MAX}(S_o - S_{or}, 0.0) \qquad \text{[EQ 51.15]}$$

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.

$$S_g{}^* = \mathrm{MAX}(S_g - S_{gc}, 0.0) \qquad \text{[EQ 51.16]}$$

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`.

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.

$$\mu_{o\ \mathrm{eff}} = \mu_o^{1-\omega} \cdot \mu_{mos}^{\omega}$$
$$\mu_{s\ \mathrm{eff}} = \mu_s^{1-\omega} \cdot \mu_m^{\omega} \qquad \text{[EQ 51.17]}$$
$$\mu_{g\ \mathrm{eff}} = \mu_g^{1-\omega} \cdot \mu_{msg}^{\omega}$$

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:

$$\mu_{mos} = \frac{\mu_o \mu_s}{\left(\frac{S_o}{S_{os}}\mu_s^{1/4} + \frac{S_s}{S_{os}}\mu_o^{1/4}\right)^4} \qquad \text{[EQ 51.18]}$$

$$\mu_m = \frac{\mu_o \mu_s \mu_g}{\left(\frac{S_o}{S_n}\mu_s^{1/4}\mu_g^{1/4} + \frac{S_s}{S_n}\mu_o^{1/4}\mu_g^{1/4} + \frac{S_g}{S_n}\mu_o^{1/4}\mu_s^{1/4}\right)^4}$$

[EQ 51.19]

$$\mu_{msg} = \frac{\mu_s \mu_g}{\left(\frac{S_s}{S_{sg}}\mu_g^{1/4} + \frac{S_g}{S_{sg}}\mu_s^{1/4}\right)^4} \qquad \text{[EQ 51.20]}$$

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 51.17] to [EQ 51.20] above. The value of each effective component viscosity is then used in turn in equations [EQ 51.18] to [EQ 51.20] to yield an effective saturation fraction to be used in oil and solvent density calculations. The effective saturation fractions used for density calculations are:

$$\left(\frac{S_o}{S_n}\right)_{oe} = \frac{(\mu_{o\ \mathrm{eff}}^{1/4}\mu_o^{1/4}) - (\mu_s^{1/4}\mu_o^{1/4})}{\mu_{o\ \mathrm{eff}}^{1/4}\left(\mu_o^{1/4} - \mu_s^{1/4}\right)} \qquad \text{[EQ 51.21]}$$

$$\left(\frac{S_o}{S_n}\right)_{se} = \frac{(s_{of}\mu_o^{1/4}\mu_g^{1/4}) - \left(\dfrac{\mu_o^{1/4}\mu_s^{1/4}\mu_g^{1/4}}{\mu_{s\ \mathrm{eff}}^{1/4}}\right) + (s_{gf}\mu_o^{1/4}\mu_s^{1/4})}{\left(s_{of}\mu_o^{1/4}\mu_g^{1/4}\right) - \left(\mu_o^{1/4}\mu_g^{1/4}\right) + \left(s_{gf}\mu_o^{1/4}\mu_s^{1/4}\right)} \qquad \text{[EQ 51.22]}$$

$$\left(\frac{S_o}{S_n}\right)_{ge} = \frac{(-\mu_{g\ \mathrm{eff}}^{1/4}\mu_s^{1/4}) + (\mu_g^{1/4}\mu_s^{1/4})}{\mu_{g\ \mathrm{eff}}^{1/4}\left(\mu_s^{1/4} - \mu_g^{1/4}\right)} \qquad \text{[EQ 51.23]}$$

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 51.21] to [EQ 51.23], and the pure component densities (ρ_o, ρ_s, ρ_g) using the following formulae

$$\rho_{o\ \mathrm{eff}} = \rho_o\left(\frac{S_o}{S_n}\right)_{oe} + \rho_s\left[1 - \left(\frac{S_o}{S_n}\right)_{oe}\right] \qquad \text{[EQ 51.24]}$$

$$\rho_{s\ \mathrm{eff}} = \rho_s \left(\frac{S_s}{S_n}\right)_{se} + \rho_g S_{gf}\left[1 - \left(\frac{S_s}{S_n}\right)_{se}\right] + \rho_o S_{of}\left[1 - \left(\frac{S_s}{S_n}\right)_{se}\right]$$

[EQ 51.25]

$$\rho_{g\ \mathrm{eff}} = \rho_g \left(\frac{S_g}{S_n}\right)_{ge} + \rho_s\left[1 - \left(\frac{S_g}{S_n}\right)_{ge}\right]$$

[EQ 51.26]

3-component model

For the 3-component case the above viscosity model reduces to:

$$\mu_{o\ \mathrm{eff}} = \mu_o^{1-\omega}\mu_m^{\omega}$$

[EQ 51.27]

$$\mu_{s\ \mathrm{eff}} = \mu_s^{1-\omega}\mu_m^{\omega}$$

[EQ 51.28]

$$\mu_m = \frac{\mu_o \mu_s}{\left(\frac{S_s}{S_n}\mu_o^{1/4} + \frac{S_o}{S_n}\mu_s^{1/4}\right)^4}$$

[EQ 51.29]

$$S_n = S_o + S_s$$

[EQ 51.30]

The density model reduces to:

$$\left(\frac{S_o}{S_n}\right)_{oe} = \frac{(\mu_{o\ \mathrm{eff}}^{1/4}\mu_o^{1/4}) - (\mu_s^{1/4}\mu_o^{1/4})}{\mu_{o\ \mathrm{eff}}^{1/4}(\mu_o^{1/4} - \mu_s^{1/4})}$$

[EQ 51.31]

$$\left(\frac{S_o}{S_n}\right)_{se} = \frac{(\mu_{s\ \mathrm{eff}}^{1/4}\mu_o^{1/4}) - (\mu_s^{1/4}\mu_o^{1/4})}{\mu_{s\ \mathrm{eff}}^{1/4}(\mu_o^{1/4} - \mu_s^{1/4})}$$

[EQ 51.32]

$$\rho_{o\ \mathrm{eff}} = \rho_o\left(\frac{S_o}{S_n}\right)_{oe} + \rho_s\left[1 - \left(\frac{S_o}{S_n}\right)_{oe}\right]$$

[EQ 51.33]

$$\rho_{s\ \mathrm{eff}} = \rho_o\left(\frac{S_o}{S_n}\right)_{se} + \rho_s\left[1 - \left(\frac{S_o}{S_n}\right)_{se}\right]$$

[EQ 51.34]

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 51.33] and [EQ 51.34], 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):

$$\frac{1}{B_g} = \frac{1}{B_{gm}} M_p + \frac{1}{B_{gi}}(1 - M_p) \qquad \text{[EQ 51.35]}$$

$$\frac{1}{(B\mu)_g} = \frac{1}{(B\mu)_{gm}} M_p + \frac{1}{(B\mu)_{gi}}(1 - M_p) \qquad \text{[EQ 51.36]}$$

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 767.)

The capillary pressure is treated as follows:

$$P_{cog} = P_{cog}(\text{misc}) M_p + P_{cog}(\text{immisc})(1 - M_p) \qquad \text{[EQ 51.37]}$$

$$P_{cog}(\text{misc}) = P_{cog}(S_g)$$
$$P_{cog}(\text{immisc}) = P_{cog}(S_g + S_s) \qquad \text{[EQ 51.38]}$$

where

S_g	is the lean gas saturation
S_s	is the 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 equation [EQ 51.4]. 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 51.1.

Figure 51.1 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

$$F_{ij,c} = T_{ij} \cdot \frac{K_{ci}}{B_{ci}\,\mu_{ci}} \cdot \delta\Phi_{ij,c} \qquad \text{[EQ 51.39]}$$

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 51.39] by a linearly extrapolated relative permeability, K_{ci}^*, obtained from cells I and K,

$$K_{ci}^* = \frac{2\mathrm{DX}_i + \mathrm{DX}_k}{\mathrm{DX}_i + \mathrm{DX}_k} K_{ci} - \frac{\mathrm{DX}_i}{\mathrm{DX}_i + \mathrm{DX}_k} K_{ck} \qquad \text{[EQ 51.40]}$$

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,

$$\mathrm{MIN}(K_{ci}, K_{cj}) < K_{ci}^* < \mathrm{MAX}(K_{ci}, K_{cj}) \qquad \text{[EQ 51.41]}$$

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.

Using the solvent model

Initiating the model

Two keywords in RUNSPEC control the Solvent and Miscible models:

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:

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:

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:

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 781 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.

Summary of keywords

RUNSPEC section

SOLVENT	Activates the separate Solvent component.
MISCIBLE	Initiates the mixing calculation.

PROPS section

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

MISCNUM	Miscibility region numbers.
RPTREGS	Controls output from the REGIONS section. Mnemonic MISCNUM outputs miscibility region numbers.

SOLUTION section

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 51.1 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
					BKRN	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

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 52

Evaluation of surface tensions

x	ECLIPSE 100 ECLIPSE 300 SPECIAL

The treatment of surface tension effects follows that of [Ref. 50].

Surface tensions are obtained using the Macleod-Sugden correlation, given by

$$\sigma = \left[\sum_{i=1}^{N_c} [P]_i (b_L^m x_i - b_V^m y_i) \right]^4 \qquad \text{[EQ 52.1]}$$

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:

$$F = \left(\frac{\sigma}{\sigma_0}\right)^N \qquad \text{[EQ 52.2]}$$

where σ_0 is a reference surface tension. 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:

$$K_{ro} = FK_{ro}^{imm} + (1 - F)K_{ro}^{mis} \qquad \text{[EQ 52.3]}$$

The two contributions are scaled so that they both have the same critical saturation, given by $FS_{cr}^{imm} + (1 - F)S_{cr}^{mis}$.

Normally the critical miscible saturation S_{cr}^{mis} will be zero, but this may not be true if a miscible saturation table has been specified using the `MISCNUM` keyword.

The critical immiscible saturation S_{cr}^{imm} is generally obtained from a user-defined rock curve, but in the case of hysteresis it may be the critical point of a scanning curve (see "Hysteresis" on page 394 for further information).

To consider these contributions in more detail, we first take K_{ro}^{imm} in the absence of hysteresis. If the user-entered rock curve has a critical saturation of S_{cr}, it is scaled to $S_{cr}^{mod} = FS_{cr} + (1 - F)S_{cr}^{mis}$:

Figure 52.1 Two graphs to illustrate the scaling of a rock curve

In a similar way the K_{ro}^{mis} contribution is a straight line, scaled so that its end point is also $S_{cr}^{mod} = FS_{cr} + (1-F)S_{cr}^{mis}$:

Figure 52.2 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. Thus the critical saturation will vanish as the surface tension goes to zero.

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 576.

Capillary pressures

Capillary pressures must also go to zero as the surface tension vanishes, and here one would expect a linear dependence:

$$P_{cog}^{\text{corr}} = \left(\frac{\sigma}{\sigma_0}\right) P_{cog} \qquad \text{[EQ 52.53]}$$

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:

$$F_K = \left(\frac{\sigma}{\sigma_0}\right)^P \qquad \text{[EQ 52.54]}$$

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:

$$K_r = FK_r^{imm} + (1 - F)K_r^{mis} \qquad \text{[EQ 52.55]}$$

The two contributions are scaled so that they both have the same critical saturation, given by FS_{cr}^{imm}.

FVST avoids the use of both P and σ_0 in relative permeability calculations. σ_0 is still used in modifying capillary pressures:

$$P_{cog}^{corr} = \left(\frac{\sigma}{\sigma_0}\right) P_{cog} \qquad \text{[EQ 52.56]}$$

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. 51] 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 53

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:

$$N_c = \frac{|K \cdot \mathrm{grad}P|}{ST} C_{\mathrm{unit}} \qquad \text{[EQ 53.1]}$$

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.
$\lvert K \cdot \mathrm{grad}P \rvert$	is calculated as

$$|K \cdot \mathrm{grad}P| = \sqrt{(K_x \cdot \mathrm{grad}P_x)^2 + (K_y \cdot \mathrm{grad}P_y)^2 + (K_z \cdot \mathrm{grad}P_z)^2} \qquad \text{[EQ 53.2]}$$

where for cell i

$$K_x \cdot \mathrm{grad}P_x = 0.5\left[\left(\frac{K_x}{D_x}\right)_{i-1,i} \cdot (P_i - P_{i-1}) + \left(\frac{K_x}{D_x}\right)_{i,i+1} \cdot (P_{i+1} - P_i)\right] \qquad \text{[EQ 53.3]}$$

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 to values. This procedure is illustrated below for the oil to water curve:

Figure 53.1 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.

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:

$$P_{cow} = P_{cow}(S_w)\frac{ST(C_{surf})}{ST(C_{surf} = 0)} \qquad \text{[EQ 53.4]}$$

where

$ST(C_{surf})$ is the surface tension at the present surfactant concentration.

$ST(C_{surf} = 0)$ is the surface tension at zero concentration.

Water PVT properties

The surfactant modifies the viscosity of the pure water phase. The viscosity of the water (at the PVTW reference pressure) is input as a function of surfactant concentration. (See the SURFVISC keyword.)

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). The quantity of surfactant adsorbed on to the rock is given by:

$$\text{Mass of adsorbed surfactant} = \text{PORV} \cdot \frac{1-\phi}{\phi} \cdot \text{MD} \cdot \text{CA}(C_{\text{surf}}) \qquad \text{[EQ 53.5]}$$

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, a weighted average of *F* times the oil-wet value and (1-*F*) times the water-wet value is used. The fraction, *F*, is a function of the adsorbed surfactant concentration, and **must** be defined by tables input using the SURFADDW keyword in the PROPS section; the number of the table used being the same as that given in the SURFWNUM keyword.

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:

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 timestep is available, which sets a target surfactant change in a timestep. 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 907 for a more detailed discussion.

Summary of keywords

RUNSPEC section

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

RPTGRID	Controls output from the GRID section. Mnemonic KOVERD outputs the K/D values used in the calculation of the capillary number.

PROPS section

RPTPROPS	Controls output from the PROPS section. Mnemonics SURFVISC etc. 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.
SURFCAPD	Surfactant capillary de-saturation data.
SURFROCK	Surfactant-rock properties and adsorption model indicator.
SURFST	Water-oil surface tension in the presence of surfactant.
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

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

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 53.1 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
					BTCNFSUR	Concentration
FTIPTSUR				RTIPTSUR	BTIPTSUR	In Solution
				RTFTTSUR		Flow Total
FTADSUR				RTADSUR	BTADSUR	Adsorption total
					BTCASUR	Log (Capillary Num.)

SCHEDULE section

RPTSCHED — Controls output from the SCHEDULE section.
Mnemonic FIPSURF outputs surfactant fluid in place reports.
Mnemonic SURFADS outputs the current adsorbed surfactant concentration on the formation rock.
Mnemonic SURFBLK outputs the surfactant concentrations and the capillary number.

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 54

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.

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 826. 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°C / 705.5°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 a reference pressure and Rs value; the reference values must be specified using the VISCREF keyword. The oil and water viscosities are calculated as follows:

$$\mu_o = \mu_o(T) \cdot \frac{\mu(P, R_s)}{\mu(P_{\text{ref}}, R_{s\ \text{ref}})} \qquad \text{[EQ 54.1]}$$

where

$\mu_o(T)$	is the viscosity from the OILVISCT keyword
$\mu(P, R_s)$	is the viscosity from the PVTO (or PVDO) keyword at the prevailing pressure and R_s
$\mu(P_{\text{ref}}, R_{s\ \text{ref}})$	is the viscosity from the PVTO (or PVDO) keyword at the reference conditions

$$\mu_w = \mu_w(T) \cdot \frac{\mu(P)}{\mu(P_{\text{ref}})} \qquad \text{[EQ 54.2]}$$

where

$\mu_w(T)$	is the viscosity from the WATVISCT keyword
$\mu(P)$	is the viscosity from the PVTW keyword at the prevailing pressure
$\mu(P_{ref})$	is the viscosity from the PVTW keyword at the reference conditions

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 the THCONR keyword in the grid section to specify the rock 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 thermal conductivity and thermal transmissibility can be printed out using the THCONR and 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 56.42] with default values of the acentric factors and critical temperatures given in Table 54.1.

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 54.1.

Table 54.1 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 water 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 water is taken to be at the initial reservoir temperature.

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 828.

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 54.2 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 54.3 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 828.

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 55

Introduction

x	ECLIPSE 100 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

$$\underline{\boldsymbol{K}} = \begin{bmatrix} k_{xx} & k_{xy} & k_{xz} \\ k_{yx} & k_{yy} & k_{yz} \\ k_{zx} & k_{zy} & k_{zz} \end{bmatrix} \qquad \text{[EQ 55.1]}$$

where by symmetry

$$k_{yx} = k_{xy}, k_{zy} = k_{yz}, k_{zx} = k_{xz}. \qquad \text{[EQ 55.2]}$$

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. 74] 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. 73].

Discretization

Figure 55.1 Stencil for MPFA flow in 2D between cells (i, j) and (i + 1, j)

Referring to "Formulation of the Equations" on page 232, the standard two-point flow between cells A = (i, j) and B = (i + 1, j) for the corner point grid in Figure 55.1 can be written for a component c embedded in a phase p (p = o, w, g) as

$$F^{c}_{pAB} = T_{AB} M^{c}_{pAB} dP_{pAB} \qquad \text{[EQ 55.3]}$$

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 18.26]

and

dP_{pAB}	is the potential difference of phase p between cells A and B, given by

$$dP_{pAB} = P_A - P_B + P_{cpA} - P_{cpB} - \rho_{pAB} G(D_A - D_B) \qquad \text{[EQ 55.4]}$$

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 55.1 as

$$F^{c}{}_{pAB} = M^{c}{}_{pAB}\left(T_{AB}\,dP_{pAB} + \sum_{S} T_{AS}\,dP_{AS}\right) \qquad \text{[EQ 55.5]}$$

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 55.5].

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 18.35], so that

$$dP_{pAS} = P_A - P_S + P_{cpA} - P_{cpS} - \rho_{pAB}G(D_A - D_S) \qquad \text{[EQ 55.6]}$$

The transmissibilities T_{AB} and T_{AS} are calculated following the procedure described in [Ref. 74] 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 567).

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 893).

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 $\underline{\boldsymbol{K}}$ is diagonal

In general, when the grid is either non-orthogonal or $\underline{\boldsymbol{K}}$ 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 55.2 K-orthogonality error indicator

Consider an interface PQ between two cells 1 and 2. The permeabilities in each cells are $\underline{\boldsymbol{K}}_1$ and $\underline{\boldsymbol{K}}_2$. 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 $\underline{K}$*-orthogonal* (see [Ref. 73] and [Ref. 74]) 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:

- The parallel option (keyword `PARALLEL`)
- Velocity dependent relative permeabilities (keyword `VELDEP`)
- Cell coarsening (keyword `COARSEN`)
- The Diffusion option (keyword `DIFFUSE`)
- The Thermal option (keyword `THERMAL`)
- Radial geometry (keyword `RADIAL`)

Thermal Option

Chapter 56

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 869)
- combustion (see "Combustion" on page 863).

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 74)

'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 863.

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 814.

The next section contains a list of keywords that can only 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 components cannot appear in the gas phase. A simpler set of property keywords are used.

BLACKOIL properties are specified using the Temperature option keywords, see the "Temperature option" on page 814.

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

BLACKOIL	Request use of Black Oil properties
DEADOIL	Request use of Dead Oil properties
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

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
SIGMATH	Matrix-fracture thermal conductivity multiplier
THCONMF	Matrix-Fracture thermal conductivity
THCONR	Thermal conductivity of rock
THCONSF	Thermal conductivity saturation-dependence

EDIT section keywords

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 56.1, Table 56.2 and Table 56.3. Some keywords can be used with DEADOIL, some with LIVEOIL, and some with both.

Table 56.1 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 56.2 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*		x		Component densities
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

Table 56.2 Hydrocarbon properties

Reservoir Keyword	Surface Keyword	Liveoil	Deadoil	Description
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
PCRIT		x		Component critical pressure (used with KVWI)
PREF	PREFS	x		Component reference pressures
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
VISCREF		x	x	Reference conditions for the viscosity/temperature tables
ZFACTOR	ZFACTORS	x		Z-factors

Table 56.3 Water properties

Keyword	Liveoil	Deadoil	Description
THANALB	x	x	Analytic water and steam densities
THANALH	x	x	Analytic water and steam enthalpies
THTABB	x	x	Tabular water and steam densities
WATDENT	x	x	Water density temperature dependence
WATVISCT	x	x	Water viscosity table as a function of T

REGIONS section keywords

THERMNUM Thermal region numbers

SOLUTION section keywords

RPTRST* Controls solution output for GRAF: see Table 56.9

RPTSOL* Controls PRINT file output: see Table 56.9

SUMMARY section keywords

A list of SUMMARY keywords specific to the Thermal option is given below.

Table 56.4 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
FHTT		WHTT	Heater energy total

Table 56.4 Field, Group and Well energy rates (Continued)

Field	Group	Well	Information
		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 56.5 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 56.6 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
SHFR	Segment Enthalpy Flow Rate
SSQU	Segment Steam Quality
STEM	Segment Temperature

Table 56.7 Inter-region flow quantities

Region	Information
REFR	Inter-region energy flow rates
REFT	Inter-region energy flow totals

Table 56.8 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
BLAMB	Saturation dependent conductivity factor

SCHEDULE section keywords

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 56.9
RPTSCHED*	Controls print file solution output: see Table 56.9
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
WELSEGS*	Defines the multi-segment well structure Defines thermal properties of the tubing/casing
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 56.9 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
LAMB	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

Table 56.9 RPTRST, RPTSOL and RPTSCHED output controls (Continued)

Input Keyword or Mnemonic	Output
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
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

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.

Formulation

The equations used to describe thermal processes are similar to those used in the compositional simulation, "Formulation of the Equations" on page 232, 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 timestep:

$$R_{fl} = \frac{d}{dt}(V_p m_{fl}) + F_{fl} + Q_{fl} = 0 \qquad \text{[EQ 56.1]}$$

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 timestep.

The non-linear residual, R_e, is derived from the energy conservation equation in each grid block at each timestep:

$$R_e = \frac{d}{dt}(V_b e) + F_e + C_e + Q_{HL} + Q_e = 0 \qquad \text{[EQ 56.2]}$$

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 timestep.

The non-linear residual, R_v, is derived from the conservation of volume in each grid block at each timestep:

$$R_v = V_p - V_f = 0 \qquad \text{[EQ 56.3]}$$

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

$$m_c = m_t \cdot L \cdot x^c{}_o + m_t \cdot V \cdot x^c{}_g + m_t \cdot W \cdot x^c{}_w, \qquad \text{[EQ 56.4]}$$

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

$$m_t = \sum_c m_c + m_w \qquad \text{[EQ 56.5]}$$

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:

$$x^w{}_o = 0,$$

$$x^c{}_w = 0, \qquad c = 1,\ldots,N$$

$$x^w{}_w = 1, \qquad \text{if } W > 0$$

It is also assumed the three phases are in thermodynamic equilibrium, determined by user defined K-values:

$$x^c{}_g = K^c(P, T) \cdot x^c{}_o \qquad \text{[EQ 56.6]}$$

$$x^w{}_g = K^w(P, T) \cdot x^w{}_w \qquad \text{[EQ 56.7]}$$

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:

$$x^c{}_o = \frac{m_c}{m_t} \cdot \frac{1}{L + K^c \cdot V} \qquad c = 1,\ldots,N \qquad L > 0 \qquad \text{[EQ 56.8]}$$

$$x^c{}_g = \frac{m_c}{m_t} \cdot \frac{K^c}{L + K^c \cdot V} \qquad c = 1,\ldots,N \qquad V > 0 \qquad \text{[EQ 56.9]}$$

$$x^w{}_g = K^w \qquad V > 0,\ W > 0 \qquad \text{[EQ 56.10]}$$

Total mole fractions

The total mole fractions, which are used in [EQ 56.8] and [EQ 56.9] are defined as

$$z_c = \frac{m_c}{m_T}, \qquad z_w = \frac{m_w}{m_T}$$

[EQ 56.5] can be written in terms of the mole fractions

$$\sum_c z_c + z_w = 1 \qquad \text{[EQ 56.11]}$$

The total mole fractions differ from the hydrocarbon mole fractions:

$$\zeta_c = \frac{m_c}{m_T - m_w} = \frac{z_c}{1 - z_w} \quad \text{[EQ 56.12]}$$

Substituting this into [EQ 56.11] gives

$$\sum_c \zeta_c = 1 \quad \text{[EQ 56.13]}$$

Thermal flash

The four flash variables can be found from the primary variables by solving the four flash equations in each cell:

$$\sum_c x^c{}_o = 1 \qquad \text{if} \qquad L > 0, \quad \text{[EQ 56.14]}$$

$$\sum_c x^c{}_g + x^w{}_g = 1 \qquad \text{if} \qquad V > 0, \quad \text{[EQ 56.15]}$$

$$L + V + W = 1 \qquad \text{and} \quad \text{[EQ 56.16]}$$

$$V_b \cdot e = V_p \cdot m_t \cdot (L \cdot e_o + V \cdot e_g + W \cdot e_w) + (V_b - V_p) \cdot e_r, \quad \text{[EQ 56.17]}$$

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):

$$V_o = \frac{m_t L}{b_o}, \qquad V_g = \frac{m_t V}{b_g}, \qquad V_w = \frac{m_t W}{b_w} \quad \text{[EQ 56.18]}$$

The volume balance residual [EQ 56.3] can then be written as:

$$R_v = V_p - V_p \cdot (V_o + V_g + V_w) = 0 \quad \text{[EQ 56.19]}$$

The phase saturations S_o, S_g and S_w are defined as

$$S_o = \frac{V_o}{V_o + V_g + V_w}, \qquad S_g = \frac{V_g}{V_o + V_g + V_w}, \qquad S_w = \frac{V_w}{V_o + V_g + V_w} \quad \text{[EQ 56.20]}$$

When equation [EQ 56.19] holds, the phase saturations are equal to the phase specific volumes. Therefore

$$S_o = \frac{m_t L}{b_o} \qquad S_g = \frac{m_t V}{b_g} \qquad S_w = \frac{m_t W}{b_w} \qquad \text{[EQ 56.21]}$$

The number of moles of each component [EQ 56.4] and the energy density [EQ 56.17] can now be written in terms of the saturations:

$$m_c = S_o \cdot b_o \cdot x^c{}_o + S_g \cdot b_g \cdot x^c{}_g + S_w \cdot b_w \cdot x^c{}_w \qquad \text{[EQ 56.22]}$$

$$V_b \cdot e = V_p \cdot (S_o \cdot b_o \cdot e_o + S_g \cdot b_g \cdot e_g + S_w \cdot b_w \cdot e_w) + (V_b - V_p) \cdot e_r \qquad \text{[EQ 56.23]}$$

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,

$$F_{ci} = \sum_n \Upsilon_{ni} \sum_p M^c_p \cdot dP_{pni} \qquad \text{[EQ 56.24]}$$

The net flow rate of energy from cell i into neighboring cells is obtained in a similar manner:

$$F_{ei} = \sum_n \Upsilon_{ni} \sum_p M^e_p \cdot dP_{pni} \qquad \text{[EQ 56.25]}$$

In these formulae

Υ_{ni}	is the transmissibility between cells n and i,
M^c_p	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

$$M^c_p = x^c_p \cdot k_{rp}(S_p) \cdot \frac{b_p}{\mu_p} \text{ and} \qquad \text{[EQ 56.26]}$$

$$M_p^e = H_p \cdot k_{rp}(S_p) \cdot \frac{b_p}{\mu_p}$$ [EQ 56.27]

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,

$$dP_{pni} = P_n - P_i + P_{cpn} - P_{cpi} - \rho_{pni} G(D_n - D_i)$$ [EQ 56.28]

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:

$$Q_{cwi} = -\Upsilon_{wi} \cdot \sum_p (P_{pi} - H_{iw} - P_{bh}) \cdot M_{pi}^c$$ [EQ 56.29]

Similarly the rate of flow of energy into a production well w from cell i is

$$Q_{ewi} = -\Upsilon_{wi} \cdot \sum_p (P_{pi} - H_{iw} - P_{bh}) \cdot M_{pi}^e$$ [EQ 56.30]

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 964 for further information.

Heat conduction

The heat conduction term for each cell i is given by summing conduction between all neighboring cells n:

$$C_{ei} = -\sum_{n} \Psi_{ni} \cdot \Lambda \cdot (T_i - T_n) \qquad \text{[EQ 56.31]}$$

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 69. If solid components are modeled, then a number of modifications are required to the formulation described above.

The volume balance residual [EQ 56.3] 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 56.5], but the sum is taken over the fluid components only.

The energy density equation, [EQ 56.17], 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 56.19], 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 56.20] 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 56.20] 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 56.26] and [EQ 56.27].

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 56.1], energy conservation [EQ 56.2] and volume balance [EQ 56.3]). 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,

$$X_n = X_{n-1} + \Delta X_{n-1} \qquad \text{[EQ 56.32]}$$

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:

$$J = \begin{array}{l} \\ \text{Comp1} \\ \text{Comp2} \\ \dots \\ \text{Energy} \\ \text{Volume bal} \end{array} \begin{array}{c} \begin{array}{ccccc} P & m_1 & m_2 & \dots & e \end{array} \\ \begin{bmatrix} x & x & 0 & \dots & 0 \\ x & 0 & x & \dots & 0 \\ \dots & \dots & \dots & \dots & \dots \\ x & 0 & 0 & \dots & x \\ x & x & x & \dots & x \end{bmatrix} \end{array}$$

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

$$H_r = C_{r0}(T - T_{\text{ref}}) + \frac{C_{r1}(T - T_{\text{ref}})^2}{2} \qquad \text{[EQ 56.33]}$$

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.

Rock thermal conductivity and transmissibility

The rock thermal conductivity is given by the THCONR keyword.

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

$$1/H^{i \to j} = 1/H_i + 1/H_j \qquad \text{[EQ 56.34]}$$

$$H_i = THCONR_i \cdot \frac{A \cdot D_i}{D_i \cdot D_i} \qquad \text{[EQ 56.35]}$$

$$A \cdot D_i = A_X \cdot D_{iX} + A_Y \cdot D_{iY} + A_Z \cdot D_{iZ} \qquad \text{[EQ 56.36]}$$

and

$$D_i \cdot D_i = D_{iX}^2 + D_{iY}^2 + D_{iZ}^2. \qquad \text{[EQ 56.37]}$$

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 893 for further details.

Thermal transmissibilities can also be entered directly, using the keywords HEATTX, HEATTY, HEATTZ (or HEATTR, HEATTTHT, HEATTZ in radial coordinates).

Thermal transmissibilities can be multiplied by a saturation dependent factor Λ. Following Coats [Ref. 52], it is assumed that the thermal transmissibility is a function of the gas saturation, S_g, only. This implies that the thermal transmissibility of a rock filled with liquid is independent of whether the liquid is oil or water, and is the same whatever the porosity of the rock. The multiplier Λ is defined to be:

$$\Lambda = (1 - \alpha S_g) \qquad \text{[EQ 56.38]}$$

where α is specified by the THCONSF keyword.

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 687.

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.

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. 52] and [Ref. 56], where the K-value for each oil component is given by

$$K(P, T) = (A + B/P + C \cdot P) \cdot e^{-D/(T-E)} \qquad \text{[EQ 56.39]}$$

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:

$$K(P, T) = (B/P) \cdot e^{-D/T} \qquad \text{[EQ 56.40]}$$

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 56.10 and Table 56.11.

Table 56.10 K-value coefficients (Field units)

		B (Psi)	D (°R)
C1	Methane	168900	1904
C2	Ethane	170400	3062

Table 56.10 K-value coefficients (Field units) (Continued)

		B (Psi)	D (°R)
C3	Propane	528400	4571
C6	Hexane	686800	6598
C10	Decane	1214000	9966

Table 56.11 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. 66], where the K-value for each oil component is given by

$$K(P, T) = e^{5.372697 \cdot (1 + A_c) \cdot (1 - T_{crit}/T)} \cdot P_{crit}/P \qquad \text{[EQ 56.41]}$$

This is the same as [EQ 56.40] with coefficients

$$B = e^{5.372697 \cdot (1 + A_c)} \cdot P_{crit}$$

$$D = 5.372697 \cdot (1 + A_c) \cdot T_{crit} \qquad \text{[EQ 56.42]}$$

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 56.41] for surface conditions.

Density

Oil phase density

The molar density b_o of the oil phase is given by:

$$b_o = 1/V_{oil} \qquad \text{[EQ 56.43]}$$

where the molar specific volume V_{oil} of the oil phase is calculated using Amagat's law of partial volumes:

$$V_{oil} = \sum_{c=1}^{N_c} x^c V_{oil}{}^c \qquad \text{[EQ 56.44]}$$

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:

$$V_{oil}{}^c = \frac{MW^c}{\rho^c} \qquad \text{[EQ 56.45]}$$

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:

$$\rho^c = \frac{\rho_{ref}^c}{(1 + C_{T1}(T - T_{ref}))(1 - C_p(P - P_{ref}))} \qquad \text{[EQ 56.46]}$$

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 56.45] and [EQ 56.46]. 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:

$$b_g = 1/V_{gas} \qquad \text{[EQ 56.47]}$$

where the molar specific volume V_{gas} of the gas phase is

$$V_{\text{gas}} = \sum_{c=1}^{N_c} y^c V_{gas}{}^c + y^w V_{gas}{}^w \qquad \text{[EQ 56.48]}$$

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,

$$PV_{gas}{}^c = Z^c RT \qquad \text{[EQ 56.49]}$$

where Z^c is specified with the ZFACTOR or ZCRIT keywords. The molar specific water vapor volume $V_{gas}{}^w$ is calculated from the steam density.

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 863). This can be done using keywords CCTYPE. In this case [EQ 56.45] is replaced by

$$V_{oil}{}^c = \frac{Z^c RT}{P} \qquad \text{[EQ 56.50]}$$

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:

$$V_{sol} = \sum_{c=1}^{N_c} s^c V_{sol}{}^c \qquad \text{[EQ 56.51]}$$

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:

$$V_{sol}{}^c = \frac{MW^c}{\rho^c} \qquad \text{[EQ 56.52]}$$

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:

$$\rho^c = \frac{\rho_{ref}^c}{(1 + C_{T1}(T - T_{ref}))(1 - C_p(P - P_{ref}))} \quad \text{[EQ 56.53]}$$

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:

$$H_{oil} = \sum_{c=1}^{N_c} x^c \cdot MW^c \cdot H_{oil}^c \quad \text{[EQ 56.54]}$$

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:

$$H_{gas} = \sum_{c=1}^{N_c} y^c \cdot MW^c \cdot H_{gas}^c \quad \text{[EQ 56.55]}$$

where y^c is the mole fraction of component c in the gas phase.

The enthalpy of the hydrocarbon components in the solid phase is calculated using a mole fraction weighted average of the component enthalpies:

$$H_{sol} = \sum_{c=1}^{N_c} s^c \cdot MW^c \cdot H_{sol}^c \quad \text{[EQ 56.56]}$$

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

$$C^c_{oil} = C^c_{oil_1} + C^c_{oil_2} \cdot (T - T_{st})$$

$$C^c_{gas} = C^c_{gas_1} + C^c_{gas_2} \cdot (T - T_{st})$$

$$C^c_{sol} = C^c_{sol_1} + C^c_{sol_2} \cdot (T - T_{st})$$

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:

$$H^c_{oil} = \int_{T_{st}} C^c_{oil} dT = C^c_{oil_1} \cdot (T - T_{st}) + \frac{1}{2} \cdot C^c_{oil_2} \cdot (T - T_{st})^2 \qquad \text{[EQ 56.57]}$$

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

$$H^c_{gas} = H^c_{vaps} + \int_{T_{st}} C^c_{gas} dT \qquad \text{[EQ 56.58]}$$

$$= H^c_{vaps} + C^c_{gas_1} \cdot (T - T_{st}) + \frac{1}{2} \cdot C^c_{gas_2} \cdot (T - T_{st})^2$$

where H^c_{vaps} is the heat of vaporization at the standard temperature, defined by HEATVAPS.

If a solid phase is simulated, the molar enthalpy of component c in the solid phase is

$$H^c_{sol} = \int_{T_{st}} C^c_{sol} dT = C^c_{sol_1} \cdot (T - T_{st}) + \frac{1}{2} \cdot C^c_{sol_2} \cdot (T - T_{st})^2 \qquad \text{[EQ 56.59]}$$

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

$$H_{oil}{}^{c} = H_{gas}{}^{c} - \Delta H^c(T) \qquad \text{[EQ 56.60]}$$

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

$$H_{gas}{}^{c} = H_{oil}{}^{c} + \Delta H^c(T) \qquad \text{[EQ 56.61]}$$

The heat of vaporization is given by [Ref. 57]:

$$\Delta H^c(T) = A \cdot (1 - T/T_{crit}{}^{c})^{B} \qquad T < T_{crit}{}^{c} \qquad \text{[EQ 56.62]}$$

$$\Delta H^c(T) = 0 \qquad T \geq T_{crit}{}^{c} \qquad \text{[EQ 56.63]}$$

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 :

$$A = \frac{\Delta H^c(T_{nb})}{(1 - T_{nb}/T_{crit})^{B}} \qquad \text{[EQ 56.64]}$$

where $\Delta H^c(T_{nb})$ is the heat of vaporization at the normal boiling point. The exponent B, entered with keyword `HEATVAPE`, is usually defaulted to 0.38.

Viscosity

Oil viscosity temperature dependence

Oil component viscosities can be input in functional form (`OILVISCC` or `OILVISCF`) or tabular form (`OILVISCT`).

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:

$$\log_{10}\left(\mu_o^c + A\right) = B \cdot T^{-C} \qquad \text{[EQ 56.65]}$$

the Andrade formula ([Ref. 53] and[Ref. 54]):

$$\log_{10}\left(\mu_o^c\right) = A + B/T \qquad \text{[EQ 56.66]}$$

the Vogel formula ([Ref. 53] and [Ref. 54]):

$$\log_{10}\left(\mu_o^c\right) = A + B/(T + C) \qquad \text{[EQ 56.67]}$$

or a logarithmic formula:

$$\log_{10}\left(\mu_o^c\right) = A + B \cdot \log_{10}(T) \qquad \text{[EQ 56.68]}$$

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. 55]:

$$\mu_o^c = \left(\mu_{o1}^c + A\right)^{10^{-x}} - A \qquad \text{[EQ 56.69]}$$

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

$$\mu_o^c = D \cdot T^B \qquad \text{[EQ 56.70]}$$

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

$$\log(\mu_o) = \sum_{c=1}^{N_c} f_c(x^c) \cdot \log\left(\mu_o^c\right) \qquad \text{[EQ 56.71]}$$

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

$$\mu_g^c = A \cdot T^B \qquad \text{[EQ 56.72]}$$

where T is the temperature and c is the component number. The gas phase viscosity is calculated to be

$$\mu_g = \sum_{c=1}^{N_c+1} y^c \cdot \mu_g^c \qquad \text{[EQ 56.73]}$$

Example

For a 2 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

$$\rho_o = \frac{\rho_{ref,o}}{[1 - c_{Po}(P - P_{ref})][1 + c_{T1}(T - T_{ref}) + c_{T2}(T - T_{ref})^2]} \quad \text{[EQ 56.74]}$$

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 functional form follows Wingard [Ref. 58]:

$$\log_{10}(\mu_o) = A_o + B_o / T_K \quad \text{[EQ 56.75]}$$

where $A_o = -0.8772$, $B_o = 356.681$ and T_K is the temperature in ^{o}K. This is the same as the live oil viscosity given by the Andrade formula [EQ 56.66].

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.

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:

$$K_{ww} = \frac{P_b}{P} \qquad \text{[EQ 56.76]}$$

The saturation pressure and temperature can be found from the formula of Farouq Ali [Ref. 64]:

$$P_b = \left(\frac{T}{a}\right)^{1/b} \qquad \text{[EQ 56.77]}$$

$$T_b = aP^b \qquad \text{[EQ 56.78]}$$

Ejiougu et al. [Ref. 62] give the following values for the constants a and b when the temperature is measured in °C, and in MPa:

$a = 180.89$, $b = 0.2350$

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. 59] is used for the water density (default / THANALB):

$$\rho_w = \frac{A_0 + A_1 T_C + A_2 T_C^{\ 2} + A_3 T_C^{\ 3} + A_4 T_C^{\ 4} + A_5 T_C^{\ 5}}{1 + A_6 T_C} e^{c_{Pw}(P_P - A_7)} \qquad \text{[EQ 56.79]}$$

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 56.46]:

$$\rho_w = \frac{\rho_{ref,\,w}}{[1 - c_{Pw}(P - P_{ref})][1 + c_{T1}(T - T_{ref}) + c_{T2}(T - T_{ref})^2]} \qquad \text{[EQ 56.80]}$$

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. 53].

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:

$$\rho_g = \rho_{gb}\frac{T_b}{T} \qquad \text{[EQ 56.81]}$$

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. 60]:

$$\rho_{gb} = \exp\left[C_0 + C_1 T_{bK} + C_2 T_{bK}^{\ 2} + C_3 T_{bK}^{\ 3} + C_4 T_{bK}^{\ 4} + C_5 T_{bK}^{\ 5}\right] \qquad \text{[EQ 56.82]}$$

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. 53].

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. 53]. Tables from Haywood [Ref. 63] 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 [Ref. 61]

$$\mu_w = \frac{1}{A_w + B_w T_C + C_w T_C^{\ 2}} \qquad \text{[EQ 56.83]}$$

where the temperature T_C is in °C and

$$A_w = 0.1323 \qquad B_w = 0.03333 \qquad C_w = 7.643 \times 10^{-6}$$

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. 62]

$$\mu_g = A_g + B_g T_C + C_g P_P^{\ D_g} \qquad \text{[EQ 56.84]}$$

where the temperature T_C is in °C, the pressure P_P is in MPa and

$$A_g = 4.9402 \times 10^{-3},\ B_g = 5.0956 \times 10^{-5},\ C_g = 2.9223 \times 10^{-6},\ D_g = 2.5077$$

Combustion

The chemical reaction keywords, "Chemical Reactions" on page 69 can be used to model combustion. An example is given in "Example 1: Combustion" on page 80

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 69. 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. 76]. 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. Three methods are available:

1. They can be read from a save file or a RESTART file generated by an earlier run.
2. They can be set directly in each grid block.
3. They can be calculated by the equilibrium facility.

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 56.22] and the energy density [EQ 56.23] can be determined from the component phase mole fractions and the saturations [EQ 56.21]:

$$m_c = S_o \cdot b_o \cdot x^c{}_o + S_g \cdot b_g \cdot x^c{}_g + S_w \cdot b_w \cdot x^c{}_w \qquad \text{[EQ 56.85]}$$

$$e = \frac{V_p}{V_b} \cdot (S_o \cdot b_o \cdot e_o + S_g \cdot b_g \cdot e_g + S_w \cdot b_w \cdot e_w) + \left(1 - \frac{V_p}{V_b}\right) \cdot e_r \qquad \text{[EQ 56.86]}$$

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 the user to ensure the fluid composition is in thermodynamic equilibrium, see "Thermodynamic equilibrium condition" on page 837. Otherwise, the solution will change at the first time step when 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 56.7]. 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 56.28] is zero. For thermal simulations the heat conduction terms [EQ 56.31] 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 425.

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 56.12].

Aquifers

The temperature of the 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.

The heater rate is given by

$$H = min\left(H_{max}, V \cdot \frac{\Delta E}{\Delta t}\right) \qquad \text{[EQ 56.87]}$$

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 timestep.

In conditions where there is no heat flow from a cell, a heater governed by [EQ 56.87] 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 56.87] 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 will depend on the size of the heat flow and the size of the timestep.

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. 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 964 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. Steam quality Q_i and temperature T_i
2. Steam quality Q_i and pressure P_i
3. Pressure P_i and temperature T_i
4. 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

$$H_i = H_w(P_i, T_i) \text{ if } P_i > P_{sat}(T_i),$$

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 at the specified 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.

The steam trap can not be used in multi-segment wells

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.

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 56.32] 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 56.18]. 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}:

$$\lambda \cdot |\Delta P| \leq P_{con}$$

$$\lambda \cdot \left| \Delta X' \cdot \frac{\partial V}{\partial X'} \right| \leq S_{con}$$

$$\lambda \cdot \left| \Delta e \cdot \frac{\partial T}{\partial e} \right| \leq T_{con}$$

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 timestep 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}:

$$\sum_{X'} \left| X'_t - X'_{t-\Delta t} \right| \cdot \overline{\left| \frac{\partial V}{\partial X'} \right|} \leq S_{max} \qquad \text{[EQ 56.88]}$$

$$TPT \leq TPT_{max} \qquad \text{[EQ 56.89]}$$

$$TTE \leq TTE_{max} \qquad \text{[EQ 56.90]}$$

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 119. 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 119.

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}:

$$\left|SAT_{p,\,t} - SAT_{p,\,t-\Delta t}\right| \le SAT_{max} \qquad \text{[EQ 56.91]}$$

$$\left|T_t - T_{t-\Delta t}\right| \le T_{max} \qquad \text{[EQ 56.92]}$$

$$\left|e_t - e_{t-\Delta t}\right| \le e_{max} = c_p \cdot T_{max} \qquad \text{[EQ 56.93]}$$

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.

Table 56.12 Summary of convergence criteria and time stepping criteria

	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

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 814) 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 56.40] can be used

$$K \propto e^{-D/T}$$

with values of D taken from Table 56.10 or Table 56.11 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

$$B_o = \left(V_{oil,r}^{oil} + \frac{R_s}{F} \cdot V_{oil,r}^{gas} \right) / V_{oil,s} \qquad \text{[EQ 56.94]}$$

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 56.45] and [EQ 56.46]), which can be chosen to give a good fit for [EQ 56.94] 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:

$$B_o = \frac{\rho_{oil,s}}{\rho_{ref}^{oil}} + R_s \cdot \frac{\rho_{gas,s}}{\rho_{ref}^{gas}} \qquad \text{[EQ 56.95]}$$

Gas phase density

The gas formation volume factor B_g is given by

$$B_g = (F \cdot R_v \cdot V_{gas,r}^{oil} + V_{gas,r}^{gas}) / V_{gas,s} \qquad \text{[EQ 56.96]}$$

where $V_{gas,r}^{c}$ is the reservoir gas phase volumes per mole of component c:

$$V_{gas,r}^{c} = \frac{Z^c \cdot R \cdot T}{P}$$

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 56.96] 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:

$$V_{gas,\,s} = \frac{Z^{gas} \cdot R \cdot T}{P} = \frac{MW^{gas}}{\rho_{gas,\,s}}$$

Oil phase viscosity

The thermal oil phase viscosity is given by [EQ 56.71] as a function of temperature and mole fractions. It can be rewritten in terms of R_s to give:

$$\log \mu_{oil} = \frac{1}{F + R_s} \cdot (F \cdot \log \mu_{oil}{}^{oil} + R_s \cdot \log \mu_{oil}{}^{gas}) \qquad \text{[EQ 56.97]}$$

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 56.97]. 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 56.73] as a function of temperature and mole fractions. It can be rewritten in terms of R_v to give:

$$\mu_{gas} = \frac{1}{F \cdot R_v + 1} \cdot (F \cdot R_v \cdot \mu_{gas}{}^{oil} + \mu_{gas}{}^{gas}) \qquad \text{[EQ 56.98]}$$

Values of the component gas phase viscosities $\mu_{gas}{}^c$ can be found at the reservoir temperature to give a good fit for [EQ 56.98]. 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. However, default values can be used.

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

$$z_{oil} = \frac{F}{F + R_s}, \qquad z_{gas} = \frac{R_s}{F + R_s}$$

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 56.13 Thermal example files

.DATA File	Description	Auxiliary files
THERM1	Based on problem 1A of single well cyclic steam injection radial grid 1 component dead oil	
THERM2	Based on problem 2A of steam drive in 1/8th of a nine spot pattern 1 component dead oil	THERM2.GRF
THERM3	Based on problem 3A of steam drive in 1/8th of a nine spot pattern 3 component live oil	THERM3.GRF
THERM4	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. 82] 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. 82] single well SAGD 2 component live oil MSW	
THERM7	Based on case 3 of [Ref. 82] single well cyclic steam injection 3 component live oil MSW	
THERM8	Based on case 3 of [Ref. 82] single horizontal well, cyclic steam injection 2 component live oil heat loss from surface to reservoir MSW	
THERM9	Based on [Ref. 52] 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	

Table 56.13 Thermal example files

.DATA File	Description	Auxiliary files
THERM11	Based on [Ref. 52] and combustion with coke 5 component live oil solid phase chemical reactions	THERM11.GRF
THERM12	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) , with 2000 cells and 9 wells.	THERM13.GRF
THERM13F		
THERM13I		
THERM14D	Comparison of drift-flux and homogeneous flow models in a multi-segment well.	THERM14.GRF
THERM14H		
CASE3E100	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. References with particular relevance to thermal simulation are repeated below in alphabetical order of author.

Aziz, K., Ramesh, A. B. and Woo, P. T. *Fourth SPE Comparative Solution Project: Comparison of Steam Injection Simulators* [Ref. 1]
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. 2]
SPE 18752,SPE Reservoir Engineering, May 1991

Coats, K. H. *In-Situ Combustion Model* [Ref. 3]
SPE Journal, 533-554, December 1980

Crookston, H. B., Culham, W. E. and Chen, W. H. *Numerical Simulation Model for Thermal Recovery Processes* [Ref. 4]
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. 5]
Gulf Publishing Company, Second Edition, 1984

Ejiogu, G. C. and Fiori, M. *High-Pressure Saturated-Steam Correlations* [Ref. 6]
J. Pet. Tech. 39, 12, *Page 1585-1590,*1987

Farouq Ali, S. M. *Oil Recovery by Steam Injection* [Ref. 7]
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. 8]
The Journal of Canadian Petroleum Technology (Montreal), *Page 79-89, April-June 1981*

Haywood, R. W. *Thermodynamic Tables in SI (Metric) Units* [Ref. 9]
Cambridge University Press, Third Edition, 1990

Perry, R. H. and Green, D.W. *Perry's Chemical Engineers' Handbook* [Ref. 10]
McGraw Hill, Sixth Edition, 1984

Prats, M. *Thermal Recovery* [Ref. 11]
SPE Monograph 7, 1986

Reid, R.C., Prausnitz, J. M. and Polling, B. E. The Properties of Gases and Liquids
McGraw-Hill, 1987

Stone, T.W., Bennett, J., Law, D.H.-S., Holmes, J.A. *Thermal Simulation with Multisegment Wells* [Ref. 12]
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. 13]
SPEJ, 11, *Page 471-474,* 1989

Trangenstein, J. A. *Analysis of a Model and Sequential Numerical Method for Thermal Reservoir Simulation* [Ref. 14]
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. 15]
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. 16]
PhD thesis, Department of Petroleum Engineering, Stanford University,1988

Timing Reports

Chapter 57

x	ECLIPSE 100 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.

```
=======================================================
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 timestep chops then the number following NX (41) is the number of timesteps so far. Thus the average number of non-linear updates per timestep 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 timestep, 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.

Total Compressibility Checks

Chapter 58

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

$$V_g = \frac{V \cdot S_g}{B_g} + \frac{V \cdot R_s \cdot S_o}{B_o} \qquad \text{[EQ 58.1]}$$

$$V_o = \frac{V \cdot S_o}{B_o} + \frac{V \cdot R_v \cdot S_g}{B_g}. \qquad \text{[EQ 58.2]}$$

[EQ 58.1] and [EQ 58.2] 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

$$C_t = \frac{S_g}{B_g}\left(-\frac{dB_g}{dP} + \frac{dR_v}{dP} \cdot \frac{B_o - R_s B_g}{1 - R_s R_v}\right) + \frac{S_o}{B_o}\left(-\frac{dB_o}{dP} + \frac{dR_s}{dP} \cdot \frac{B_g - R_v B_o}{1 - R_s R_v}\right) \quad \text{[EQ 58.3]}$$

where the pressure derivatives are taken along the saturation curve.

This can also be written as

$$C_t = S_g C_{t,\,gas} + S_o C_{t,\,oil} \quad \text{[EQ 58.4]}$$

which splits the total compressibility into gas and oil dependent terms.

To derive this expression, we assume without loss of generality that

$$S_o + S_g = 1 \quad \text{[EQ 58.5]}$$

and note that

$$dV_o/dp = dV_g/dp = 0 \quad \text{[EQ 58.6]}$$

since the volumes of stock tank oil and separator gas are fixed.

Solving for S_o and S_g gives

$$S_o = \frac{R_v V_g - V_o}{V} \cdot \frac{B_o}{R_s R_v - 1} \quad \text{[EQ 58.7]}$$

and

$$S_g = \frac{R_s V_o - V_g}{V} \cdot \frac{B_g}{R_s R_v - 1} \quad \text{[EQ 58.8]}$$

And since $S_o + S_g = 1$ we have

$$V = \frac{1}{(R_s R_v - 1)} \cdot (V_o(R_s B_g - B_o) + V_g(R_v B_o - B_g)) \quad \text{[EQ 58.9]}$$

Hence

$$C_t = -\frac{1}{V} \cdot \frac{dV}{dp} \quad \text{[EQ 58.10]}$$

$$= -\frac{(R_s dR_v/dp + R_v dR_s/dp)}{(R_s R_v - 1)} + \frac{\left\{V_o\left[R_s \frac{dB_g}{dp} + B_g \frac{dR_s}{dp} - \frac{dB_o}{dp}\right] + V_g\left[R_v \frac{dB_o}{dp} + B_o \frac{dR_v}{dp} - \frac{dB_g}{dp}\right]\right\}}{V_o(R_s B_g - B_o) + V_g(R_v B_o - B_g)}$$

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 58.3].

- If $R_s = R_v = 0$ then

$$C_t = -\frac{S_g}{B_g} \cdot \frac{dB_g}{dP} - \frac{S_o}{B_o} \cdot \frac{dB_o}{dP} > 0 \qquad \text{[EQ 58.11]}$$

  since the formation volume factor pressure derivatives are < 0.

- If $R_s > 0$ and $R_v = 0$ then

$$C_t = -\frac{S_g}{B_g} \cdot \frac{dB_g}{dP} - \frac{S_o}{B_o} \cdot \left(\frac{dB_o}{dP} - B_g \frac{dR_s}{dP}\right) \qquad \text{[EQ 58.12]}$$

  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 58.3].

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 58.3]. 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 58.3] 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 59

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 the FIoGrid or GRID pre-processors, 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

$$\mathrm{TRANX}_i = \frac{\mathrm{CDARCY} \cdot \mathrm{TMLTX}_i \cdot A \cdot \mathrm{DIPC}}{B} \qquad \text{[EQ 59.1]}$$

where

TRANX_i	Transmissibility between cell i and cell j, its neighbor in the positive X-direction	
CDARCY	Darcy's constant	
	= 0.00852702	(`METRIC`)
	= 0.00112712	(`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:

$$A = \frac{\mathrm{DX}_j \cdot \mathrm{DY}_i \cdot \mathrm{DZ}_i \cdot \mathrm{RNTG}_i + \mathrm{DX}_i \cdot \mathrm{DY}_j \cdot \mathrm{DZ}_j \cdot \mathrm{RNTG}_j}{\mathrm{DX}_i + \mathrm{DX}_j} \qquad \text{[EQ 59.2]}$$

$$B = \frac{\left(\frac{\mathrm{DX}_i}{\mathrm{PERMX}_i} + \frac{\mathrm{DX}_j}{\mathrm{PERMX}_j}\right)}{2} \qquad \text{[EQ 59.3]}$$

$$\mathrm{DIPC} = \frac{\mathrm{DHS}}{\mathrm{DHS} + \mathrm{DVS}} \qquad \text{[EQ 59.4]}$$

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:

$$\mathrm{TRANZ}_i = \frac{\mathrm{CDARCY} \cdot \mathrm{TMLTZ}_i \cdot A}{B} \qquad \text{[EQ 59.5]}$$

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

$$A = \frac{\mathrm{DZ}_j \cdot \mathrm{DX}_i \cdot \mathrm{DY}_i + \mathrm{DZ}_i \cdot \mathrm{DX}_j \cdot \mathrm{DY}_j}{\mathrm{DZ}_i + \mathrm{DZ}_j} \qquad \text{[EQ 59.6]}$$

$$B = \frac{\left(\frac{\mathrm{DZ}_i}{\mathrm{PERMZ}_i} + \frac{\mathrm{DZ}_j}{\mathrm{PERMZ}_j}\right)}{2} . \qquad \text{[EQ 59.7]}$$

Note that there is no dip correction in the Z-transmissibility case.

ECLIPSE 100

OLDTRANR transmissibility calculations

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

$$\mathrm{TRANX}_i = \frac{\mathrm{CDARCY} \cdot \mathrm{TMLTX}_i \cdot \mathrm{DIPC}}{B} \qquad \text{[EQ 59.8]}$$

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:

$$B = \frac{\left(\frac{DX_i}{A_i \cdot PERMX_i} + \frac{DXj}{A_j \cdot PERMX_j}\right)}{2} \qquad \text{[EQ 59.9]}$$

where

$$A_i = DY_i \cdot DZ_i \cdot NTG_i$$

and

$$A_j = DY_j \cdot DZ_j \cdot NTG_j$$

with

$$DIPC = \frac{DHS}{DHS + DVS} \qquad \text{[EQ 59.10]}$$

$$DHS = \left(\frac{DX_i + DX_j}{2}\right)^2$$

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:

$$TRANZ_i = \frac{CDARCY \cdot TMLTZ_i}{B} \qquad \text{[EQ 59.11]}$$

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

$$B = \frac{\left(\frac{DZ_i}{A_i \cdot PERMZ_i} + \frac{DZ_j}{A_j \cdot PERMZ_j}\right)}{2} \qquad \text{[EQ 59.12]}$$

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:

$$TRANX_i = \frac{CDARCY \cdot TMLTX_i}{\frac{1}{T_i} + \frac{1}{T_j}} \qquad \text{[EQ 59.13]}$$

where

$$T_i = PERMX_i \cdot RNTG_i \cdot \frac{A \cdot D_i}{D_i \cdot D_i}$$

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 59.1 Y- and Z- transmissibility expressions

ECLIPSE 100

HALFTRAN transmissibility calculations

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

$$\bar{T}_{ir} = Kx_{ir} \cdot \mathrm{RNTG}_i \cdot \frac{A \cdot D_i}{D_i \cdot D_i} \qquad \text{[EQ 59.14]}$$

and

$$\bar{T}_{jl} = Kx_{jl} \cdot \mathrm{RNTG}_j \cdot \frac{A \cdot D_j}{D_j \cdot D_j}, \qquad \text{[EQ 59.15]}$$

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

$$\mathrm{TRANR}_i = \frac{\mathrm{CDARCY} \cdot \mathrm{TMLTR}_i \cdot \mathrm{DIPC}}{\dfrac{1}{T_i} + \dfrac{1}{T_j}} \qquad \text{[EQ 59.16]}$$

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:

$$\mathrm{TRANT}_i = \frac{CDARCY \cdot TMLTT_i \cdot DIPC}{\frac{1}{T_i} + \frac{1}{T_j}} \qquad \text{[EQ 59.17]}$$

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:

$$\mathrm{TRANZ}_i = \frac{\mathrm{CDARCY} \cdot \mathrm{TMLTZ}_i}{\frac{1}{T_i} + \frac{1}{T_j}} \qquad \text{[EQ 59.18]}$$

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 ECLIPSE100 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.

ECLIPSE 100

Use of linking permeabilities in the transmissibility calculation

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 966). 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

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:

$$\mathrm{TRANX}_i = \frac{\mathrm{CDARCY} \cdot \mathrm{TMLTX}_i \cdot A \cdot \mathrm{DIPC}}{B} \qquad \text{[EQ 59.19]}$$

with B rewritten as:

$$B = \frac{\mathrm{DX}_i + \mathrm{DX}_j}{2 \cdot \mathrm{PERMX}_{\mathrm{av}}}, \qquad \text{[EQ 59.20]}$$

and the average permeability is given by a harmonic average:

$$\mathrm{PERMX}_{\mathrm{av}} = \frac{(W_i + W_j)}{\left(\dfrac{W_i}{\mathrm{PERMX}_i} + \dfrac{W_j}{\mathrm{PERMX}_j}\right)}, \qquad \text{[EQ 59.21]}$$

with weights

$$W_i = \mathrm{DX}_i$$
$$W_j = \mathrm{DX}_j \qquad \text{[EQ 59.22]}$$

This is just a special case of the power law average given by:

$$\mathrm{PERMX}_{\mathrm{av}} = \left[\frac{W_i \cdot \mathrm{PERMX}_i^{\mathrm{POW}} + W_j \cdot \mathrm{PERMX}_j^{\mathrm{POW}}}{W_i + W_j}\right]^{\frac{1}{\mathrm{POW}}} \qquad \text{[EQ 59.23]}$$

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

$$\mathrm{PERMX}_{\mathrm{av}} = \left[\mathrm{PERMX}_i^{W_i} \cdot \mathrm{PERMX}_j^{W_j}\right]^{1/(W_i + W_j)}. \qquad \text{[EQ 59.24]}$$

Secondly, consider the OLDTRANR case. The formulation for OLDTRANR is analogous with OLDTRAN, but now the weights are given by

$$W_i = \frac{\mathrm{DX}_i}{A_i} = \frac{\mathrm{DX}_i}{\mathrm{DY}_i \cdot \mathrm{DZ}_i \cdot \mathrm{RNTG}_i}$$

$$W_j = \frac{\mathrm{DX}_j}{A_j} = \frac{\mathrm{DX}_j}{\mathrm{DY}_j \cdot \mathrm{DZ}_j \cdot \mathrm{RNTG}_j}$$

[EQ 59.25]

Finally, consider the NEWTRAN case. The weights corresponding to the default harmonic average of block permeabilities are given by:

$$\frac{1}{W_i} = \mathrm{RNTG}_i \cdot \frac{A \cdot D_i}{D_i \cdot D_i}$$

$$\frac{1}{W_j} = \mathrm{RNTG}_j \cdot \frac{A \cdot D_j}{D_j \cdot D_j}$$

[EQ 59.26]

and the alternative permeability averages are readily expressed in the same manner as above.

Tracer Tracking

Chapter 60

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:

ECLIPSE 100

Cascade algorithm

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 timestep sizes.

The cascade algorithm is always used for **active** tracers (API, Surfactant, Foam) 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.

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.

ECLIPSE 100

Flux limiting scheme

The 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. Again, this option only applies 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. The tracer concentrations are updated explicitly using a mini-timestep to prevent instability. As the tracer equation is linear, these explicit timesteps are very cheap, and hence the method can be applied to most problems (see [Ref. 67]).

The scheme can be used with LGRs but not if diffusion or dispersion are also present.

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.

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.

Tracking original fluids

ECLIPSE 300

Introduction

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 the 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.

The NORM option is provided so that you can request that the volumes be normalized to equal the total rate of the well.

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:

$$WOPR1 = WOPR \cdot \sum_{i=1}^{N} WXMF_i \cdot T_i \qquad \text{[EQ 60.1]}$$

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:

$$R_i = M_i^{T+\Delta T} \cdot C_i^{T+\Delta T} - M_i^T C_i^T + \Delta T \sum_n F^{i \to n} C_{in}^{T+\Delta T} + \Delta T C_i^{T+\Delta T} Q_{M_i} = 0$$

[EQ 60.2]

where:

M_i	is the moles of carrier in cell i
C_i	is the tracer concentration in cell i
$F^{i \rightarrow n}$	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. 68], 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 60.1 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

ECLIPSE 300

Detailed compositional simulation using tracers

This feature, which is available in ECLIPSE 300, is a straightforward implementation of paper SPE 49068 by Leibovici et al. [Ref. 78]. 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.

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 tracer concentration 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.

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`.

Treating passive tracers as non-linear

ECLIPSE 100

Convergence problems in 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 61

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 61.1.

Table 61.1 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	lbm/ft.^{3}	g/cc	kg/m^3
Pressure (absolute)	Barsa	Psia	Atma	Atma
Pressure (difference)	Bars	Psi	Atm	Atm
Temperature (absolute)	$^\circ K$	$^\circ R$	$^\circ K$	$^\circ K$
Temperature (difference)	$^\circ C$	$^\circ F$	$^\circ C$	$^\circ C$
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

Table 61.1 A table of the units used for four conventions (Continued)

Quantity	Metric	Field	Lab	PVT-M
Gas surface volume	sm^3	Mscf	scc	sm^3
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

Constants

Table 61.2 gives the values of some principal constants in the four unit conventions.

Table 61.2 Constants used in the four unit conventions

Quantity	Metric	Field	Lab	PVT-M
Gravity constant	0.0000980665	0.00694444	0.000967841	0.0000967841
Darcy constant	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 61.3 gives some useful conversion factors between the unit systems.

Table 61.3 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	
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	

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:

$$PV = nRT \qquad \text{[EQ 61.1]}$$

where:

P	is pressure
V	is volume
n	is number of moles
R	is the gas constant
T	is temperature

Using [EQ 61.1], 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:

$$V_{\text{molar}} = 379.39445\,\text{ft}^3$$

[EQ 61.2]

- the number of moles in unit volume is:

$$n_{\text{moles}} = 0.00263578\,\text{mol}$$

[EQ 61.3]

- When $P = 1.013\,\text{Bar}$ and $T = 288.15\,\text{K}$,
  - the volume occupied by one mole of gas is:

$$V_{\text{molar}} = 23.650203 \times 10^{-6}\,\text{m}^3$$

[EQ 61.4]

  - the number of moles in unit volume is:

$$n_{\text{moles}} = 0.04228293\,\text{mol}$$

[EQ 61.5]

Vertical Equilibrium

Chapter 62

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 62.1), 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 62.1 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 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 62.2 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 62.3 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 62.4 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 62.5. 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_{ocrg}, with $S_w=S_{wco}$ and $S_g=1-S_{ocrg}-S_{wco}$.
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_{ocrw}, with $S_w=1-S_{ocrw}-S_{gco}$.
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 62.5 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 62.6 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_{ocrw}.	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_{ocrg}.	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:

$$P_{cow} = g(\rho_w - \rho_o)(Z_{owc} - Z_{\text{centre}}) \qquad \text{[EQ 62.1]}$$

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 timestep and then held constant over the next timestep. 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.

$$K_{r\ \mathrm{tot}} = K_{r\ \mathrm{VE}} + K_{rr}(S_o) \qquad \text{[EQ 62.2]}$$

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 62.7 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 62.8 V.E Curves (SATNUM)

Figure 62.9 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 the GRID or GRAF programs. 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
SOCRW	Oil saturation at critical water (residual oil to water)
SOCRG	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 SOCRW, SGCO
KRW2	K_{rw} at SOWMX, SGCO
KRG1	K_{rg} at SOGMX, SWCO
KRG2	K_{rg} at SOCRG, SWCO

Cell face tables:

DEPTH	
AREA	Normalized surface area tables
KRO2	K_{ro} at SWCO, SGCO
KRW1	K_{rw} at SOCRW, SGCO
KRW2	K_{rw} at SOWMX, SGCO
KRG1	K_{rg} at SOGMX, SWCO
KRG2	K_{rg} at SOCRG, 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.

The Wellbore Friction Option

Chapter 63

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

$$\delta P_f = 2f \cdot \frac{L}{D} \cdot \rho \cdot v^2 \qquad \text{[EQ 63.1]}$$

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

$$\delta P_f = \frac{C_f \cdot f \cdot L \cdot \rho \cdot Q^2}{D^5} \qquad \text{[EQ 63.2]}$$

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 ($\mathrm{Re} < 2000$),

$$f = \frac{16}{\mathrm{Re}} \qquad \text{[EQ 63.3]}$$

For $\mathrm{Re} > 4000$ we use Haaland's formula ([Ref. 69]),

$$\sqrt{\frac{1}{f}} = -3.6 \log_{10}\left(\frac{6.9}{\mathrm{Re}} + \left(\frac{e}{3.7D}\right)^{10/9}\right) \qquad \text{[EQ 63.4]}$$

where

e	is the absolute roughness of the tubing in the same units as D

In the "uncertain region" ($2000 < \mathrm{Re} < 4000$) 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. 70], 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

$$\mathrm{Re} = \frac{\rho v D}{\mu} \qquad \text{[EQ 63.5]}$$

where

μ is the viscosity of the fluid.

Converting v to the volumetric flow rate Q and including the unit conversion factors, this becomes

$$\mathrm{Re} = \frac{C_r \rho Q}{D \mu} \qquad \text{[EQ 63.6]}$$

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 63.1, we would enter

```
FRICTION
 1  2   /
```

Figure 63.1 A trunk and branch multi-lateral well

For the "bird's foot" type well shown in Figure 63.2, we would enter

```
FRICTION
 1   3    /
```

Figure 63.2 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 63.3. 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 63.3 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 63.4.

Figure 63.4 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 63.5 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 63.5 Friction effects in a horizontal well

Production rate vs. length of perforations

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 954 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

FRICTION	Enables the Wellbore Friction option and sets array dimensions.

SUMMARY section

The following items are relevant to the Wellbore Friction model:

Table 63.1 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

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. 71] - 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 timestep and increase linear iteration limit to 40

```
TUNING
1 50 /
/
2* 40 /
```

Timesteps 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 64

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:

$$q_{p,j} = T_{wj} M_{p,j} (P_j - P_w - H_{wj}) \qquad \text{[EQ 64.1]}$$

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.

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:

$$\hat{q}_{p,j} = T_{wj}\left(\frac{k_{rp,j}}{\mu_{p,j}}\right)(P_j - P_w - H_{wj}) \qquad \text{[EQ 64.2]}$$

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

$$q_{p,j} = \hat{q}_{p,j}\, b_{p,j} \qquad \text{[EQ 64.3]}$$

where $b_{p,j} = \dfrac{1}{B_{p,j}}$, 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:

$$q_{i,j} = x_{i,j}\, b^m_{o,j}\, q_{o,j} + y_{i,j}\, b^m_{g,j}\, q_{g,j} \qquad \text{[EQ 64.4]}$$

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

$$T_{wj} = \frac{c\theta Kh}{\ln(r_o/r_w) + S} \qquad \text{[EQ 64.5]}$$

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 (= 2π), 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 block, 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.

$$r_o = 0.28\frac{\left[D_x^2\left(\frac{K_y}{K_x}\right)^{1/2} + D_y^2\left(\frac{K_x}{K_y}\right)^{1/2}\right]^{1/2}}{\left(\frac{K_y}{K_x}\right)^{1/4} + \left(\frac{K_x}{K_y}\right)^{1/4}} \qquad \text{[EQ 64.6]}$$

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 64.5] & [EQ 64.6] 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 64.6], and in [EQ 64.5] $Kh = D_x(K_yK_z)^{1/2}$.

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

$$T_{wj} = \frac{c\theta Kh}{\frac{r_2^2}{r_2^2 - r_w^2}\ln(r_2/r_w) - 0.5 + S} \qquad \text{[EQ 64.7]}$$

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

$$T_{wj} = \frac{c\theta Kh}{\frac{r_1^2}{r_2^2 - r_1^2}\ln(r_1/r_2) + 0.5 + S} \qquad \text{[EQ 64.8]}$$

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 $D_y = \theta(r_1 + r_2)/2$

ECLIPSE 300

Fractured completions

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

$$T_{wj} = \frac{8cK_yD_xD_z}{D_y + SD_x/\pi} \qquad \text{[EQ 64.9]}$$

In the case of FY, the completion connection factor is

$$T_{wj} = \frac{8cK_xD_yD_z}{D_x + SD_y/\pi} \qquad \text{[EQ 64.10]}$$

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,

$$S \rightarrow S + D\left|q_{fg}\right| \qquad \text{[EQ 64.11]}$$

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 64.11], 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

$$D_j = D_w \frac{\sum_i T_{wi}}{T_{wj}} \qquad \text{[EQ 64.12]}$$

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

$$D_m = \frac{G}{G + D|q_{fg}|}$$ [EQ 64.13]

where G is a geometric factor:

- For the Cartesian case:

$$G = \ln\left(\frac{r_o}{r_w}\right) + S$$

[EQ 64.14]

- For the inner cell radial case:

$$G = \frac{r_2^2}{r_2^2 - r_w^2}\ln\left(\frac{r_2}{r_w}\right) - 0.5 + S$$

[EQ 64.15]

- For the outer cell radial case:

$$G = \frac{r_1^2}{r_2^2 - r_1^2}\ln\left(\frac{r_1}{r_2}\right) + 0.5 + S$$

[EQ 64.16]

- For the fractured completion cases:
  - Fracture in x-direction:

$$G = \pi\frac{\Delta_y}{\Delta_x} + S$$

[EQ 64.17]

  - Fracture in y-direction:

$$G = \pi\frac{\Delta_x}{\Delta_y} + S$$

[EQ 64.18]

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

$$q_{fg} = T_w D_m M_{fg} \Delta P$$ [EQ 64.19]

where ΔP is the drawdown into the connection.

Substituting [EQ 64.13] for D_m gives the quadratic equation

$$T_w M_{fg} \Delta P = q_{fg} + \left(\frac{D}{G}\right) q_{fg} \left| q_{fg} \right| \qquad \text{[EQ 64.20]}$$

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 64.1] 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

$$M_{fp,j} = k_{p,j}\,\lambda_{p,j} \qquad \text{[EQ 64.21]}$$

where

$k_{p,j}$ is the relative permeability of the phase,

$\lambda_{p,j}$ is defined by

$$\lambda_{p,j} = \frac{1}{B_{p,j}\,\mu_{p,j}} \qquad \text{[EQ 64.22]}$$

$\mu_{p,j}$ is the phase viscosity, and

$B_{p,j}$ is the phase formation volume factor.

The quantities in [EQ 64.21] and [EQ 64.22] 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

$$\begin{aligned} M_{o,j} &= M_{fo,j} + R_{v,j}M_{fg,j} \\ M_{g,j} &= M_{fg,j} + R_{s,j}M_{fo,j} \end{aligned} \qquad \text{[EQ 64.23]}$$

If a flow-dependent skin factor is used, the free gas mobility is multiplied by the factor D_m defined in [EQ 64.13].

$$M_{fg,j} \rightarrow D_m M_{fg,j} \qquad \text{[EQ 64.24]}$$

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.

ECLIPSE 100

Russell Goodrich

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 64.1] if the term $\lambda_{g,j}$ defined in [EQ 64.22] 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.

ECLIPSE 100

Dry gas pseudo-pressure

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 64.1] 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

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. 72].

At the beginning of each timestep 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,

$$\beta_j = \frac{\int_{(P_w + H_{wj})}^{P_j} (M_g + M_o) dP}{(M_{g,j} + M_{o,j})\,(P_j - (P_w + H_{wj}))} \qquad \text{[EQ 64.25]}$$

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,

$$M_o / M_g = M_{o,j} / M_{g,j} \qquad \text{[EQ 64.26]}$$

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 64.25] 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 64.26] 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 64.1]. 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 timestep, and recalculated at the beginning of each subsequent timestep. 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 timestep,

$$\beta_{\text{used}} = f\,\beta_{\text{new}} + (1 - f)\,\beta_{\text{previous}} \qquad \text{[EQ 64.27]}$$

where the weighting factor f is specified in the `PICOND` keyword.

Generalized pseudo-pressure in ECLIPSE 300

The generalized pseudo-pressure method of Whitson and Fevang [Ref. 72] 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:

$$v_j = \frac{Q_j}{2\pi r h} = \frac{K K_{rj}}{\mu_j}\frac{dP}{dr} \qquad \text{[EQ 64.28]}$$

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:

$$Q_j = n_j V_{mj} \qquad \text{[EQ 64.29]}$$

where V_{mj} is phase molar volume. Combining [EQ 64.28] and [EQ 64.29] and re-arranging gives:

$$n_j \frac{dr}{r} = 2\pi K h \frac{K_{rj} b_j}{\mu_j} dp \qquad \text{[EQ 64.30]}$$

where $b_j = 1/V_{mj}$, the phase molar density. For component, i, the component phase molar rate will be:

$$n_{ij} \frac{dr}{r} = 2\pi K h x_{ij} \frac{K_{rj} b_j}{\mu_j} dp \qquad \text{[EQ 64.31]}$$

where x_{ij} is the mole fraction of the ith component in the jth phase.

Integrating [EQ 64.31] between the Peaceman pressure equivalent radius r_B and the well radius r_W gives:

$$n_{ij} \int_{r_W}^{r_B} \frac{dr}{r} = 2\pi K h \int_{p_W}^{p_B} x_{ij} \frac{K_{rj} b_j}{\mu_j} dp \qquad \text{[EQ 64.32]}$$

Summing component phase molar rates in the oil and gas phases for component i:

$$(n_{io} + n_{ig})\left[\ln\left(\frac{r_B}{r_W}\right) + S\right] = 2\pi Kh \int_{p_W}^{p_B} \left[x_i \frac{K_{ro} b_o}{\mu_o} + y_i \frac{K_{rg} b_g}{\mu_g}\right] dp \qquad \text{[EQ 64.33]}$$

or:

$$n_i = T \int_{p_W}^{p_B} M_i(p)dp \qquad \text{[EQ 64.34]}$$

where the well connection factor, T, including the skin factor, S, is given by:

$$T = \frac{2\pi Kh}{\ln\left(\frac{r_B}{r_W}\right) + S} \qquad \text{[EQ 64.35]}$$

and the Component Generalized Molar Mobility (CGMM) is given by:

$$M_i = x_i \frac{K_{ro} b_o}{\mu_o} + y_i \frac{K_{rg} b_g}{\mu_g} \qquad \text{[EQ 64.36]}$$

Normally, we do not consider the pressure dependency in [EQ 64.34]. Instead, we compute the CGMM at the block pressure:

$$\int_{p_W}^{p_B} M_i(p)dp \rightarrow M_i(p_B)[p_B - p_W] \qquad \text{[EQ 64.37]}$$

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 64.33].

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:

$$z_{pi} = \frac{M_i}{M_T} \qquad \text{[EQ 64.38]}$$

where:

$$M_T = \sum_i M_i \qquad \text{[EQ 64.39]}$$

Therefore, [EQ 64.34] can be written:

$$n_i = Tz_{pi} \int_{p_W}^{p_B} M_T(p)dp \qquad \text{[EQ 64.40]}$$

where:

$$M_T = \frac{K_{ro}b_o}{\mu_o} + \frac{K_{rg}b_g}{\mu_g} \qquad \text{[EQ 64.41]}$$

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:

$$n_i = TF_{Bi}M_i(p)[p_B - p_W] \qquad \text{[EQ 64.42]}$$

where:

$$F_{Bi} = \frac{1}{M_i(p_B)}\frac{1}{(p_B - p_W)}z_{pi}\int_{p_W}^{p_B} M_T(p)dp \qquad \text{[EQ 64.43]}$$

Because of [EQ 64.38], we can write:

$$M_i(p_B) = z_{pi}M_T(p_B) \qquad \text{[EQ 64.44]}$$

therefore, we define the total dimensionless flow blocking factor, F_B, (DFBF)

$$F_B = \frac{1}{M_T(p_B)}\frac{1}{(p_B - p_W)}\int_{p_W}^{p_B} M_T(p)dp \qquad \text{[EQ 64.45]}$$

and hence the modified inflow equation becomes:

$$n_i = TF_BM_i(p)[p_B - p_W] \qquad \text{[EQ 64.46]}$$

This final form of the GPP inflow equation, [EQ 64.46], differs only from the normal inflow equation by the additional multiplication factor, F_B, which we calculate from [EQ 64.45]. 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 64.45] appears to be a function of two variables:

$$Q(p_B, p_W) = \int_{p_W}^{p_B} M_T(p)dp \qquad \text{[EQ 64.47]}$$

but in fact it can be constructed as the difference of two integrals, namely:

$$Q(p_B, p_W) = R(p_B) - R(p_W) = \int_{p_L}^{p_B} M_T(p)dp - \int_{p_L}^{p_W} M_T(p)dp \qquad \text{[EQ 64.48]}$$

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,

$$\frac{M_o(S_o)}{M_g(S_g)} = \frac{L}{V} \qquad \text{[EQ 64.49]}$$

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:

$$M_j = \frac{K_{rj} b_j}{\mu_j} \qquad \text{[EQ 64.50]}$$

[EQ 64.49] is a simple non-linear equation in one variable, the oil saturation. The single integral in [EQ 64.48] is constructed by trapezium rule, following the suggestions made by Whitson and Fevang [Ref. 72]. In practical terms, the integral is constructed at the beginning of each timestep, only if the block conditions have changed significantly.

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 64.45] and the phase molar mobility, M_j [EQ 64.50] 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:

$$\frac{\ln r - \ln r_w}{\ln r_o - \ln r_w} = \frac{m(p) - m(p_w)}{m(p_B) - m(p_w)} \qquad \text{[EQ 64.51]}$$

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 576. The 'Pipe-Equivalent' velocity is calculated from:

$$v_g = \frac{n_g}{2\pi r h \phi (1 - S_{wc}) b_g} \qquad \text{[EQ 64.52]}$$

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 model" on page 579. This effect is modelled as a non-Darcy flow factor as described in "Non-Darcy Flow" on page 571.

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 64.21] and [EQ 64.22] with its sum over all phases, to give

$$M_{p,j} = \frac{\frac{k_{o,j}}{\mu_{o,j}} + \frac{k_{w,j}}{\mu_{w,j}} + \frac{k_{g,j}}{\mu_{g,j}}}{B_{p,j}} \qquad \text{[EQ 64.53]}$$

where p here refers to the injected phase. The connection injectivity is thus a function of the total fluid mobility in the grid block.

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, 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. The program 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

$$M_{p,j} = \alpha_p \left[\frac{\frac{k_{o,j}}{\mu_{o,j}} + \frac{k_{w,j}}{\mu_{w,j}} + \frac{k_{g,j}}{\mu_{g,j}}}{\alpha_{fo} B_{o,j} + \alpha_w B_{w,j} + \alpha_{fg} B_{g,j}} \right] \quad \text{[EQ 64.54]}$$

The expression reduces to the standard injection mobility, [EQ 64.53], 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 64.13]. 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 64.53] or [EQ 64.54] 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.

ECLIPSE 300

High mobility injection wells

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 64.1] 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,

$$\bar{\rho} = \rho_{o,\mathrm{surf}}\alpha_o + \rho_{w,\mathrm{surf}}\alpha_w + \rho_{g,\mathrm{surf}}\alpha_g \qquad \text{[EQ 64.55]}$$

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 64.55] 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,

$$\rho_{\mathrm{local}} = \frac{\rho_{o,\mathrm{surf}}Q_o + \rho_{w,\mathrm{surf}}Q_w + \rho_{g,\mathrm{surf}}Q_g}{Q_{fo}B_o + Q_wB_w + Q_{fg}B_g} \qquad \text{[EQ 64.56]}$$

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

$$J = \frac{Q_p}{P_d - P_w} \qquad \text{[EQ 64.57]}$$

where Q_p is the production rate of the chosen phase, and P_d is the pressure at the drainage radius. [EQ 64.57] 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:

$$J = \sum_j \left[T_{w_j} M_{p,j} \left(\frac{\ln(r_o/r_w) + S}{\ln(r_d/r_w) + S} \right) \right] \qquad \text{[EQ 64.58]}$$

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 64.58].

The simulator's calculation of the productivity index should not be used for a horizontal well, as the assumptions used in [EQ 64.58] will not be valid. [EQ 64.58] 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 64.57]. 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 64.58] simply reduces to

$$J = \sum_j [T_{wj} M_{p,j}] \qquad \text{[EQ 64.59]}$$

and the PI should be regarded as the PI of the well within its connecting grid blocks rather than within its drainage region.

ECLIPSE 100

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 64.57] 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 keyword WPAVE.

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 65

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 964.

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`.

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 973. 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 is activated by entering the keyword PSEUPRES.

ECLIPSE 100

Scale deposition model

A scale deposition model is available in ECLIPSE 100, 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.

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. Reservoir fluid volume (or voidage) rate
6. Bottom hole pressure
7. Tubing head pressure
8. *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)
9. *ECLIPSE 300 only* Total molar rate
10. *ECLIPSE 300 only* Calorific value 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.

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 992), 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.

ECLIPSE 100

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.

ECLIPSE 100

Automatic opening of wells and well connections

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 383.

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 65.1 A five-level hierarchy

An example of a five-level hierarchy is shown in Figure 65.1. 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
- 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)
- 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

- 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 631). 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 992 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 383.

Note The drilling queue should not be confused with the Automatic Opening facility described on page -992. 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 page page - 995, 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 995. 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.

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 995, 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 65.1 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 995. 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 997, 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.

ECLIPSE 300

Defining the composition of injected gas

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.

ECLIPSE 300

Gas accounting tables

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.)

4 Excess volume rate
The rate difference between the gas produced and the gas used to meet the fuel, sales and re-injection requirements.

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 1003.

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.

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.

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.)

9 Gas injection volume rate
The sum of 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.

ECLIPSE 300

Export tables

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. Note that 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).

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).

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 995. 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 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.

ECLIPSE 100

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 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. 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. The alternative MAXR option also places the group/field under gas production rate control with a similarly calculated production target. But it also sets the re-injection fraction limit to 1.0 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. 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 minus 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.

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.

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:

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	
LEAKOFF	Output of fracture fluid loss (fracture modeling only)	28	
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

Table B.1 RPTPROPS output controls (Continued)

Input Keyword or Mnemonic	Output	E100 Integer Control	E300
ROCK,	Output of rock properties	8	
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 /
```

ECLIPSE 100

Example 3

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

Table C.1 RPTGRID output controls (Continued)

Input Keyword Or Mnemonic	Output (Continued)	E100 Integer Control	E300 Availability
PERMX	Output of X direction permeabilities	4	x
PERMY	Output of Y direction permeabilities	5	x
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 /
```

ECLIPSE 100

Example 2

Changing the level of output

```
RPTGRID
 COORD=2 ZCORN=2 /
```

ECLIPSE 100

Example 3

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 886 for an explanation of the outputs. Internal clocks within TS are not recorded during intermediate timesteps 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	
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

Table E.1 RPTSCHED output controls (Continued)

Input Keyword or Mnemonic	Output	E100 integer control	E300
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	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	

Table E.1 RPTSCHED output controls (Continued)

Input Keyword or Mnemonic	Output	E100 integer control	E300
OILAPI	Output of grid block oil 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 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 646)	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 timestep. 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

Table E.1 RPTSCHED output controls (Continued)

Input Keyword or Mnemonic	Output	E100 integer control	E300
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
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	

Table E.1 RPTSCHED output controls (Continued)

Input Keyword or Mnemonic	Output	E100 integer control	E300
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	
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 liquid)		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 /
```

RTPSOL: 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	

Table F.1 RPTSOL output controls (Continued)

Input Keyword or Mnemonic	Output	E100 Integer value	E300
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	
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

Table F.1 RPTSOL output controls (Continued)

Input Keyword or Mnemonic	Output	E100 Integer value	E300
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
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

Table F.1 RPTSOL output controls (Continued)

Input Keyword or Mnemonic	Output	E100 Integer value	E300
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	
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	

Table F.1 RPTSOL output controls (Continued)

Input Keyword or Mnemonic	Output	E100 Integer value	E300
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 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	
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

Table F.1 RPTSOL output controls (Continued)

Input Keyword or Mnemonic	Output	E100 Integer value	E300
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	
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

Table F.1 RPTSOL output controls (Continued)

Input Keyword or Mnemonic	Output	E100 Integer value	E300
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 /
```

Bibliography

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

Chang, Y-B, Coats B. K., and Nolen, J. S. *A Compositional Model for CO2 Floods Including Solubility in Water.* *[Ref. 3]*
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. 4]*
SPE 5719, SPEJ *Page 317-326,* Dec 76

Dake, L.P. *Fundamentals of Reservoir Engineering* *[Ref. 5]*
Elsevier, *Page 256,* 1978

Reid, R. C., Prausnitz, J. M. and Sherwood, T. K. *The properties of Liquids and Gases* *[Ref. 6]*
McGraw-Hill, *Page 546,* 1977

Quandalle, P. and Sabathier, J. C. *Typical Features of a Multipurpose Reservoir Simulator* *[Ref. 7]*
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. 8]*
SPE 16010, San Antonio, 1987

Martin, J. J. *Cubic Equations of State-Which?* *[Ref. 9]*
I and EC Fundamentals, Vol. 18, *Page 81,* May 1973

Coats, K. H. *Simulation of Gas Condensate Reservoir Performance* *[Ref. 10]*
SPE paper no. 10512, presented at the Sixth SPE Symposium onReservoir Simulation, New Orleans, 1982

Lorentz, J., Bray, B. G. and Clark, C. R. J. *Calculating Viscosity of Reservoir Fluids from their Composition* *[Ref. 11]*
J.Pet.Tech., 1171, *Page 231,* 1964

Michelsen, M. L. *The isothermal flash problem. Part I. Stability* *[Ref. 12]*
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. 13]*
Prentice-Hall, 1983

Crowe, C. M. and Nishio, M. *Convergence Promotion in the Simulation of Chemical Processes - The General Dominant Eigenvalue Method* *[Ref. 14]*
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. 15]*
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. 16]*
SPE 4274, 1974

Aziz, K. and Settari, A. *Petroleum Reservoir Simulation* *[Ref. 17]*
Applied Science Publishers, London, *Page 398,* 1979

Killough, J. E. *Reservoir Simulation with History-dependent Saturation Functions* *[Ref. 18]*
Trans. AIME 261, *Page 37-48,* 1976

Carlson, F. M. *Simulation of Relative Permeability Hysteresis to the Non-Wetting Phase* *[Ref. 19]*
SPE 10157, San Antonio, 1981

Land, C. E. *Calculation of Imbibition Relative Permeability for Two- and Three-Phase Flow from Rock Properties* *[Ref. 20]*
Soc. Pet. Eng. J., *Page 149-156,* June 1968

Pedrosa, O. A., and Aziz, K. *Use of Hybrid Grid in Reservoir Simulation* *[Ref. 21]*
SPE 13507, 1985

Ewing, R. E. et al. *Efficient Use of Locally Refined Grids for Multiphase Reservoir Simulation* *[Ref. 22]*
SPE 18413, 1989

Todd, M. and Longstaff, W. *The Development, Testing and Application of a Numerical Simulator for Predicting Miscible Flood Performance* *[Ref. 23]*
SPE 3484, 1972

Zuber, N. and Findlay, J. A. *Average Volumetric Concentration in Two-Phase Flow Systems* *[Ref. 24]*
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. 25]*
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. 26]*
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. 27]*
SPE Production Engineering, *Page 263-272,* May 1988

Shi, H. et al. *Drift-Flux Modeling of Multiphase Flow in Wellbores* *[Ref. 28]*
SPE 84228, presented at the SPE Annual Technical Conference and Exhibition, Denver, Oct. 2003

Harmathy, T. Z. *Velocity of Large Drops and Bubbles in Media of Infinite and Restricted Extent* *[Ref. 29]*
AIChEJ, Vol. 6, *Page 281,* 1960

Wallis, G. B. and Makkenchery, S. *The Hanging Film Phenomenon in Vertical Annular Two-phase Flow* *[Ref. 30]*
Trans. ASME, Ser. I, Vol. 96, *Page 297,* 1974

Wallis, G. B. *One-dimensional Two-phase Flow* *[Ref. 31]*
McGraw-Hill, 1969

Hasan, A. R. and Kabir, C. S. *A Simplified Model for Oil/Water Flow in Vertical and Deviated Wellbores* *[Ref. 32]*
SPE Production & Facilities, *Page 56-62,* Feb. 1999

Beggs, H. D. *Production Optimization Using Nodal Analysis* *[Ref. 33]*
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. 34]*
J. Pet. Tech., *Page 475-484,* April 1965

Rozon, B. J. *A Generalized Finite Volume Discretization Method for Reservoir Simulation* *[Ref. 35]*
SPE 18414, Proceedings of the Reservoir Simulation Symposium, Houston, Texas, Feb. 1989

Ponting, D. K. *Corner Point Geometry in Reservoir Simulation* *[Ref. 36]*
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. 37]*
Trans. AIME, *Page 573-592*Aug. 1979

Hegre, T. M., Dalen, V. and Henriquez, A. *Generalised Transmissibilities for Distorted Grids in Reservoir Simulation* *[Ref. 38]*
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. 39]*
SPE Reservoir Engineering, Feb. 1991

Stone, H. L. *Probability Model for Estimating Three-Phase Relative Permeability* *[Ref. 40]*
Trans AIME (JPT), 249, *Page 214-218,* 1970

Stone, H. L. *Estimation of Three-Phase Relative Permeability and Residual Oil Data,* *[Ref. 41]*
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. 42]*
SPE 5729, 4th SPE Symposium on Reservoir Simulation, Los Angeles, 1976

Appleyard, J. and Cheshire, I. M. *Nested Factorization* *[Ref. 43]*
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. 44]*
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. 45]*
SPE 12259, 7th SPE Symposium on Reservoir Simulation, San Francisco, 1983

Ponting, D. K., et al. *An Efficient Fully Implicit Simulator* *[Ref. 46]*
Soc. Pet. Eng. J., 23, *Page 544,* 1983

Cheshire, I. M. and Pollard, R. *Recent Advances in Nested Factorization* *[Ref. 47]*
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. 48]*
SPE 13536, 1985

Saad, Y. *Iterative Methods for Sparse Linear Systems* *[Ref. 49]*
PWS Publishing Company, Boston, 1996

Coats, K. H. *An Equation of State Compositional Model* *[Ref. 50]*
SPE 8284, SPEJ, *Page 363,* Oct. 1980

Bardon, C. and Longeron, D. G. *Influence of Very Low Interfacial Tensions on Relative Permeabilities* *[Ref. 51]*
SPE 7609, SPEJ, Oct. 1980

Coats, K. H. *In-Situ Combustion Model* *[Ref. 52]*
SPE Journal, *Page 533-554,* December 1980

Perry, R. H. and Green, D.W. *Perry's Chemical Engineers' Handbook* *[Ref. 53]*
McGraw Hill, Sixth Edition, 1984

Reid, R.C., Prausnitz, J. M. and Polling, B. E. *The Properties of Gases and Liquids* *[Ref. 54]*
McGraw-Hill, 1987

Prats, M. *Thermal Recovery* *[Ref. 55]*
SPE Monograph 7, *1986*

Crookston, H. B., Culham, W. E. and Chen, W. H. *Numerical Simulation Model for Thermal Recovery Processes* *[Ref. 56]*
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. 57]*
Gulf Publishing Company, Second Edition, 1984

Wingard, J. S. *Multicomponent Multiphase Flow in Porous Media with Temperature Variation* *[Ref. 58]*
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. 59]*
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. 60]*
SPEJ, 11, *Page 471-4741989*

Grabowski, J. W. and Rubin, B. *A Preliminary Numerical Simulation Study of In-situ Combustion in a Cold Lake Oil Sands Reservoir* *[Ref. 61]*
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. 62]*
J. Pet. Tech. 39, 12, *Page 1585-15901987*

Haywood, R. W. *Thermodynamic Tables in SI (Metric) Units* *[Ref. 63]*
Cambridge University Press, Third Edition, 1990

Farouq Ali, S. M. *Oil Recovery by Steam Injection* *[Ref. 64]*
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. 65]*
Journal of Petroleum Technology, *Page 1576-1584,* December 1987

Wilson, G. *A Modified Redlich-Kwong EOS, Application to General Physical Data Calculations* *[Ref. 66]*
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. 67]*
SPE 21222, 11th SPE Symposium on Reservoir Simulation, Anaheim, 1991

Kenyon, D. E. and Behie, A. *Third SPE Comparative Solution Project: Gas Cycling of Retrograde Condensate Reservoirs* *[Ref. 68]*
SPE 12278, 7th SPE Symposium on Reservoir Simulation, San Francisco, 1983

Haaland, S.E. *Journal of Fluid Engineering, volume 105* *[Ref. 69]*
Trans. ASME, *Page* 89, 1983

Welty, J.R., Wicks, C.E., and Wilson, R.E. *Fundamentals of Momentum Heat and Mass Transfer* *[Ref. 70]*
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. 71]*
SPE 21221, 11th SPE Symposium on Reservoir Simulation, Anaheim, 1991

Whitson, C. H. and Fevang, Ø. *Generalised Pseudopressure Well Treatment in Reservoir Simulation* *[Ref. 72]*
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. 73]*
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. 74]*
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. 75]*
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. 76]*
The Journal of Canadian Petroleum Technology, (Montreal), July-September 1980, *Page 87-90*

Mott, R. *Calculating Well Deliverability in Gas Condensate Reservoirs* *[Ref. 77]*
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. 78]*
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. 79]*
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. 80]*
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. 81]*
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. 82]*
SPE 78131, SPE Reservoir Evaluation & Engineering, *Page 206-218,* June 2002

Hibbitt, Karlsson & Sorensen, Inc. *ABAQUS Theory Manual* *[Ref. 83]*
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. 84]*
SPE/ISRM Eurock '98, Trondheim, Norway, 1998

Stone, T., Bowen, G., Papanastasiou, P., Fuller, J. *Fully Coupled Geomechanics in a Commercial Reservoir Simulator* *[Ref. 85]*
SPE European Petroleum Conference, Paris, France, 23-24 October, 2000

Coats, K.H. *IMPES Stability: The Stable Step* *[Ref. 86]*
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. 87]*
Butterworth-Heinemann, Oxford, 1988

Mavko, G., Mukerji T. and Dvorkin J. *The Rock Physics Handbook: Tools for Seismic Analysis in Porous Media* *[Ref. 88]*
Cambridge University Press, 1998

Brie A., Pampuri F.,Marsala A.F. and Meazza O. *Shear Sonic Interpretation in Gas-Bearing Sands* *[Ref. 89]*
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. 90]*
Journal of Petroleum Science and Engineering 1064, *Page 1-16,* 2003

Zimmerman, R.W. *Compressibility of Sandstones* *[Ref. 91]*
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. 92]*
6th European Conference on the Mathematics of Oil Recovery, Leoben, September 1996

Gill,P.E., Murray, W. and Wright, M.H. *Practical Optimization* *[Ref. 93]*
Academic Press, 1986

Sokolnikoff, I.S. and Redheffer, R.M. *Mathematics of Physics and Modern Engineering* *[Ref. 94]*
McGraw-Hill,1967

Stone, T.W., Xian, C., Fang, Z., Manalac, E., Marsden, R., Fuller, J., *Coupled Geomechanical Simulation of Stress Dependent Reservoirs,* *[Ref. 95]*
SPE 79697, SPE Reservoir Simulation Symposium, Houston, Texas, USA, 3-5 February, 2003

Stüben, K. *SAMG USER's Manual Release 21b1* *[Ref. 96]*
Fraunhofer Institute SCAI, Schloss Birlinghoven, D-53754 St. Augustin, Germany, July 2002

Index

A

Adaptive Implicit Method 26

Adsorption
- and COALPP 87
- and environmental tracer169, 171
- Competitive, CO2 into coal . . 86
- Foam. 217
- Gas re-adsorption into coal. . . 85
- on coal surface 84
- Polymer 611
- Surfactant 800

AIM Method 238

ALQ
- in VFP table. 545

ALQ Values
- Changing. 389

Amalgamation
- Cartesian LGRs. 438, 442
- Cell 434

API Tracking 30, 51

Aqueous Phase 67

Aquifer. 36
- Flux Boundary Conditions . . 212

Aquifer Modeling 55
- Carter-Tracy 58
- Constant Flux 64
- Fetkovich. 61
- Introduction 55
- Numerical 56

Artificial Lift 389, 1009

Artificial Lift Quantity
- and gas injection 277
- in network branch. 284
- in network flow lines 542
- in outlet branch . . . . . .564 to 565
- in VFP table 545
- in VFP tables 282
- Multi-level Compression . . . 554
- Optimization 282

Automatic
- Chokes 555, 561
- Compressors . 260, 265, 554, 560
- Refinement 466
  - Restrictions. 468
- Workover 453, 526, 949

Automatic Opening 385, 992

B

Balancing, Network 545

Block Centre Geometry 25

Brine 40

Brine Tracking 30, 65

C

Calorific Value
- Control . . . . . .302 to 303, 310
- Gas. 301, 309

Calorific Values. 296

Capillary
- Number 797
- Pressure 799

Carbon Dioxide Solution. 67

Cell Amalgamation 434

Chemical Reactions
- Examples 80
- Formulation. 72
- Introduction. 69
- Keywords. 70
- Output 79

Chokes
- Automatic 555, 561

Closing Wells. 392, 989

Coal Bed Methane 43, 83
- Compositional Effects 89
- Dual Porosity. 84
- Example
  - Basic Test 94
  - CO_2 Injection. 99
- Examples 94
- Introduction. 83
- Keywords. 92
- Using 88

COALADS 86

Cold Water Tracking 814

Combustion. 863

Co-mingling 34

Compatibility 107

Compositional Effects 86

Compressed Vertical Equilibrium 938
Compressors 41
Activating 391
Automatic 260, 265, 554, 560
Network. 542, 545
Connection
Factor 966
Non-Neighbor 27
Opening. 385, 992
Well. 964, 985, 990
Constant Flux Aquifers 64
Construction, Network 544
Control
Injection 33
Output . . .1022, 1027, 1032, 1037, 1048
Production. 32
Sales Gas 34
Well. 31
Convergence Criteria. 241
Convergence Reports. 113
Corner Point Geometry. 25
Crossflow. 34, 986, 990

D

Daily Contracted Quantity - See DCQ
Data Requirements 422
DCQ 41
Adjustment 254
Calculating 257
Decay . . . . . 170 to 171
of foam. 217
Delivery Capacity
Estimation 259
Delumping 918
Density
3-component 773
4-component 772
D-Factor. 969
Diagonal Transmissibilities. 568
Diffusion 123
and environmental tracer . . . 170
Coefficient 84
Coefficients for ECLIPSE 100 125
Coefficients for ECLIPSE 300 128
Methane from Coal 84, 86
Miscible Flow 776
with Environmental Tracer option. 171
Diffusivity. 84, 170
Diffusivity Calculations 133
Directional Relative Permeability . 29
Downtime. 992
Drainage Region 212
Drilling Queue. 384, 392, 996
Drilling Wells 384, 992, 996, 1007
Dual Porosity 28, 139, 446
Coal Bed Methane 84
Discretized matrix model. . . 166
Drainage process Sigma factor . 162
Gravity drainage . . 142, 145, 160
Integrated capillary pressure 163
Introduction 139
Keyword summary 153
Linear equation solution . . . 158
Modifying recovery versus time 156
Recovery Mechanisms 142
Relative permeability scaling 157
Restrictions. 152
Sigma factor for drainage. . . 162
Specific Facilities 150
Transfer functions 155
Transmissibility Calculations 140

E

ECLIPSE 100. 23
ECLIPSE Office 25, 48
ECLIPSE Options
Coal Bed Methane 43
Environmental Tracers 42
Flux Boundaries. 40
Foam Model 43
Gas Calorific Value Control . . 43
Gas Field Operations 41
Gas Lift Optimization. 42
GI Pseudo-Compositional Model 42
Local Grid Refinement 40
Multi-segment Wells. 44
Polymer Flood Model. 39
Reservoir Coupling. 43
Shared Memory Parallel. 42
Solvent Model 41
Surface Networks 40
Surfactant Model 42
Thermal Option 44, 826
Wellbore Friction 41
Economic Limit. 311, 453
Efficiency Factor 992
End Point Scaling
LGRs. 447
Energy Control 302, 307, 311
Environmental Tracers. 42, 169
Example 176
Introduction. 169
Keywords. 174
Partitioned Tracers 173
Restrictions 171
Using 171
Equations of State. 185
Hydrocarbon Mixtures 193
Three Parameter 192
Two parameter 185
Equilibration . . . . . 89 to 90, 421
Algorithm 425
Initial conditions. 426
Extended Network 552

F

Fetkovich Aquifers. 61
Figure 758
File Handling 201
File internal format. 203
Introduction. 201
Files
GRAF 205
FILL 25, 37
Fine Grid Equilibration 36
Flash Calculations 189
FLEXlm licenses 670
FloGrid 25, 45
FloViz 47
Flowlines 658
Fluid In Place
Accurate Calculation 427
Reports. 913
Flux Boundary Conditions 207
Boundaries. 40
Introduction. 207
Keywords. 215

LGRs. 213
Limitations 214
No-flow Boundary Conditions . 213
Pressure Boundary Conditions . 213
Using 208
Wells. 211
Flux Region
Multiple. 210
Single 208
Foam. 43, 216
Adsorption 217
Conservation Equation 217
Decay 217
Example. 223
Introduction 216
Keywords 221
Simulation. 217
Using 220
Foamy oil. 863
Formulation of Equations. 232
Convergence Criteria. 241
Flows 244
IMPES and AIM Methods. . . 238
Newton Iteration 249
Reservoir Densities 248
Variable Sets 234
Free Format Input 25
Frictional Pressure Loss 941
FrontSim 49
Fully Implicit Technology. 26

G

Gas
Calorific Value 301, 309
Calorific Value Control 43
Energy Control 302, 307, 311
Field Operations. 41
Lift Optimization 42
Mobility Reduction 218
Rate
Control. 258
Output 258
Sales 258
Supply Contracts 263
Well. 35
Gas Accounting Tables 1002
Gas Calorific Value Control 301
Adjusting Guide Rates 303
Controlling Energy Production Rate 307
Overview 301
Restrictions. 313
Summary of Keywords. 314
Using the Facility. 309
Gas Field Operations 251
Calorific Value Control. 311
Description of Facilities 253
Energy Control. 311, 314
Introduction 251
Keywords 273
Multiple Contract Groups . . 263
Network 267
Reservoir Coupling. 669
Restrictions. 272
Using. 266
Gas Lift
Network 545
Gas Lift Optimization. 276
Group 278
Individual Well 277
Introduction 276
Keywords 286
Network 279, 284
Restrictions. 285
Using. 282
VFP Tables 282
Gas Mode
Single. 88
Two 89
Gas Plants. 287
GASWAT. 291
Geomechanics 315
Geometry
Block Centre. 25
Corner Point. 25
Options 25
GI Pseudo-compositional. 331
Example 337
Introduction 331
Keywords 336
SUMMARY. 336
Using. 335
GI Pseudo-Compositional Model . 42
GLIFTOPT 283
Gradient Option 344
Data File Overview 349
Example 373
Introduction 344
Keywords. 368
Output 355
Technical Description. 364
GRAF 38
GRAF Files. 205
Graphics. 1034
GRID. 25, 45
Grid
Reports. 445
Grid Orientation Effect 568
GridSim 49
Group
Control 995, 1000
Economic Limit. 311
Economic Limits 1000
Efficiency factor 992
Hierarchy. 33, 994
Injection Control. 33
Production Control. 32
Production Rules 283, 383
Group Production Rules
Activating compressors. 391
Changing VFP table 388
Changing well ALQ. 389
Closing wells during operations 392
Drilling new wells 384
Introduction. 383
Opening new connections. . . 385
Reducing THP limit 387
Use with Network Option. . . 393
Groups
Controls 268, 453, 657
Calorific Value. . .302 to 303, 310
Energy 302, 307, 311
Network. 540
Multiple Contract. 265
Defining 263
Simulating 263
Production Rate Limits 283
Sub-sea Completion Manifolds . 541, 548
Guide Rates
Adjusting. 303
Calculating. 310

H

Hierachy. 33

Highly Deviated Well 35
Horizontal Well .35, 41, 440, 495, 941, 950, 967, 983
Hydrocarbon Mixtures
Equations Of State 193
Hydrostatic Equilibration
Basic 425
Hysteresis 29, 394
Capillary pressure 404
Introduction 394
Modeling oil wet systems . . . 403
Non-wetting phase 396
Using the option. 395
Wetting phase 401
Hysteresis in WAG Floods 408
Introduction 408
Non-wetting phase 410
Residual oil modification . . . 413
Using the option. 409
Wetting phase model. 414

I

IMPES Method 26, 232, 238
Independent Reservoir Regions. . 416
Examples 419
Introduction 416
Merging multiple corner point grids 418
Initial Conditions 421
Injection
Control. 33
Network. 542
Solvent 90, 780
In-place LGRs 451
Introduction 23
Features 25
Overview. 23

J

Jacobian matrix. . . . . 26 to 27

K

Keywords
Network 564

L

Langmuir Isotherm . . . . . .84, 86 to 87
Licenses. 50
Lift
Applying. 389
LIFTOPT. 282
Local Grid Coarsening 434, 462
Implementation 462
Introduction 434
Keywords 469
Upscaling 463
Local Grid Refinement 40, 434
Automatic. 466
Block Sizes 440
Cartesian. 439
Editing. 444
End Point Scaling 447
Example 472
Flux Boundary Conditions. . 213
Grid Data 456
Introduction 434
Keywords 469
Pore Volume. 456
Radial 435, 457
Restrictions. 452, 454
Rock Properties 442
Switching Off 447
Switching On 447
Transmissibility 456
Unstructured 446
Well 438, 453
Lumping. 918

M

Manifold Groups
Network 541, 548
Manifold Pressure 40
Master Reservoir 654, 663
Maximum Decrease Factors. 120
Minimum Step Codes. 120
Miscible. 31
Components. 765
Flow
Numerical Diffusion . . . 776
Miscible Flood Modeling. 479
Component density 484
Control with pressure 486
Effect of water saturation . . .481
Introduction. 479
Numerical diffusion 488
Relative permeability. 482
Todd-Longstaff Mixing Parameter 480
Using 490
Viscosity. 483
Mixing Parameter. 765, 774
Multi-lateral Well . 495, 497, 941, 944, 954
Multi-phase
Introduction. 492
Multi-segment Wells 44, 47, 495
Converting Friction Wells . . .531
Down-hole separators 515
Drift Flux Slip Model 506
Introduction. 315, 495
Keywords. 323, 533
Restrictions 330, 530
The Multi-segment Well Model . 317, 497
Thermal 520
Using 325, 524
VFP Tables 503

N

Nested Factorization 26
Network. 538
Balancing 545
Chokes 555, 561
Compressors . .542, 545, 554, 560
Construction 544
ECLIPSE 300 562
Efficiency Factors 557
Extended 552
Gas Field Operations 267
Gas Lift. 545
Gas Lift Optimization 284
Group Production Control . .540
Injection 542
Introduction. 538
Keywords. 564
Pumps 542, 545
Reservoir Coupling 658
Standard. 544
Structure. 552

Using 544, 558
NGLs 287
Nine-point Schemes. 567
NNC. 27
Non-Darcy Flow. 571
Non-Neighbor Connection. 27
Numbering Convention 458
Numerical Aquifers. 56

O

Oil Saturation Properties. 702
Open-ECLIPSE 45
Opening New Connections 385
Output Control1022, 1027, 1032, 1037, 1048

P

Parallel Option 581
Activating 582
Efficiency. 584
Keywords 592
LGRs. 588
Memory Scaling 591
Partitioning 583
Performance 584, 589
Pedersen Method, use of. 190
Permeability
Reductions. 611
Petro-elastic Model 593
Phase Options. 25
Phase States. 195
Pipeline 40
PlanOpt 49
Polymer Adsorption 611
Polymer Flood 39, 607
Example
Problem 1. 618
Problem 2. 623
Fluid Viscosity 610
Introduction 607
Keywords 615
Polymer Adsorption 611
Processes 607
Simulation 608
Using 614
Pore Volume
Dead 611
LGRs 456
Potential Calculations. 628
Potentials
Initial Contact Corrected . . . 629
Well . . . . . .384, 631, 995 to 996
Pressure
Boundary Conditions 213
Capillary 799
Dependence 775
Frictional Loss 941
Manifold 40
Pressure Maintenance. 1006
Production
Control 32, 34
Productivity Index 983, 987
PSEUDO 38
Pump Performance 40
Pumps
Network 542, 545
PVM. 662
PVT
Tables. 28
PVTi. 46

R

Recovery Mechanisms 142, 646
Details of calculation. 651
Introduction 646
Solution gas drive 648
Summary output 650
Water drive. 649
Recovery Plant. 289
Reducing The THP Limit 387
Relative Permeability 767
Directional 29
Near Critical Gas. 713
Near Critical Oil 713
Surfactant 798
Velocity Dependent. 576
Relative permeability
Miscible flood modeling. . . . 482
Reperforating. 385
Reports
Grid. 445
Timing 445, 584
Reservoir Coupling 43, 653
Files. 659, 671
Gas Field Operations 669
Introduction. 653
Keywords. 676
Licenses 670
Restrictions 660
Surface Networks. 658
Synchronization 656
Using 662
Restart 1043, 1053
Restart Runs 212
Restarts. 678
ECLIPSE 100 679
ECLIPSE 300 685
Introduction. 678
Restrictions
Automatic Refinement. 468
Environmental Tracers. 171
Flux Boundary Conditions . . 214
Gas Calorific Value Control . 313
Gas Field Operations 272
Gas Lift Optimization 285
Local Grid Refinement. . 452, 454
Multi-segment Wells . . . 330, 530
Reservoir Coupling 660
Retubing. 388
Rock
Compaction. 30
Data. 28
Rock Compaction 687
Introduction. 593, 687
Tabulated compaction curves688
Water induced compaction . . 692
`RPTGRID` 1027
`RPTPROPS` 1022
`RPTRST`. 1032
`RPTSCHED` 1037
`RPTSOL`. 1048
Run-time dimensioning. 27

S

Sales Gas 1004
Sales Gas Control 34
Sales Gas Rate

Control 258
Output 258
Salinity 297
Saturation
Table Scaling 29
Saturation Functions 693
Consistency requirements. . . 711
Gas saturation properties . . . 697
Introduction 693
Near Critical Relative Permeabilities 713
Oil saturation properties. . . . 699
Table end points 710
Three phase oil relative permeability models 702
Water saturation properties . 695
Saturation Table Scaling 716
Example. 731
Introduction 716
Miscellaneous points 723
Scaling of capillary pressure functions 718
Scaling of relative permeability functions 719
Special applications 725
SaturationTable Scaling
Consistency requirements. . . 730
SCAL 46
Scale Deposition. 987
Schedule 47
Seasonality Profile Generation. . . 253
Separators
Down-hole. 515
Shared Memory Parallel 42
Shear
Thinning 612
SimOpt. 49
Single Gas Mode 88
Skin Factor. 969
Slave Reservoir 654, 665
Solid phase 842
Introduction 733
Solution Change Target. 118
Solution of the Linear Equations . 744
Convergence Criteria. 752
GMRES Method 761
Initialisation 748
Material balance 745
Nested Factorization 746
Orthomin 751
Solution procedure 749
Solution Procedure for LGRs 760
Treatment of wells. 754
Solvent 41, 763
Applications. 763
Example 783
Initiating 778
Injection 90, 780
Introduction 763
Keywords 781
Numerical Diffusion 776
Pressure Dependence 775
PVT 771, 778
Relative Permeability . . 767, 779
Todd & Longstaff. 765
Using 778
Standard Network. 544
Surface
Networks 40
Surface Tensions
Effects 790
Evaluation 790
Use. 791
Use of MISCNUM 795
Surfactant 42, 796
Adsorption 800
Capillary Number 797
Capillary Pressure 799
Conservation Equation. 797
Example 806
Introduction 796
Keywords 804
Relative Permeability 798
Simulation 797
Using 802
Swing Factor 254

T

Temperature 31
Temperature Option 814
Tensor Permeability 819
Discretization 821
Introduction 819
Thermal Option 44, 826
Converting Data 877
Features. 867
Formulation 836
Introduction. 826
Keywords. 828
Properties. 845
Selected bibliography. 884
THP Limit
Reducing 387
Throughput Target 119
Time Truncation Error 119
Timesteps. 450
Timing Reports 445, 584, 886
Todd & Longstaff 765
Todd and Longstaff 41
Todd-Longstaff Mixing Parameter480
Total Compressibility Checks 889
Tracer . . . . . 170 to 171
Tracking 30
Tracer Tracking 907
Control of numerical diffusion. . 909
Delumping. 918
Introduction. 907
Tracking original fluids 912
Using the facility. 910
Tracers
Environmental 169
Gas Calorific Value. 309
Tracking
API 30, 51
Brine 30, 65
Cold water 814
Tracer 30, 907
Tracer Concentration 309
Tracer Name 309
Transmissibility
Calculations. 449
LGRs. 456
Transmissibility Calculations 893
Block Center 895
Cartesian 895
Corner Point 898
Dual Porosity. 140
HALFTRAN 899
Introduction. 893
OLDTRANR 896
Radial 901
Variations. 904
Tuning 445
Two Gas Mode 89

U

Units 296, 921
Constants 922
Conventions 921
Conversion factors 923
Unstructured LGRs 446

V

Velocity Dependent Relative Permeabilities 576
Vertical Equilibrium 28, 445, 925
Cell saturation distribution . 928
Compressed 938
Introduction 925
Miscellaneous points 936
Relative permeabilities and capillary pressures 933
Residual flow model 934
Using the VE option 927
Vertical Flow Performance 1009
VFP Table Numbers
Changing 388
VFP Tables 53, 1009
VFPi 37
Viscosity
3-phase 773
4-phase 771
Fluid 610
Viscosity Evaluation 190

W

Water Induced Compaction 692
Weighted Incremental Gradient . . . . 278 to 279
Well
Closing 392, 989
Co-mingling 34
Completion 385, 990
Coning 991
Connection . . . 385, 985, 990, 992
Control 31, 988
Crossflow 34, 986, 990
Drilling . 384, 392, 992, 996, 1007
Economic Limit 989
Efficiency factor 992
Flux Boundary Conditions . . .211
Gas 35
Highly Deviated 35
Horizontal . 35, 41, 440, 495, 941, 950, 967, 983
Limit 988
Multi-lateral . . 495, 497, 941, 944, 954
Multi-segment 495, 497
Opening 992
Potential . . . .384, 631, 995 to 996
Productivity Index 983
Treatment in Local Grids . . . 453
Workover . . 385, 392, 989 to 990, 997, 1000, 1007
Well Inflow Performance
Connection transmissibility factor 966
Flow-dependent skin factor . 969
Injecting connections 980
Introduction 964
Phase mobilities 972
Productivity index 983
Special inflow equations 973
Well bore head term 982
Well Modeling Facilities 985
Group and Field Controls . . . 994
Vertical Flow Performance . 1009
Well Completions 985
Well Controls and Limits 988
Wellbore
Friction 41
Wellbore Friction 531, 941
Example 957
Graphical Presentation 950
Introduction 941
Keywords 956
Multi-lateral Wells 944
Using 944
Workovers 949
Wells
Multi-segment 44, 47
Potentials 631
WELTEST 100 48
WELTEST 200 48
`WLIFTOPT` 282
Workover . . .385, 392, 989 to 990, 997, 1000, 1007
Automatic 453, 526, 949